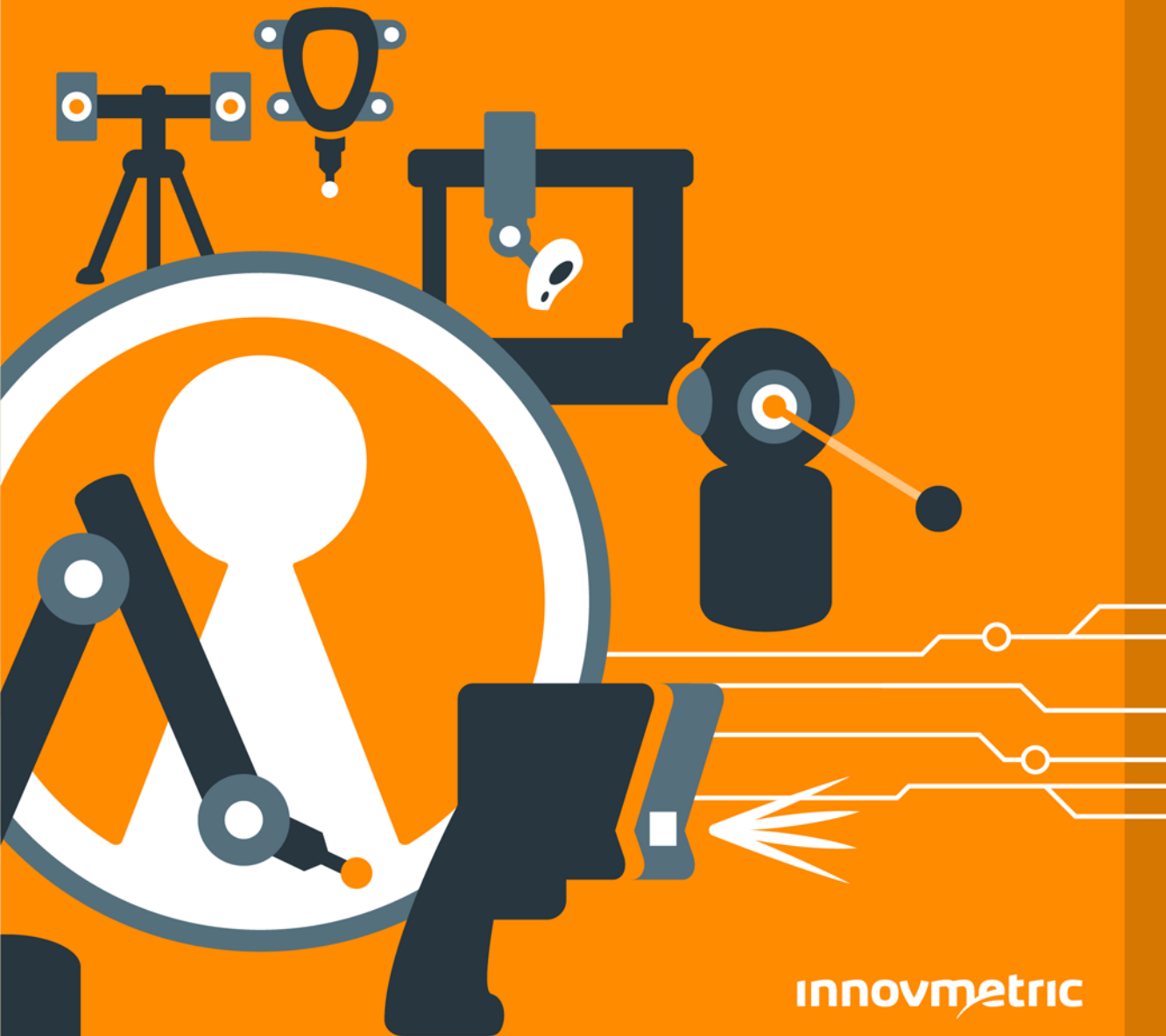


PolyWorks | Inspector™ Essentials

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Essentials – PolyWorks | Inspector™ Premium and Standard Packages

PolyWorks® Metrology Suite 2020

[2020/07/02] – TEC-0037-2020-EN-3.0

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PolyWorks Inspector[™]

PREMIUM AND STANDARD PACKAGES

PolyWorks | Inspector is a powerful industrial 3D metrology software solution that uses noncontact point cloud digitizers and single-point contact-based probing devices to acquire and control tool and part dimensions, diagnose and prevent manufacturing and assembly issues, guide assembly building through real-time measurements, and oversee the quality of assembled products.



Introduction



Introducing the Essentials

About

The *Essentials* guide is intended to be a companion for new users post-training and throughout their first year of using the PolyWorks Metrology Suite.

Approach

The *Essentials* content is presented following a workflow-based approach. The document is divided into sections according to specific workflow steps.

Each section is then organized using a tool-based approach, where each tool pertaining to the subject is documented following the established presentation layout (recipe).

Generally, the *Essentials* has been documented using the default configuration and visual layout with the exception of the background color of the 3D scene and the font size for certain annotations. Also, in most cases, only the basic parameters are described.


Document conventions

Navigation through the sections

- As mentioned previously, this document has been designed according to the inspection workflow recommended in the PolyWorks Metrology Suite (see [The inspection workflow](#) on page 31). The structure of the document is clearly identified in the header of each page using the icon and description corresponding to the workflow step.
- The Table of contents, as well as the first page of each section on measurement objects (data color maps, features, comparison points, cross-sections, and gauges), offer hyperlinks to the listed subjects.

A menu-based approach

- In the PolyWorks Metrology Suite, most of the functionalities offered on the menus are also available through toolbars. In this document, the menu paths are documented in the procedure. The corresponding toolbar buttons are shown to the right, as shown in the example below.

1. Choose <i>Align > Feature-Based > Plane, Axis, Center Point</i> .	
--	--

Key parameters

- Key parameters, used for the creation of measurement objects, are explained in the introduction of the section pertaining to the object type, to avoid redundancy. For example, when creating a feature, it is specified to enter a maximum distance. For information on this parameter, see the introduction to features.

Text formatting

The following text formatting conventions are used throughout the *Essentials*:

Objective and Required:

For each procedure, the objective is stated and the requirements are listed.

Sections:

Each procedure is organized using the following sections:

- **Steps:** Present the steps to follow for each tool.
- **Results:** Present the results in the Tree View (to the left) and the 3D scene (to the right). For simplification purposes, the in-tolerance status of measurement objects is only shown in Tree View images when relevant.
- **Notes:** Present useful additional information on the current tool.
- **Tips:** Present hints on how to use the current tool in a specific context.

Graphical user interface elements:

- Menu paths are presented in gray and italics, and chevrons are used to show the sequence.

For example: *Align > Best-Fit Data Objects > Data to Reference Objects*

- Dialog boxes are capitalized.

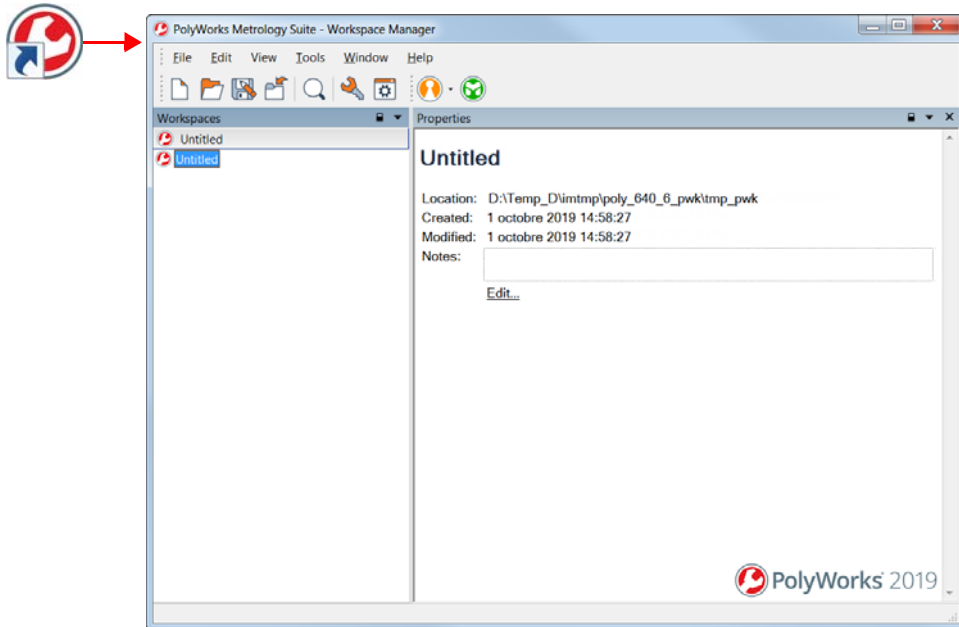
For example: The Best-Fit Data to Reference Objects dialog box opens.

- Parameters are presented in bold.

For example: In the **Method** list, click **Fit to Reference Object Surfaces**.

Introducing the Workspace Manager

When starting the PolyWorks Metrology Suite, the Workspace Manager interface displays.



The Workspace Manager manages a PolyWorks process from beginning to end. This includes the following tasks:

- Adding software license keys.
- Managing all the incoming data files and the results produced by the PolyWorks Metrology Suite modules. As work progresses, files and projects are saved to a container called a “workspace”.
- Offering simple navigation between modules.

Adding license keys

License keys are required to use the PolyWorks Metrology Suite. There are two types of licenses: node-locked or floating. Both types are managed through the PolyWorks Metrology Suite License Manager, which is available by choosing *Tools > PolyWorks License Manager* in the Workspace Manager.



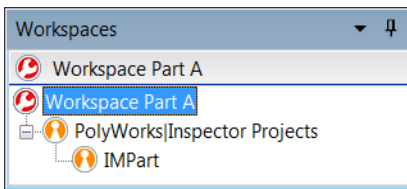
For information on how to add license keys, choose *Help > Reference Guides > License Keys*.

Introducing the workspace format

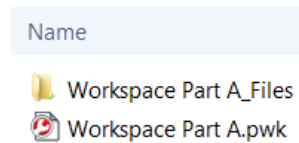
The Workspace Manager uses a workspace format which is designed to handle all the results of a point cloud processing project.

Here is a view of a workspace in the Workspace Manager.

When a workspace is saved, it creates one XML file with the *.pwk* file and one folder with the *_Files* suffix. Both file and folder include the workspace name. The *.pwk* file is only an index – the folder contains all the data.



=



The .pwk file and associated _Files folder must remain in the same folder.

Starting the PolyWorks Metrology Suite modules

The Workspace Manager provides direct access to the PolyWorks Metrology Suite modules. The main modules are accessible through the Main Modules toolbar. Click a module button to launch the corresponding module. The present document covers the PolyWorks|Inspector module.

Note that the modules offered depend on the PolyWorks package used. All modules are described below.



PolyWorks|Inspector

PolyWorks|Inspector

This module is a software tool that allows performing data alignments to Reference objects, measuring the deviations of Data point clouds and polygonal models to Reference surfaces, measuring the dimensions of specific features, and generating comparison and verification reports. Data acquisition can be performed in real time by probing and scanning the part.

PolyWorks | Inspector Probing

Accessible through the down arrow of PolyWorks | Inspector, this module is a software tool that allows performing data alignments to Reference objects, measuring the dimensions of specific features, and generating comparison and verification reports. Data acquisition can be performed in real time by probing the part.

PolyWorks | Inspector for Modeler

Accessible through the down arrow of PolyWorks | Inspector, this module offers tools for data acquisition, data alignments, coordinate systems, and feature creation.



PolyWorks | Modeler

PolyWorks | Modeler

This module offers a comprehensive set of advanced editing tools for polygonal models where NURBS patches can be created to conceive NURBS models or repair polygonal models. PolyWorks | Modeler also allows creating parametric sketches, which can be imported into CAD modeling software.

PolyWorks | Modeler Light

This module is available with the PolyWorks | Inspector Premium package only. It offers editing tools for polygonal models, such as tools for reducing models, verifying their watertightness, or improving their appearance.

Utility modules

The complete list of modules is offered in *Tools* and in *Tools > Utility Modules*:



PolyWorks | Reviewer

This module allows viewing inspection projects and polygonal models.

IMAlign



This module is a software tool that accepts point clouds from all digitizers, and permits capturing point clouds in real time, and allows aligning these point clouds using the shape of the points clouds and, if desired, reference points.

IMMerge



This module allows automatically merging a set of aligned 3D scans of an object into a high-resolution polygonal model.

Setting essential project options

Certain default values and parameters used in the PolyWorks Metrology Suite can be set in the Workspace Manager. The following procedure presents the options that are most frequently used. For information on the other options, see the PolyWorks Reference Guide available from the *Help* menu.

Objective: Set the units of length, the language, the plug-in(s) available in the module, and enable the Talisman functionality.

Required: The Workspace Manager is open.

Steps

To set project options, choose *Tools > Options*.



The **Workspace Manager Options** dialog box opens.

Note that changes made to the options do not affect an open module; they apply the next time the module is launched.

Units of length

On the **General** page:

1. Select the units of length that will be used by default for the PolyWorks Metrology Suite modules.
2. Click **Apply**.

Regardless of what units of length are set in the Workspace Manager options, it is always possible to change the units within a module after opening it.

Language

On the **Display** page:

1. Select the language for the user interface.
2. Click **Apply**.

Plug-ins

The PolyWorks Metrology Suite offers a wide choice of plug-ins that allow the capture of 3D coordinate points by way of line scanning and probing devices.

To use one or more plug-ins, the appropriate option must be selected. Only selected plug-ins will be loaded when PolyWorks|Inspector is launched.

On the **Plug-ins** page:

1. Select one or more plug-ins.
2. Click **Apply**.

*For information on a plug-in, see the custom document specific to this plug-in, which is available from the **Help** menu of the application.*

Talisman functionality

PolyWorks|Talisman™ is a remote control application developed to allow controlling a probing/scanning session using a mobile device (such as Apple® iPod touch®, iPhone®, and iPad™ as well as Android mobile devices running Android 4.1 or higher).

To use the Talisman functionality, the appropriate option must be selected.

On the **Talisman** page:

1. Select **PolyWorks server for Talisman**.
2. The change applies to all open modules.

*For information on PolyWorks|Talisman, see the **Talisman Android or Apple Mobile Device User Guides and the Talisman Network Security and Setup Guide**, which are available from the **Help > Reference Guides > PolyWorks|Talisman menu** of the application.*

PolyWorks server for Talisman
 Network connection:
 (Automatic)
 Custom network settings:
 Server static port: 28081
 Server dynamic ports: 1025-65535

3. Click **Apply**.

Notes

If the Bonjour™ networking technology, a trademark of Apple Inc., is not installed on the workstation, the user will be prompted to install it.

PolyWorks | Inspector™

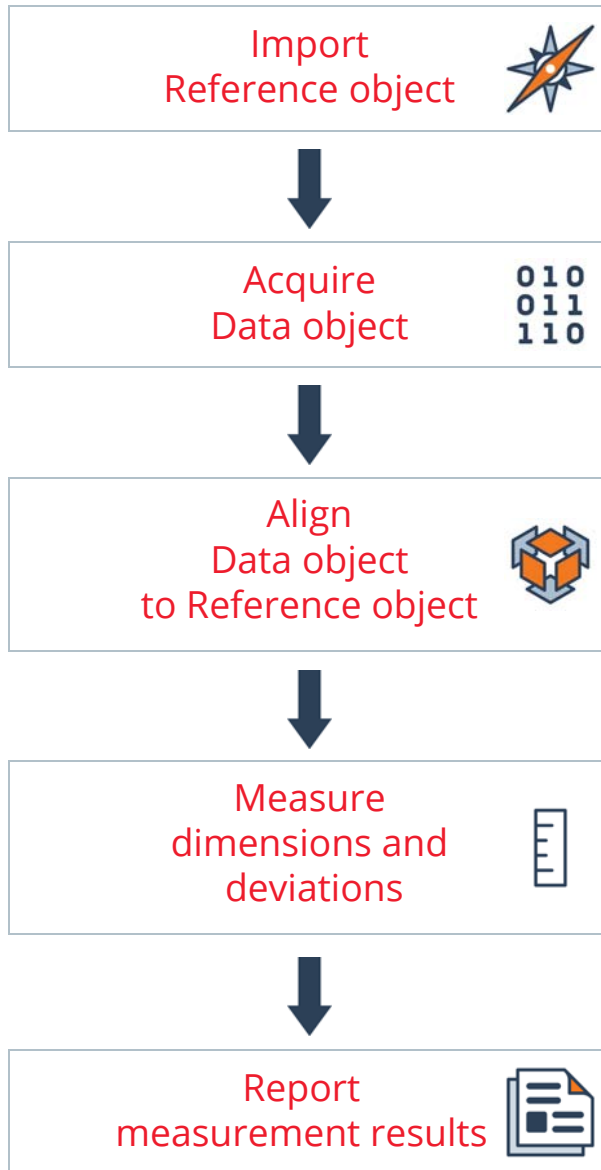
The PolyWorks | Inspector module is a software tool that allows measuring the deviations of Data objects to Reference surfaces, measuring the dimensions of specific features, and generating comparison and verification reports.

PolyWorks | Inspector offers an object-oriented approach to comparison and verification tasks. A global object called the PolyWorks | Inspector project encloses all other objects. Inspection projects describe the list of objects imported and created within PolyWorks | Inspector, and preserve editing operations performed by users. Within a PolyWorks | Inspector project, several categories of objects can be generated.



The inspection workflow

A basic inspection workflow is illustrated below.

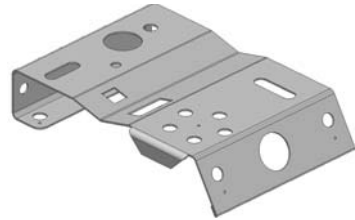


Understanding the workflow



Import Reference object

Import the nominal information of the part (CAD model) into the inspection project.



Acquire Data object

Acquire or import the data of the actual part to inspect, as a point cloud or a polygonal model.



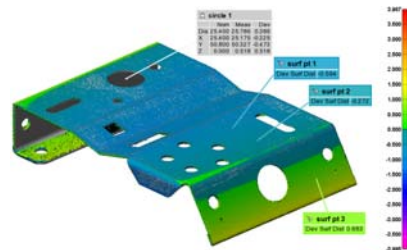
Align Data object to Reference object

Move the data close to the CAD model. This allows measuring deviations of the Data object to the Reference object, and to take advantage of powerful tools such as measurement extraction and visual guidance.



Measure dimensions and deviations

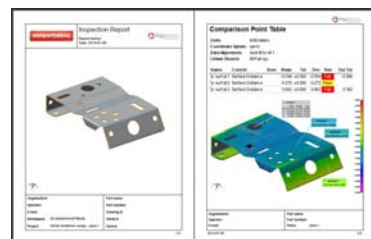
Perform the inspection of the part. Measure the deviations between the part and the nominal, in form, dimension, and position.



Report measurement results

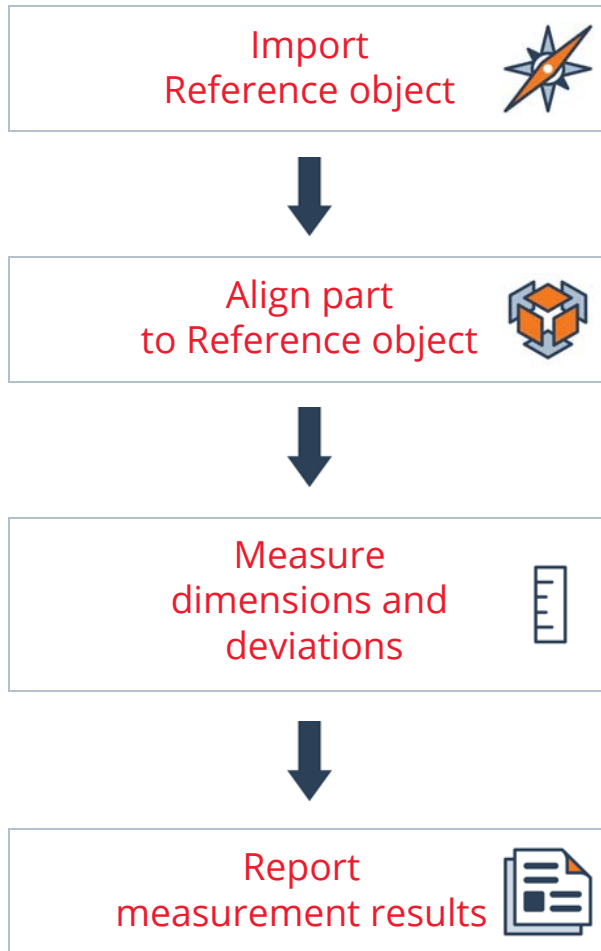
Report the inspection measurement results using:

- Control views in the Control Reviewer;
- Tables and snapshots in formatted reports.



The probing workflow with a CAD model

Here is the basic inspection workflow when using only a probing device to perform the inspection and a CAD model of the inspected part is available.

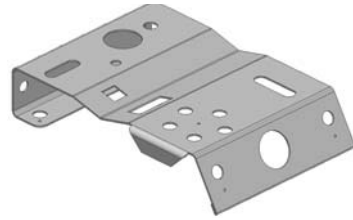


Understanding the workflow



Import Reference object

Import the nominal information of the part (CAD model) into the inspection project.

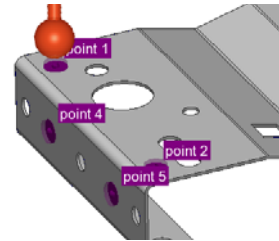


Align part to Reference object



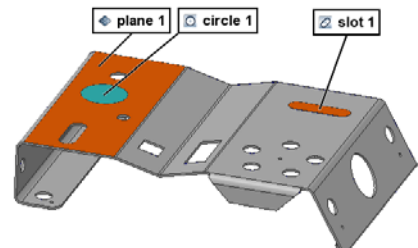
Prealign part to Reference object

Prealign the part to the CAD model to take advantage of the visual guidance tools.



Refine part alignment to Reference object

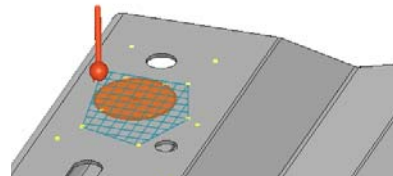
Refine the part alignment to the CAD model using feature-based alignments, reference targets, or datum reference frames.



Measure dimensions and deviations



Perform the inspection of the part by probing point clouds and/or measurement objects such as features and comparison points.





Report measurement results

Report the inspection measurement results using:

- Control views in the Control Reviewer;
- Tables and snapshots in formatted reports.

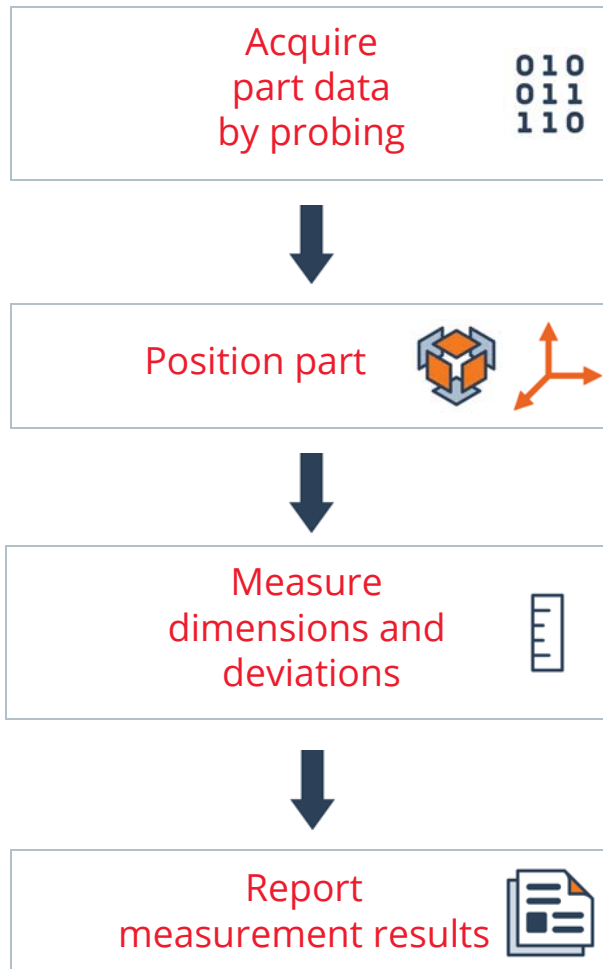
The screenshot displays the PolyWorks software interface. At the top, there are three data tables for different views: 'View 1', 'View 2', and 'View 3'. Each table lists various measurement points with their coordinates and values. Below these is a 3D model of a metal part with orange measurement points. The 'Control View' section contains a table with the following data:

Char No	Meas Name	Unit	Min	Max	Tol	Stat	Stat %
01	width 1	mm	20.000	20.001	±0.001	Pass	
02	width 2	mm	20.000	20.000	±0.000	Pass	
03	width 3	mm	16.000	16.001	±0.001	Pass	
04	width 4	mm	16.000	16.000	±0.000	Pass	
05	width 5	mm	16.000	16.000	±0.000	Pass	
06	width 6	mm	16.000	16.000	±0.000	Pass	
07	width 7	mm	16.000	16.000	±0.000	Pass	
08	width 8	mm	16.000	16.000	±0.000	Pass	
09	width 9	mm	16.000	16.000	±0.000	Pass	
10	width 10	mm	16.000	16.000	±0.000	Pass	
11	width 11	mm	16.000	16.000	±0.000	Pass	
12	width 12	mm	16.000	16.000	±0.000	Pass	
13	width 13	mm	16.000	16.000	±0.000	Pass	
14	width 14	mm	16.000	16.000	±0.000	Pass	
15	width 15	mm	16.000	16.000	±0.000	Pass	
16	width 16	mm	16.000	16.000	±0.000	Pass	
17	width 17	mm	16.000	16.000	±0.000	Pass	
18	width 18	mm	16.000	16.000	±0.000	Pass	
19	width 19	mm	16.000	16.000	±0.000	Pass	
20	width 20	mm	16.000	16.000	±0.000	Pass	

Below the table, there are fields for 'Organization', 'Operator', 'E-mail', 'Part name', 'Part number', and 'Piece'. The date '2018-02-09' is visible at the bottom left.

The probing workflow without a CAD model

Here is the basic inspection workflow when using only a probing device to perform the inspection and no CAD model of the inspected part is available.

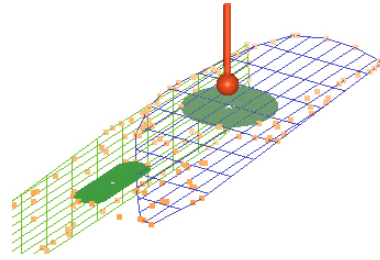


Understanding the workflow

010
011
110

Acquire part data by probing

Acquire part data by probing point clouds and/or measurement objects such as features and comparison points on the actual part.



Position part

Align part to standard axes



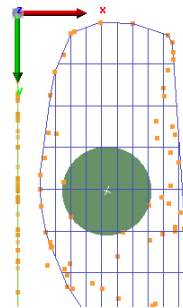
Align the probed data points to standard axes (X, Y, Z) to facilitate reporting. Note that this step is optional.

OR

Create coordinate systems



Create a coordinate system using the probed features to orient and locate the probed data points. This facilitates reporting. Note that this step is optional.



Measure dimensions and deviations



Complete the part measurement. Enter object control nominal values and tolerances, if they are known, into the Geometry Controls pane.

circle 1				
	Nom	Meas	Dev	Test
Dia	25.400	25.381	0.019	Pass
X	25.400	25.381	0.019	Pass
Y	50.800	50.781	0.019	Pass
Z	0.000	0.028	0.028	Pass

slot 2				
	Nom	Meas	Dev	Test
Len	28.575	28.631	0.056	Pass
Width	9.525	9.401	0.124	Pass
X	25.400	25.418	0.018	Pass
Y	106.363	106.280	0.083	Pass
Z	0.000	-0.008	0.008	Pass



Report measurement results

Report the inspection measurement results using:

- Control views in the Control Reviewer;
- Tables and snapshots in formatted reports.

The screenshot displays a measurement report interface. At the top, there are two 3D CAD models of a part, one showing a specific feature being measured. Below the models is a table of measurement data. The table includes columns for Name, Control, Nom, Meas, Tol, Dev, and Test. The data is organized into sections for different features: circle 1 Diameter, slot 1, and circle 2 Diameter. Each feature has multiple rows for different control points (X, Y, Z) and their respective measurements and tolerances. The 'Test' column contains status indicators like 'Pass' or 'Fail'.

Units: Millimeters						
Coordinate System: world						
Data Alignment: original						
Name	Control	Nom	Meas	Tol	Dev	Test
circle 1 Diameter		25.400	25.431	±1.000	0.031	Pass
	X	25.400	25.309	±1.000	-0.091	Pass
	Y	50.800	50.810	±1.000	0.010	Pass
	Z	0.000	0.003	±1.000	0.003	Pass
slot 1	Length	34.825	35.032	±1.000	0.107	Pass
	Width	8.825	9.494	±1.000	-0.031	Pass
	X	139.700	139.349	±1.000	-0.351	Pass
	Y	25.400	25.417	±1.000	0.017	Pass
circle 2 Diameter		25.400	25.445	±1.000	0.045	Pass
	X	177.540	177.425	±1.000	-0.115	Pass
	Y	63.500	63.399	±1.000	-0.101	Pass
	Z	18.933	18.932	±1.000	-0.101	Pass

Organization: _____ Part name: _____
 Operator: _____ Part number: _____
 E-mail: _____ Piece: piece 1

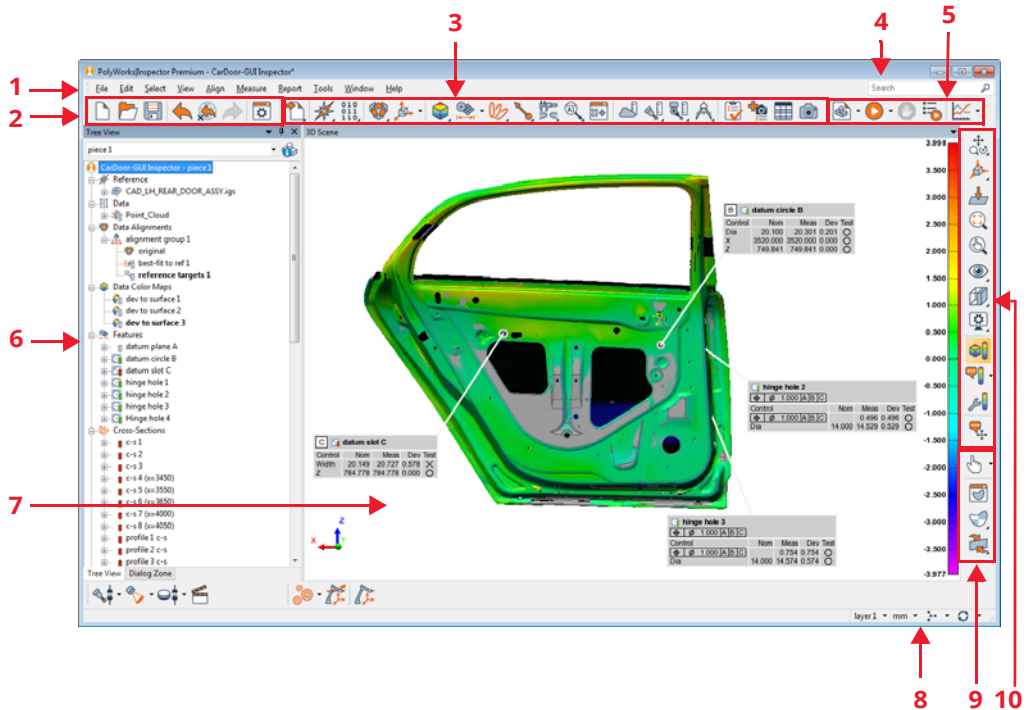
2014-03-11 3/3

Fundamentals



The graphical user interface

The user interface offers a menu bar, toolbars, a status bar, and several main dockable panes.



- 1 Menu bar: Access to all PolyWorks | Inspector functionalities.
- 2 Standard toolbar: Quick access to common operations (e.g., Open, Save, and Undo).
- 3 Main Objects toolbar: Quick access to the most common tools for an inspection.
- 4 Search box: Used to enter a query and search through Tree View objects or menu items.
- 5 Multipiece Inspection toolbar: Quick access to the various multipiece tools.
- 6 Tree View: Pane where all the objects of the project are stored.
- 7 3D Scene: 3D rendering window where objects are displayed. Provides information and allows changing project units, the active layer, the propagation status, and the automatic update status.
- 8 Status bar: Provides information and allows changing project units, the active layer, the propagation status, and the automatic update status.
- 9 Selection toolbar: Used to select objects or elements in the project.
- 10 3D Scene toolbar: Used to navigate in the 3D scene, to change the view, and to control the visibility and display of objects.

Specifying the required plug-ins

Measurement devices can be used for data acquisition in the PolyWorks Metrology Suite. To do so, the required plug-ins must be specified.

The plug-ins that will be made available in PolyWorks | Inspector depend on the settings in the Workspace Manager Options. These options are displayed the first time that the PolyWorks Metrology Suite is opened. They are also available at any time by following these steps:

1. In the Workspace Manager, choose *Tools > Options*. The Workspace Manager Options dialog box opens.
2. On the **Plug-ins** page, select the plug-ins that correspond to the scanning or probing device that will be used.
3. Click **Apply**.
4. Start PolyWorks | Inspector.



The options set on the **Plug-ins** page are automatically saved and are used at the next module startup.

Connecting to devices

To use a scanning or a probing plug-in, a connection must first be established with the device. To do so, use the Devices toolbar, which is displayed in the lower left-hand corner of PolyWorks | Inspector. It offers the following standard functionalities:



Connect to/Disconnect from Device

Connects to or disconnects from the current probing device.



Scan

Opens the Scan dialog box.



Offline Simulation Mode

Activates or deactivates a simulation mode, where operations can be performed without having to be physically connected to a device.

Particularities when scanning

An additional toolbar is offered when scanning. The first step consists in specifying the appropriate plug-in, then connecting to the device.

Connecting to a scanning device

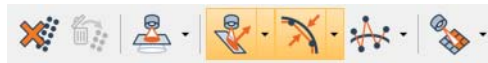
Proceed as follows to establish a connection with the device:

1. Choose *Tools > Plug-ins*.
2. Choose a device.

Alternatively, click the arrow next to the Scan button on the Devices toolbar. Choose the device from the list, then click the Scan button.



The plug-in dialog box opens and, once the scan operation is launched, a scanning toolbar appears in the 3D scene. The functions offered through this toolbar depend on the specified parameters (real-time quality meshing enabled or not) while scanning.



It offers the following buttons:



Delete Last Scan Pass

Deletes the last scan passes of the current scanning session. Offered in both cases, for scanning using real-time quality meshing or not.



Feature Scanning Guidance

Activates or deactivates feature scanning guidance.

When a clipping plane has been defined, the following button becomes available:



Clipping Plane

Activates or deactivates the use of the clipping plane when surface scanning is performed.

The following buttons are only offered when real-time quality meshing is used:



Recover Last Deleted Elements

Recovers the last deleted elements of the current scanning session.



High Scanner-to-Surface-Normal Angle

Activates or deactivates the **High Scanner-to-Surface-Normal Angle** quality metric.



Low Scan Density for Mesh Curvature

Activates or deactivates the **Low Scan Density for Mesh Curvature** quality metric.



High Noise Level

Activates or deactivates the **High Noise Level** quality metric.

The following buttons are only offered when the scanning session is launched from a measurement sequence:



Start Scan

Starts or resumes the scanning session.



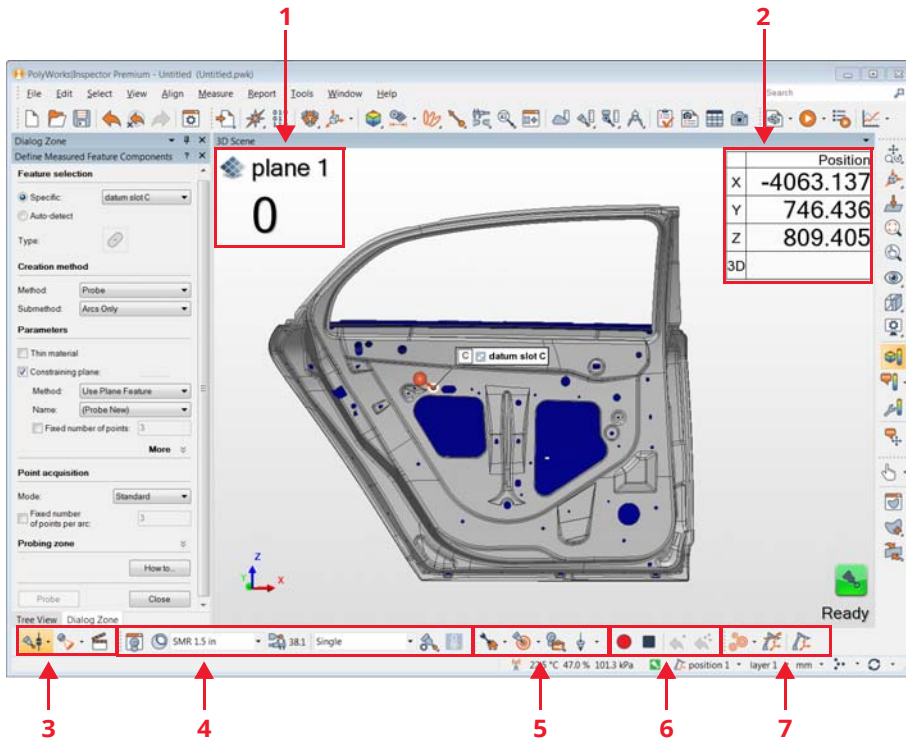
End Scan

Pauses or ends the scanning session, depending on whether scanning is performed with or without real-time quality meshing.

For information about scanning plug-ins, including real-time quality meshing and quality metrics, see [Scanning a part using a point cloud digitizer plug-in](#) on page 91.

The graphical user interface specific to probing

When probing, the user interface offers different probing toolbars, probing status information, and a Digital Readout.



- 1 Probing Status: Displays the progression in capturing points.
- 2 Digital Readout: Displays real-time information about the position of the probe and information related to the object being probed.
- 3 Devices toolbar: Offers items to connect to the probing device.
- 4 Probing Device toolbar: Offers quick access to set probing device properties.
- 5 Probing Device Control toolbar: Available for specific probing devices, such as laser trackers, to control the device.
- 6 Probing Control toolbar: Offers items to control the probing session that is being performed.
- 7 Device Positions toolbar: Offers quick access to device-position-related operations such as defining targets, moving the device, and managing device positions and targets.

Particularities when probing

Additional toolbars are offered when probing. The first step consists in specifying the appropriate plug-in, then connecting to the device. According to the selected device, one or more toolbars are added to the user interface.

Connecting to a probing device

To use a probing plug-in, proceed as follows:

1. On the Devices toolbar (located in the lower part of PolyWorks | Inspector), click the arrow next to the **Connect to Device/Disconnect from Device** button.
2. Choose the device from the drop-down list.
3. Click the **Connect to Device/Disconnect from Device** button.

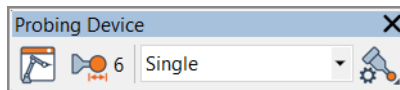


The button changes, as shown to the right.

Probing toolbars are displayed in the lower part of PolyWorks | Inspector. They contain tools and settings to facilitate the probing session. The tools and settings offered depend on the current device. In this section, only the most common functionalities are presented. For information on a functionality specific to a device, see the Plug-in Guide of this device which is located on the *Help* menu of the Workspace Manager or the PolyWorks | Inspector module.

Probing Device toolbar

For articulated arms, the Probing Device toolbar contains the following standard functionalities:



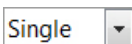
Device Properties

Opens the property sheet of the current probing device.



Tip Diameter

Displays the current probe tip diameter.



Probing Mode

Sets the probing mode for the current probing device.

Probing Options



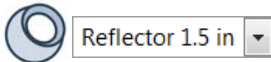
Provides quick access to options typically located on the Measurement Mode and Compensation Method pages of the PolyWorks | Inspector Options dialog box.

For laser trackers, the Probing Device toolbar offers the following standard functionalities:



Device Properties

Opens the property sheet of the current probing device.



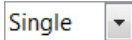
Reflector

Sets the reflector used by the current probing device.



Reflector Diameter

Displays the current reflector diameter or the Reflector Normal/Tangent offset.



Probing Mode

Sets the probing mode for the current probing device.



Probing Options

Provides quick access to options typically located on the Measurement Mode and Compensation Method pages of the PolyWorks | Inspector Options dialog box.



Temperature Stability Monitoring

Opens a dialog box used to control temperature stability monitoring.

Probing Device Control toolbar

For laser trackers, the Probing Device Control toolbar offers items to control the probing device. The following standard functionalities are offered:



Go Home

Orders the current probing device to move to, or to target, a location called "Home".



Go Front

Orders the current probing device to move to, or to target, its front.



Go to Object

Orders the current probing device to move to, or to target, the location of the selected object.



Go to XYZ

Orders the current probing device to move to, or to target, the specified XYZ coordinates.



Find Target

Orders the current probing device to find a nearby target.



Probing Device Utilities toolbar

Provides access to specific tracker functionalities.



Orient to Gravity

Measures the inclination of the current probing device in order to orient new measurements to gravity for the active device position.

Probing Control toolbar

The Probing Control toolbar offers items to control the probing session that is being performed. It offers the following standard functionalities:



Undo Last Probed Point

Removes the last probed point of the current acquisition session.



Reprobe Object

Enables an interactive mode to reprobe the last probed object.

Depending on the connected device, two additional functionalities are offered:



Start or Stop Measurement

Starts or stops a measurement session.

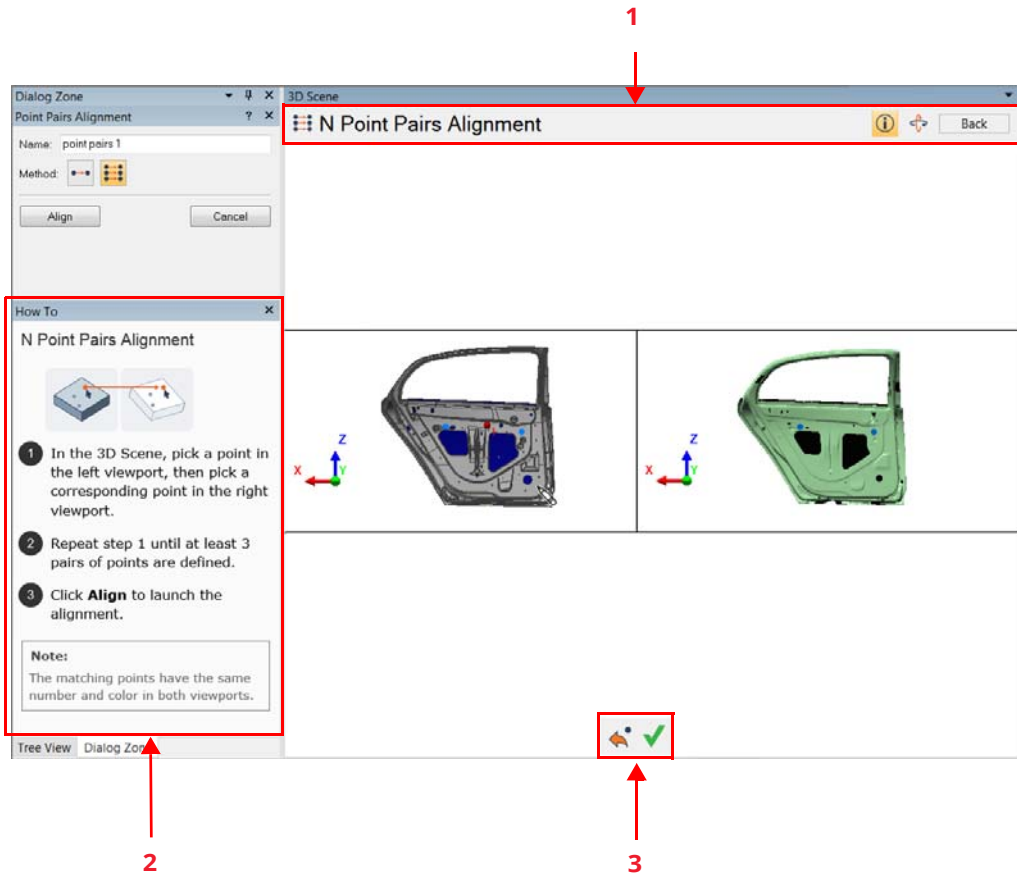


End Probing and Process Probed Data

Ends the current acquisition session and attempts to create a new object using the probed points.

Interactive mode guidance

PolyWorks offers many interactive modes. Interactive modes are modes for operations where clicking in the 3D scene is required. Interactive mode guidance helps in quickly discovering, learning, and mastering these modes. The different parts of interactive mode guidance are described below.

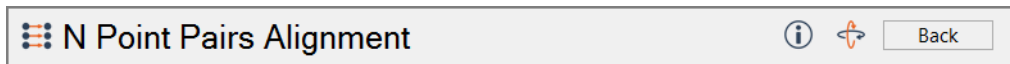


- 1 Interactive mode guidance toolbar: Provides information on the current interactive mode as well as quick access to general operations. Information on how to perform the operation can also be accessed.
- 2 How To window: Provides more information on how to perform the operation, and in some cases offers diagrams.
- 3 Contextual toolbar: When applicable, offers quick access to operations that are specific to the current interactive mode.

Interactive mode guidance toolbar

The interactive mode guidance toolbar, displayed at the top of the 3D scene while in the interactive mode, displays information regarding the current interactive mode as well as quick access to show or hide the How To window and to perform 3D scene manipulations. It can also be used to exit the current mode.

The N Point Pairs Alignment interactive mode is provided as an example.



N Point Pairs Alignment

Information on the current interactive mode

Displays the name and, when applicable, the icon for the current interactive mode.



How To Window

Opens or closes the How To window.



3D Scene Manipulations (Spacebar)

Temporarily interrupts the mode to perform translations/rotations in the 3D scene.

Close/Back

Close

Back

The **Close** button is offered to exit the current mode. If the current mode was launched through another mode, the **Back** button is offered to exit the current mode and return to the previous mode.

Notes

Shortcut keys and mouse buttons, such as the SPACEBAR and the right-click, can also be used to perform operations such as to interrupt the mode to perform translations/rotations or to exit the mode.

How To window


The How To window is displayed in the Dialog Zone when the **How To Window** button is clicked.

It offers detailed instructions and, when applicable, diagrams that guide the user in completing the operation while in the current interactive mode.

In some cases, it also offers a **Note** and a **Shortcuts** section at the bottom to provide information to further assist the user.

How To
✕

N Point Pairs Alignment



- 1 In the 3D Scene, pick a point in the left viewport, then pick a corresponding point in the right viewport.
- 2 Repeat step 1 until at least 3 pairs of points are defined.
- 3 Click **Align** to launch the alignment.

Note:
The matching points have the same number and color in both viewports.

Contextual toolbar

In some cases, a contextual toolbar is displayed at the bottom of the 3D scene. It offers buttons that provide quick access to certain operations for the current interactive mode.



Using the Search tool

PolyWorks | Inspector offers a Search tool, used to quickly find Tree View objects or menu items by entering text in the appropriate field. It is available in the upper-right corner of the interface.



Steps

1. Enter text in the Search field (partial terms can also be entered).

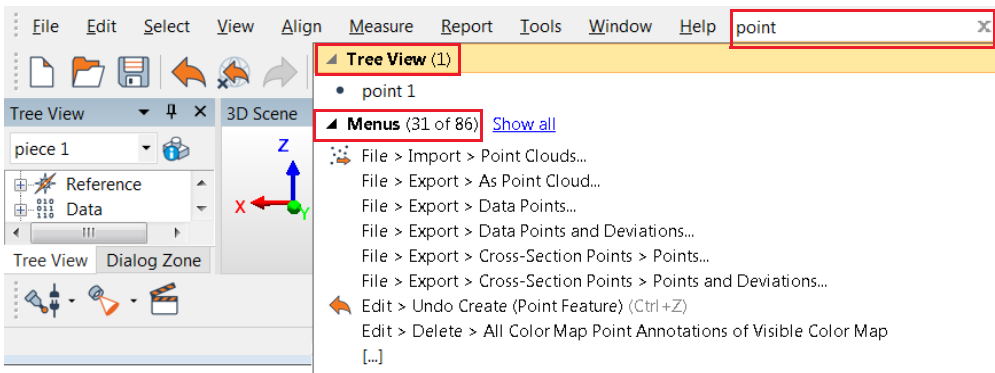
The Search tool is not case-sensitive.



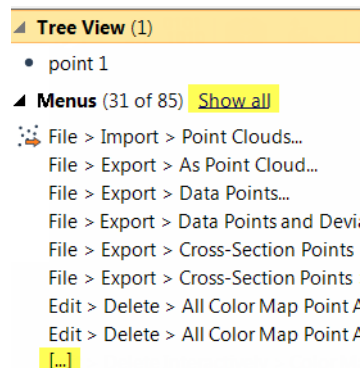
Results appear as soon as the first letter of the search string is entered.

2. If needed, continue refining the search.

The list of results is displayed under the Search field. The results are organized within the **Tree View** and **Menu** sections which can be collapsed or expanded.



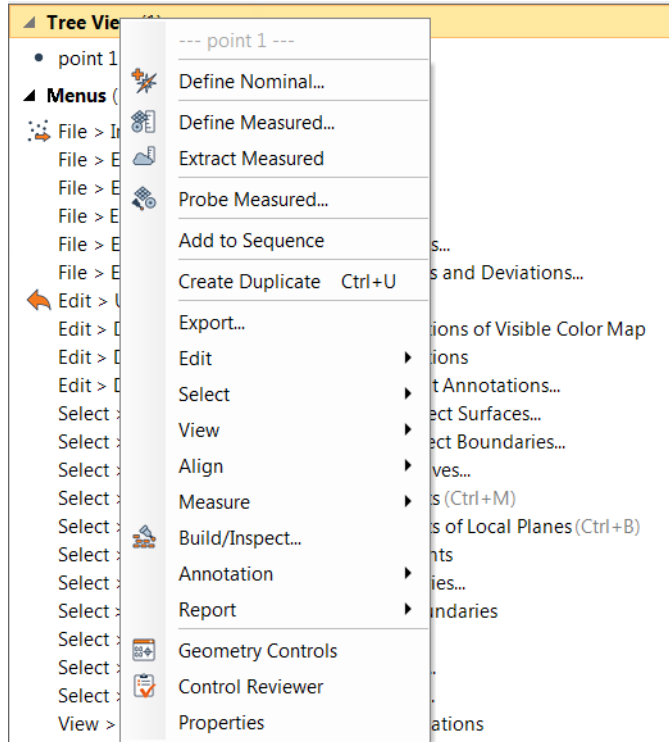
The number of hits for each section is displayed between parentheses. If a large number of results are found, they are not all displayed. To see them all, click **Show all** or the [...] symbol.



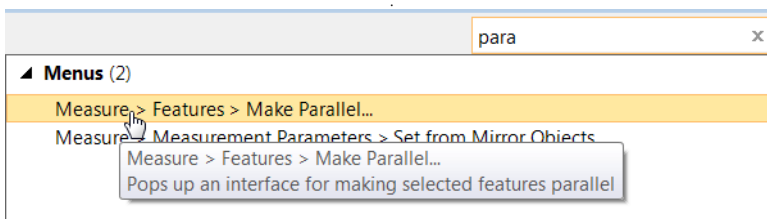
- Click the appropriate object or item in the list of search results.

Note the following:

- Right-clicking an object in the **Tree View** section displays the shortcut menu for the object, making the operations related to that object easily accessible.



- Hovering the mouse over an item in the **Menu** section displays a short description of the corresponding operation.



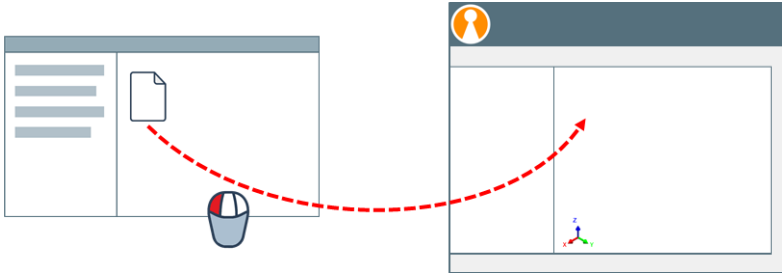
Results

- When clicking a search result for a Tree View object, the object is also selected in the Tree View and, if applicable, it is centered in the 3D scene.
- When clicking a search result for a menu item, the corresponding operation is launched.

Quickly importing Reference and Data objects

It is possible to import Reference objects and/or Data objects by dragging the selected files from a Windows file browser and dropping them anywhere in an open PolyWorks Metrology Suite module, allowing users to quickly import more than one type of object at once (i.e., point clouds, CAD models, and polygonal models).

When dragging and dropping files, file formats are automatically detected.



Objective: Quickly import Reference and/or Data objects using the drag and drop operation.

Required: A supported file format is available. A PolyWorks Metrology Suite module is open.

Steps

1. In Windows Explorer, locate the files to import.



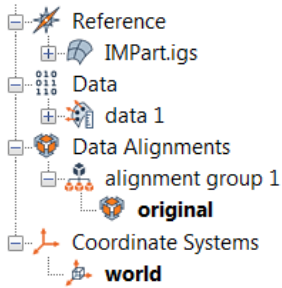
2. Drag and drop the files in an open module.



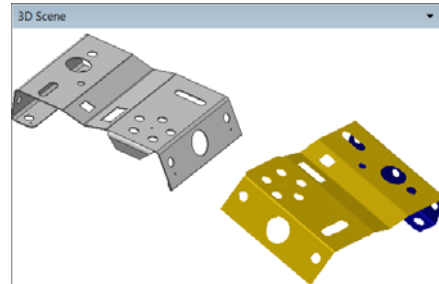
3. Specify the import parameters as needed.

Results

The files are imported and stored under the Reference branch or the Data object branch of the Tree View based on their respective formats.

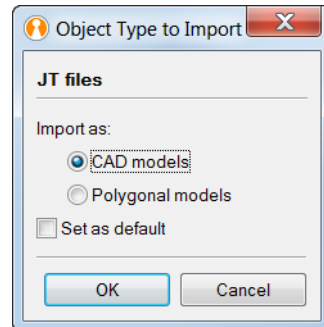


The objects are shown in the 3D scene.



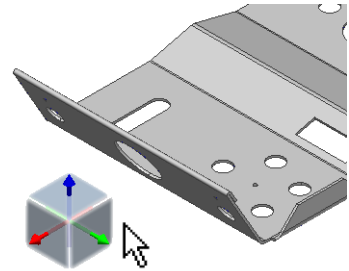
Notes

When importing JT file types, the Object Type to Import dialog box opens. Specify whether the selected objects are imported as CAD models or polygonal models.



Views in the 3D scene

The content of the 3D scene can be viewed using standard views (e.g., +X), custom user-saved views, and views based on the normal of an object.



Interactive view selector

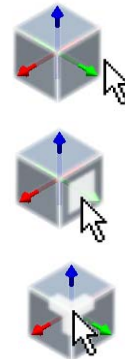
The 3D scene offers a view selector that allows specifying orthogonal or isometric views interactively.

Objective: Set a view interactively using the view selector.

Required: An object is displayed in the 3D scene.

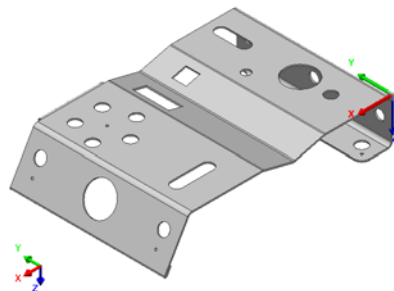
Steps

1. To display the view selector, represented by a cube, place the mouse pointer in the lower-left corner of the 3D scene.
2. To specify an orthogonal view in the 3D scene, click a face of the cube or an axis.
3. To specify an isometric view, click a corner of the cube.



Results

The 3D scene content is displayed according to the specified view clicked on the view selector. The axes are based on the active coordinate system.



Standard views

Standard views (e.g., +X) based on the active coordinate system axes are offered.

Objective: View the content of the 3D scene using a standard view.

Required: An object is displayed in the 3D scene.

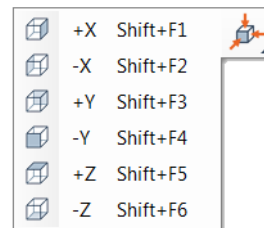
Steps

1. Choose *View > Standard Views > [choose view type]*.

Alternatively, click **Standard Views** on the 3D Scene toolbar.

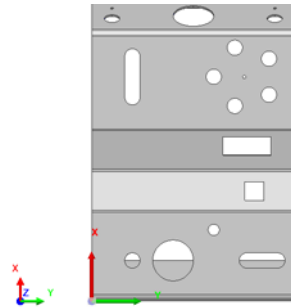


2. Choose which standard view to display in the 3D scene. This is based on the active coordinate system.



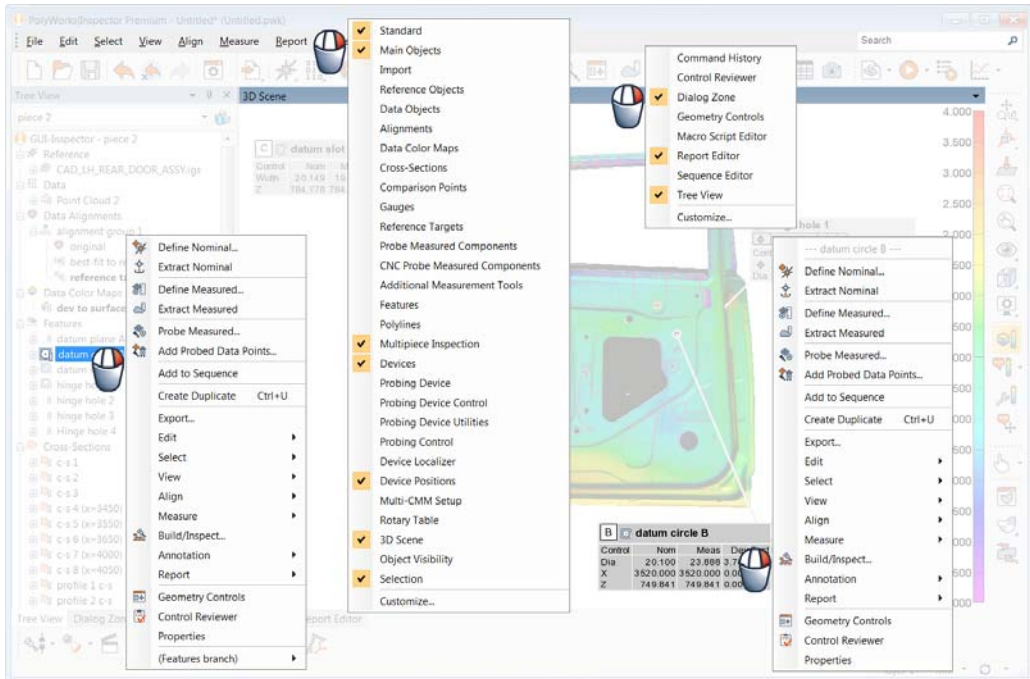
Results

The 3D scene content is displayed according to the selected view based on the active coordinate system.



Shortcuts through the interface

Shortcuts to different functionalities are offered when right-clicking elements or areas of the user interface.



Moving objects in the 3D scene

The position of objects can be modified in the 3D scene using the mouse. Rotations, translations, and zoom operations are each mapped to a specific mouse button.

Mouse-based operations:

Rotation



To rotate about the X axis and the Y axis viewing axes, click and hold the left mouse button in the 3D scene. To constrain rotations to a vertical or a horizontal viewing axis, press and hold SHIFT before clicking.

Translation



To translate along the X axis and the Y axis, click and hold the middle mouse button. Translations along the X axis or the Y axis are performed by moving the mouse in the X or Y directions respectively, in the 3D scene.

Zoom



Translating along the Z axis is equivalent to a zoom operation. Click and hold the right mouse button, then move the mouse forward and backward. When the mouse is moved forward, a positive translation is performed on the 3D object, and the scene moves nearer to the user. When the mouse is moved backward, a negative translation is performed on the 3D object, and the scene moves farther away from the user.

Zoom box

Zoom on part of an object by delimiting a rectangular area:



1. Press and hold SHIFT.
2. Middle-click to create a first rectangle corner.
3. Drag the pointer to the opposite rectangle corner.
4. Release the middle mouse button.

Rotation about the viewing axis



To rotate about the viewing axis, press and hold SHIFT, click and hold the right mouse button, then move the mouse in the right or left direction.

Mouse Button Behavior menu button



Note that the same operation can be assigned to all mouse buttons by selecting a corresponding button on the **Mouse Button Behavior** menu button, available on the 3D Scene toolbar.






An object-based approach

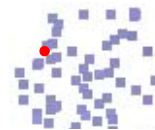
PolyWorks | Inspector offers an object-oriented approach to measurement and analysis tasks. Inspection projects describe the list of objects imported and created within PolyWorks | Inspector and they preserve editing operations performed by users. Several categories of objects can be generated within an inspection project. They are described in the subsections that follow.

Objects and their elements

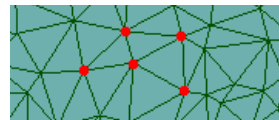
Objects are items found in the Tree View, such as Data and Reference objects. They are made up of elements, like data points in a point cloud, or surfaces in a CAD model. Both entire objects and specific elements can be considered for analysis during inspection.

Module	Objects	Elements
 PolyWorks Inspector	point cloud polygonal model CAD model	data points vertices CAD surfaces
<i>There are two interactive selection tools, one for objects and one for elements.</i>		

A point cloud contains data points.

























































A polygonal model contains triangle vertices.



A CAD model contains CAD surfaces.



As shown below, the list of objects is available in the Tree View.

 Reference	Reference object <i>Tools > Reference Objects</i>	
+  IMBlock.igs		
 Data	Data object <i>Tools > Data Objects</i>	
+  surface scan 1		
 Data Alignments		
+  alignment group 1		
+  original	Data alignment <i>Align</i>	
+  best-fit to ref 1		
 Data Color Maps		
+  dev to surface 1	Data color map <i>Measure > Deviations of Data Objects</i>	
 Features		
+  plane 1		
+  cylinder 1		
+  circle 1		
+  distance 1	Feature <i>Measure > Features</i>	
 Cross-Sections		
+  c-s 1 (y=15)		
+  c-s 2 (y=20)	Cross-section <i>Measure > Cross-Sections</i>	
 Comparison Points		
+  surf pt 1		
+  surf pt - trim pt 1	Comparison point <i>Measure > Comparison Points</i>	
+  trim pt 1		
+  surf pt - hem pt 1		
+  hem pt 1		
 Gauges		
+  Calipers	Gauge <i>Measure > Gauges</i>	
+  caliper 1		
 Custom Measurements		
+  custom 1	Custom measurements <i>Measure > Custom Measurements</i>	
+  custom 2		
 Interactive Measurements		
+  distance 2	Interactive measurements <i>Measure > Interactive Measurements > Basic</i>	
 Reference Targets		
+  surf target pt 1	Reference target <i>Measure > Reference Targets</i>	
+  surf target pt 2		
+  surf target pt 3		
 Coordinate Systems		
+  world	Coordinate system <i>Tools > Coordinate Systems</i>	
 Reports		
+  Formatted Reports		
+  report 1	Report <i>Report</i>	

Selecting objects

Objects can be selected in the Tree View or in the 3D scene by using the **Select Objects Interactively** tool. Object selection is needed in order to apply operations to specific objects.



Objective: Select an object interactively in the 3D scene.

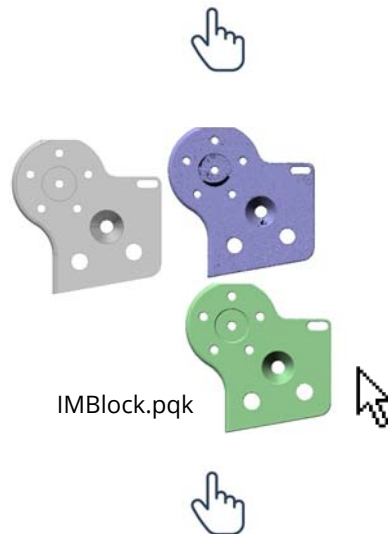
Required: The project contains a Reference or a Data object.

Steps

1. Choose *Select > Objects > Interactively*.

Alternatively, press CTRL + SHIFT + SPACEBAR to enable the object selection mode.

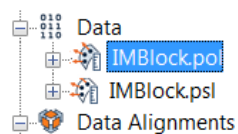
2. Pick objects in the 3D scene.
3. Press and hold CTRL to select additional objects.



4. Press the **Select Objects Interactively** button to exit the object selection mode.

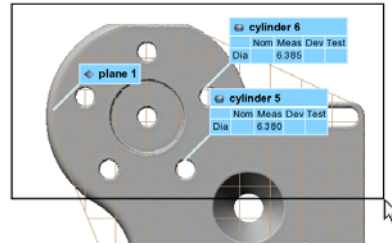
Results

The selected object is highlighted in the Tree View.

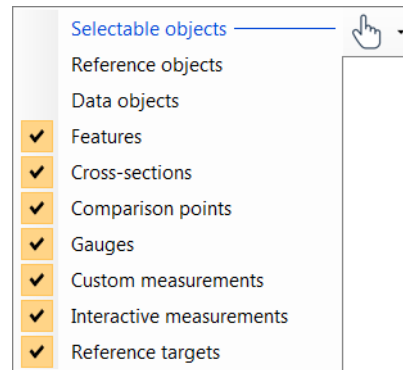


Notes

- Hold CTRL + SHIFT + middle-click while dragging to create a rectangular selection zone. The objects within the zone are selected.



- The type of objects that can be selected can be specified. Click the arrow next to the **Select Objects Interactively** button, then enable or clear the objects in the menu.



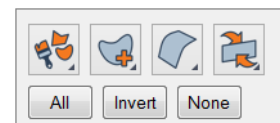
- Additional object selection tools are available on the *Select > Objects* menu, including *Select > Objects > All*, *Select > Objects > None*, and *Select > Objects > Invert*.

Selecting elements

It is possible to select a single element or groups of elements, either on the nearest surface or volumetrically (through the dataset). Element selection is needed in order to apply operations to specific elements.



The Select Elements dialog box used to select elements is available through *Select > Elements > Interactively*.



Selection tools

In the Select Elements dialog box, five selection tools are available:



Multiple

Different selection tools can be accessed depending on the mouse button and the quick access key used:

- Brush: Left-click
- Freeform Contour: Middle-click + hold
- Polygonal Contour: Middle-click + CTRL key

To select a single element: Left-click + CTRL key.



Brush

Used to perform element selection in the 3D scene by dragging the mouse over the object. Elements within the area defined by the brush tip are selected once the mouse is released.



Freeform Contour

Used to perform element selection in the 3D scene by dragging the mouse over the object and defining a free-form contour. Elements within the area defined by the contour are selected.



Polygonal Contour

Used to perform element selection in the 3D scene by left-clicking to anchor a polygonal outline until a polygonal contour has been formed. The contour is closed by left-clicking near the end of the initial line or by simply right-clicking to add a final point. Elements within the area defined by the contour are selected.



Single Element

Used to perform single element selection in the 3D scene by left-clicking the element. The type of element selected depends on the object type.

Selection modes

In the Select Elements dialog box, three Selection Modes are available:



Select

Selects elements.



Deselect

Deselects elements. Hold the ALT key as a shortcut.



Toggle

Inverts the element selection status. It can be used to select elements or deselect elements that have already been selected.

Selection types

In the Select Elements dialog box, two Selection Types are available:



Surface

Element selection is performed on the visible surfaces of the object.



Volumetric

Element selection continues through the object, including elements behind the visible surfaces.

Selectable faces

In the Select Elements dialog box, three Selectable Faces can be used to control which elements are selected:



Front

Selects only elements that are facing up in the 3D scene.



Back

Selects only elements that are facing down in the 3D scene.



Front and Back

Selects elements regardless of orientation.

Global element selection

In the Select Elements dialog box, three buttons give quick access to the global element selection operations that are available:



All

Selects all elements.



Invert

Inverts the selection of all elements.



None

Deselects all elements.

Notes

Element selection operations are also available through the *Select > Elements* menu.

Selecting elements using the Brush tool

Objective: Select multiple Reference or Data object elements with the brush tool.

Required: The project contains a Reference or Data object with selectable elements.

Steps

1. Choose *Select > Elements > Interactively*, or press SPACEBAR to enable the interactive element selection mode.



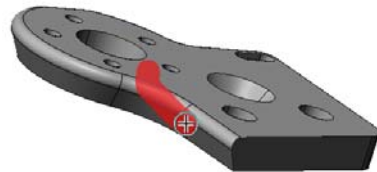
The Select Elements dialog box opens.

2. Select the **Brush** selection tool.

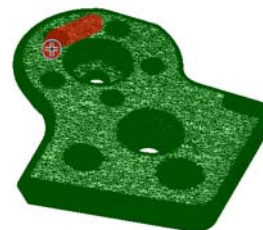


3. In the 3D scene, hold the left mouse button while dragging the mouse over the object to select all elements within the brush stroke, such as:

- CAD model surfaces.

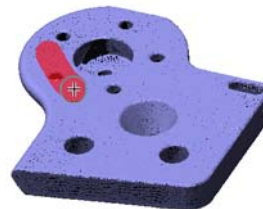


- Polygonal model vertices.



- Point cloud data points.

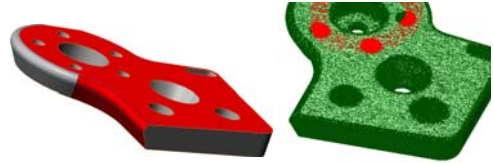
It is possible to switch to the Deselect mode by holding the ALT key and performing the same action.



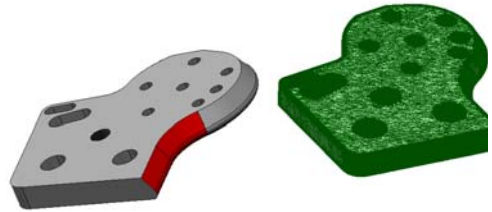
4. Press ESC or SPACEBAR to exit the selection mode and close the Select Elements dialog box.

Results

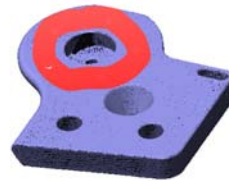
The selected elements are highlighted in red in the 3D scene.



Since the selection was performed using the **Surface** selection type, none of the elements behind the visible surfaces were selected.

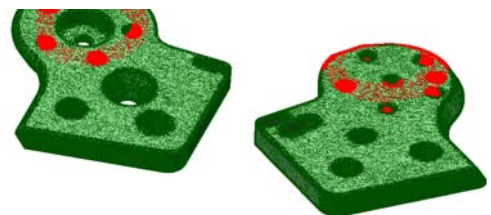


*Point clouds have no surface information, so the selection type automatically changes to **Volumetric** when it is detected that the selection is being performed on point clouds.*

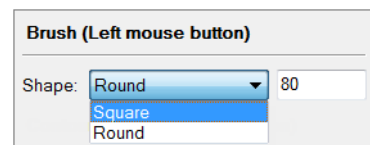


Notes

- The **Volumetric** selection type continues through the object, including elements behind the visible surfaces.



- In the **More** section, it is possible to change the shape of the brush tip to **Square**.
- The brush tip can be resized interactively in the 3D scene by scrolling the mouse wheel.



Selecting elements using the Freeform Contour tool

Objective: Select multiple Reference or Data object elements by defining a freeform contour.

Required: The project contains a Reference or Data object with selectable elements.

Steps

1. Choose *Select > Elements > Interactively*, or press SPACEBAR to enable the interactive element selection mode.



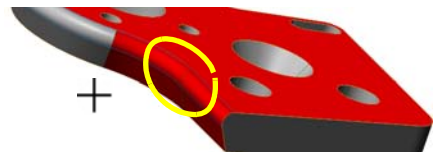
The Select Elements dialog box opens.

2. Select the **Freeform Contour** selection tool.



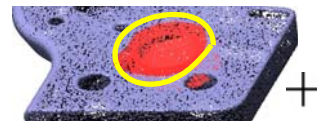
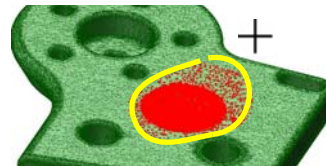
3. In the 3D scene, hold the left mouse button while dragging the mouse over the object to select elements such as:

- CAD surfaces.
- Polygonal model vertices.



- Point cloud data points.

It is possible to switch to the Deselect mode by holding the ALT key and performing the same action.



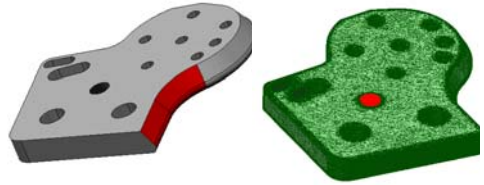
4. Press ESC or SPACEBAR to exit the selection mode and to close the Select Elements dialog box.

Results

The selected elements are highlighted in red in the 3D scene.



Since the selection was performed using the **Surface** selection type, none of the elements behind the visible surfaces were selected.

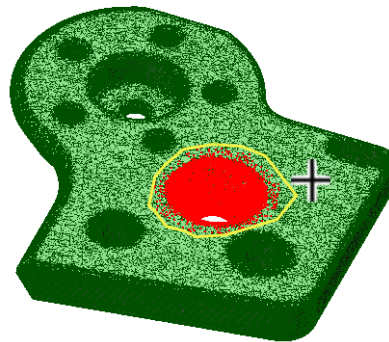


Point clouds have no surface information, so the selection type changes to **Volumetric** automatically when it is detected that the selection is being performed on point clouds.

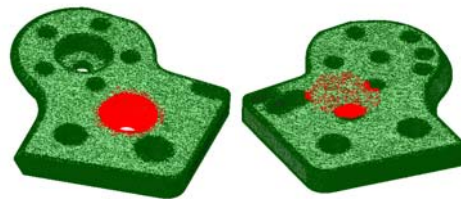


Notes

- To perform a selection with a polygonal contour, select the **Polygonal Contour** selection tool. Left-click to anchor corners of the polygonal outline; to close the contour, left-click near the end of the initial line or simply right-click to add the last point.



- The **Volumetric** selection type continues through the object, including elements behind the visible surfaces.



Selecting a single element

Objective: Select a single element on a Reference or Data object.

Required: The project contains a Reference or Data object.

Steps

1. Choose *Select > Elements > Interactively*, or press SPACEBAR to enable the interactive element selection mode.



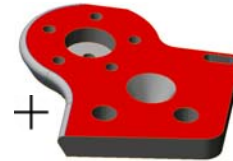
The Select Elements dialog box opens.

2. Select the **Single Element** selection tool.

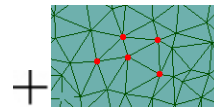


3. In the 3D scene, left-click on:

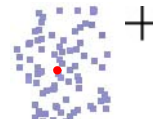
- a CAD surface to select it.



- a polygonal model to select the nearest vertex.



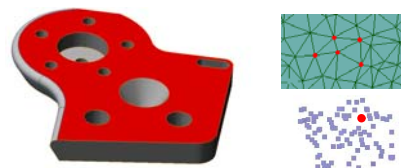
- a point cloud to select the nearest data point.



4. Press ESC or SPACEBAR to exit the selection mode and to close the Select Elements dialog box.

Results

The selected surface, vertex, or data point is highlighted in red in the 3D scene.



Importing Reference objects





Introducing Reference objects

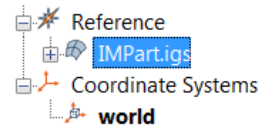
Reference objects are the nominal objects (typically a CAD model or a polygonal model) to which the Data objects are measured.

Categories of Reference objects

There are two categories of Reference objects. The following table presents each category:

Icon	Type	Description
	CAD Reference objects	All imported CAD formats.
	Polygonal Reference objects	Polygonal models as well as IMAlign projects and planar grid/mesh point clouds that have been triangulated.

Once a Reference object is imported, it is displayed in the Tree View under the Reference branch.



Assigning tolerances

Tolerances are automatically assigned to Reference objects. A tolerance is useful for obtaining pass/fail inspection results. For example, when a data point is no farther from the surface than a given tolerance value, the point “passes” the tolerance test. If the data point is farther from the surface than the same tolerance value, the point “fails” the tolerance test. See [Assigning tolerances to Reference objects](#) on page 80.

Controlling CAD model display colors

The color used to display CAD models can be controlled through the properties of the CAD Reference object.

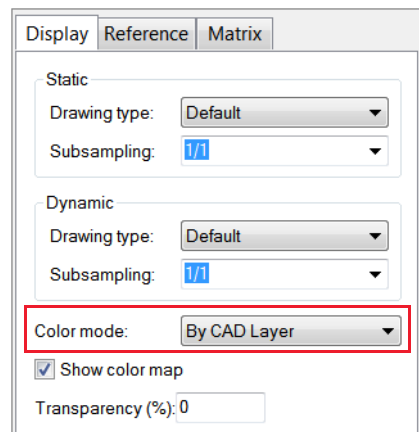
Three color display modes are offered: **Object, By CAD Layer, By CAD Surface**.

Objective: Change the color display mode of the CAD model.

Required: The project contains a CAD model with embedded surface color information imported along with layers.

Steps

1. Select the CAD model in the Tree View.
2. Choose *Edit > Object Properties*.
3. On the **Display** tab, choose from the **Color mode** list one of the following options:
 - **Object**
 - **By CAD Layer**
 - **By CAD Surface**
4. Click **Apply**.

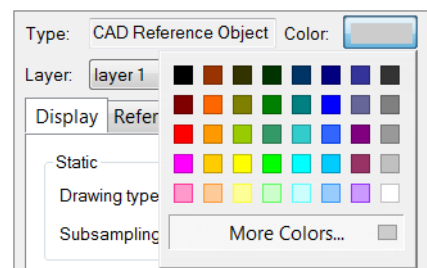


Results

Object

The CAD Reference object color is displayed using the defined color mode. The whole CAD Reference object is the same color.

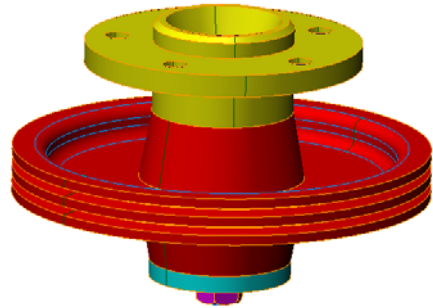
*The CAD Reference object color can be changed from its property sheet using the **Color** button.*



By CAD Layer

The CAD Reference object color is displayed using the defined CAD layer colors.

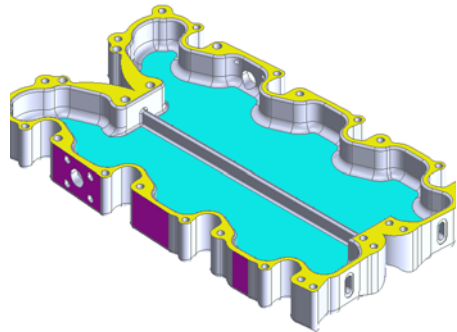
If specific surfaces of the CAD Reference object are associated to specific CAD layers, each surface is colored depending on the color definition of its associated CAD layer.



By CAD Surface

The CAD Reference object color is displayed using the defined CAD surface colors.

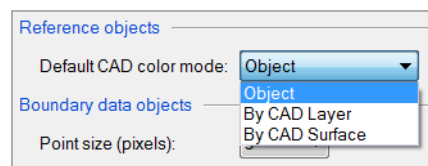
The surface color is defined in the CAD software and imported automatically in PolyWorks|Inspector.



Notes

The default color mode can be set in the PolyWorks|Inspector options and saved in the user configuration.

1. Choose *Tools > Options* to access the PolyWorks|Inspector options.
2. Open the **Display > Reference/Data** page.
3. Choose the **Default CAD color mode**.
4. Click **Apply**.
5. Choose *Tools > Save User Configuration* to save the user configuration.



Using CAD models to describe nominal parts

It is common practice, in the industry, to design the shape of parts using 3D drawing software, commonly known as computer-aided design (CAD) software. CAD models greatly simplify the task of defining nominal geometries and dimensions, an essential step in the process of inspecting parts.



Importing a CAD model

Objective: Import a CAD model, into the inspection project, as a Reference object.

Required: A CAD file in a neutral format (IGES or STEP file) or a native format (such as CATIA, NX, Creo, SolidWorks, or Inventor).

Steps

1. Choose *File > Import > CAD Models*.

A file browser opens.

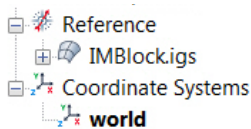


2. Browse for the CAD file and select it.

3. Click **Open**.

Results

The CAD model is added to the Tree View, under the Reference branch.



The CAD model is displayed in the 3D scene.



Importing Product and Manufacturing Information (PMI) with the CAD model

Some computer-assisted design platforms have the capability to add geometric dimensioning and tolerancing controls to the model in order to have this information embedded in the file. Depending on the CAD file format, the import options may include extracting features and their feature control frames to import the model with the GD&T control information. The import options are offered on the **CAD Models** page of the PolyWorks|Inspector Options.

Objective: Import a CAD model with embedded GD&T control information.

Required: CAD model with embedded GD&T controls in a format such as CATIA V5 and V6, Creo, JT, NX, SAT, or SolidWorks.

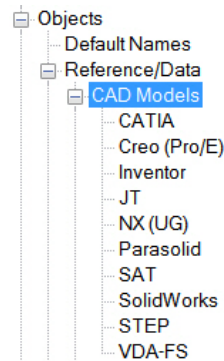
Steps

1. Choose *Tools > Options*.

The PolyWorks|Inspector Options dialog box opens.



2. Open the **Objects > Reference/Data > CAD Models > [choose a file format]** page.
3. For each required CAD file format, select **Import features and controls**.
4. Click **OK**.



5. Choose *File > Import > CAD Models*.

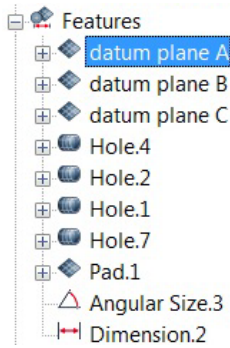
A file browser opens.



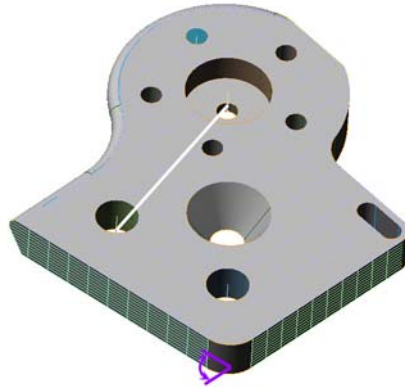
6. Browse for the CAD file and select it.
7. Click **Open**.

Results

The CAD model is imported with the GD&T controls and the nominal feature components are created automatically.



The CAD model is displayed in the 3D scene, along with the features.



Notes

- A CAD translator is required to read the native CAD file formats.
- The Features and Controls Import Log is added to the Tree View, under the Reports branch. It provides information about the operations done by the PolyWorks Metrology Suite during the import process.

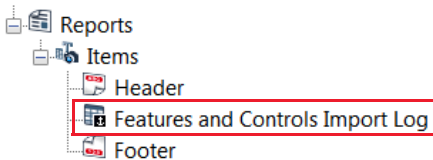


Table Type Features and Controls Import Log

CAD file name	IMBlock.CATPart
CAD part name	IMBlock.CATPart
Total number of features	11
Number of features loaded	11
Number of features with geometry errors	0
Number of features with GD&T errors	0
Number of features with dimension errors	0
Number of features with geometry not supported	0
Number of features with GD&T not supported	0
Number of features with dimension not supported	0
Number of features with geometry warnings	0
Number of features with GD&T warnings	0
Number of features with dimension warnings	1

Using polygonal models to describe nominal parts

Polygonal models are meshes composed of triangles and vertices that are the result of a merging process of the data points acquired by a scanning device. Polygonal models are generally imported as Data objects, but can also be imported as Reference objects. For example, the original model can be imported as a Reference object to monitor wear on a part over time.



Converting a polygonal Data object to a Reference object

Objective: Import a polygonal model as a Data object and convert it to a Reference object.

Required: A polygonal model file in a PolyWorks format (such as POL or PQQ), or a neutral format (such as STL or OBJ).

Steps

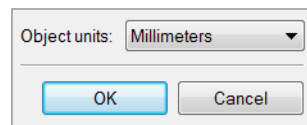
1. Choose *File > Import > Polygonal Models*.



2. Browse for the file to import.

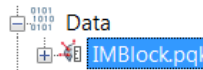
Common polygonal file formats include .pol, .pqq, .stl, .obj, and .ply.

3. In the **Object Units** list, select the native units of the file then click **OK**.



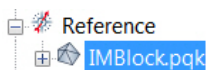
The polygonal model is imported as a Data object.

4. Select the polygonal model in the Tree View, then choose *Tools > Data Objects > Convert to Reference Objects*.



Results

The polygonal model is now added to the Tree View, under the Reference branch.



The polygonal model is displayed in the 3D scene.



Assigning tolerances to Reference objects

Tolerances, assigned to Reference objects, define upper and lower envelopes used by measurement tools to determine whether a measured value is within tolerance (pass) or out of tolerance (fail). Tolerances can be assigned to the entire Reference object at once, or to individual surfaces or areas.



The examples shown in the following subsections use a CAD model. Note that the procedure is applicable to any type of Reference object.

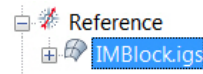
Assigning tolerances to the entire Reference object

Objective: Assign tolerances to all the surfaces of the CAD model.

Required: A CAD file, imported as a Reference object.

Steps

1. Select the CAD model in the Tree View.



2. Choose *Tools > Reference Objects > Tolerances*.

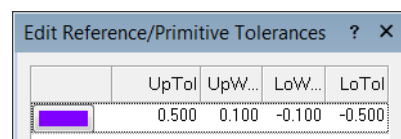
The Edit Reference/Primitive Tolerances dialog box opens.



By default, all the CAD surfaces are assigned a tolerance of +/- 0.5 mm, which is shown in purple.



3. Enter a new value to modify the tolerance.



Assigning tolerances to specific surfaces

Objective: Assign tolerances to specific surfaces of the CAD model.

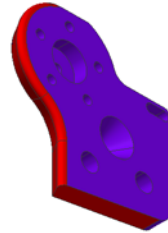
Required: A CAD file, imported as a Reference object.

Steps

1. Choose *Select > Elements > Interactively* to enable an interactive picking mode.



2. Left-click the CAD surfaces where a different tolerance must be assigned.

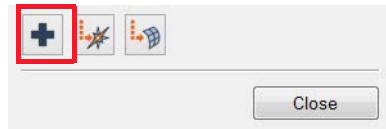


3. Choose *Tools > Reference Objects > Tolerances*.

The Edit Reference/Primitive Tolerances dialog box opens.

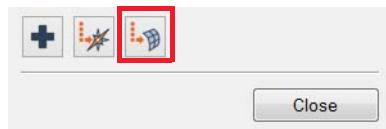


4. In the dialog box, click **Add** to create a new tolerance.



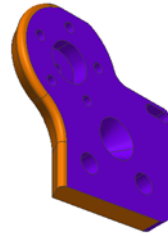
5. Enter the values for the new tolerance.

6. Click **Assign to Selected CAD Surfaces**.



Results

A new tolerance is added and is assigned to the selected CAD surfaces.



Acquiring Data objects

0 1 0
0 1 1
1 1 0









Introducing Data objects

In a typical inspection process, Data objects represent the objects that will be inspected. They can be imported from a standard format or created within PolyWorks | Inspector as a result of line scanning, probing physical objects, or using other operations (e.g., transferring).

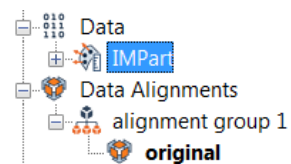
Categories of Data objects

There are three types of Data objects covered in this guide. The following table presents each type:

Icon	Type	Description
 	Surface Data objects (with normals) Surface Data objects (without normals)	<ul style="list-style-type: none"> Point Clouds (organized and unorganized¹, line scans¹, planar grid/meshes², IMAlign projects², and probed surface point clouds¹). They are obtained by importing files, scanning, or probing. Polygonal Models² obtained by importing files or by scanning.
 	Boundary Data objects (with normals) Boundary Data objects (without normals)	Boundary point clouds obtained by importing unorganized point clouds ¹ , scanning, or probing ³ .
 	Probed Data objects (with normals) Probed Data objects (without normals)	Data points obtained when: <ul style="list-style-type: none"> Probing measurement objects (i.e., features, comparison points, cross-sections, and gauges). Performing a surface point alignment.

1. With or without normal information.
2. Always have normal information.
3. Without normal information.

Data objects are displayed in the Tree View, under the Data branch.



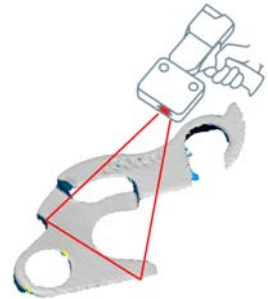
Introducing surface and boundary Data acquisition

PolyWorks | Inspector offers the capability to acquire data through the use of scanning and probing plug-ins. These plug-ins are only available if they have been selected from the list of plug-ins, which loads them during module startup such as when starting PolyWorks | Inspector. For more information, see [Specifying the required plug-ins](#) on page 41.

Data acquisition by scanning

When scanning in PolyWorks | Inspector, it is possible to capture the surface of a part or the outer trim boundaries and the inner hole boundaries of thin parts. Data is acquired in the form of a polygonal model or a point cloud.

Scanning plug-ins are offered for various devices, including articulated arms, trackers, optical systems, photogrammetry-based systems, robots, custom encoder-based translation and rotation systems, and so on.



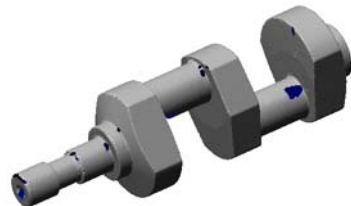
Scan types

Three types of scanning are offered:

Surface scanning

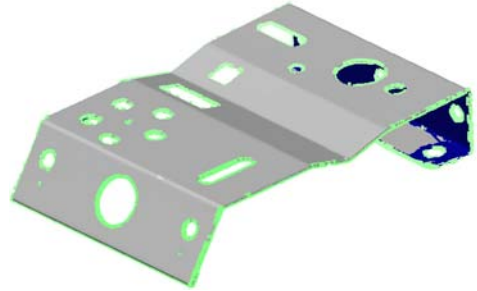
Surface Scanning is the most common scanning technique. It gathers data points across all the material within the digitizer acquisition range. If using the real-time meshing technology, those data points are converted to a polygonal model.

See [Scanning surfaces of a part to obtain a raw point cloud Data object](#) on page 98.



Surface and boundary scanning

Surface and Boundary Scanning combines the surface scanning behavior with the capability to delineate the boundaries of the part and of holes. Surface scanning captures data points across the part. Boundary scanning detects large gaps between surface points as part holes and saves the innermost data points as a boundary scan Data object. It also detects the end of scanned lines and saves the end points in the boundary scan Data object as well.



These boundary points can then be used for data color maps. This scanning method is commonly used when dealing with thin parts.

See [Scanning surfaces and boundaries of thin parts](#) on page 102.

Sharp edge scanning

Sharp Edge Scanning calculates the theoretical sharp edge of any hole or part boundary by extrapolating from laser readings on the part surface and sidewall.

See [Scanning sharp edges of thin parts](#) on page 105.



Using a clipping plane

When scanning a part, oftentimes, a portion of the table on which the part lies is also scanned. In these situations, a clipping plane is useful to automatically delete points on the table while scanning.

This reduces the manual clean-up of the scanned data, which makes it easier to scan both sides of a part.

The data points are shown during the scan pass but are permanently deleted at the end of the pass.

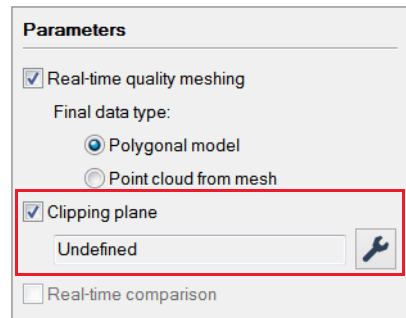
The same clipping plane can be reused from one piece to another, and from one project to another, as it is saved in the user configuration.

Objective: Define a clipping plane prior to scanning.

Required: A line scanning device is ready to use. The required plug-in is loaded in PolyWorks | Inspector.

Steps

1. From the Scan dialog box, enable **Clipping plane**.

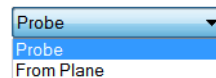


2. Click **Define Clipping Plane** to define a clipping plane.

The Define Clipping Plane dialog box opens.



3. Choose a **Method** from the following choices.



Probe

This method is used to define the clipping plane by probing a plane.

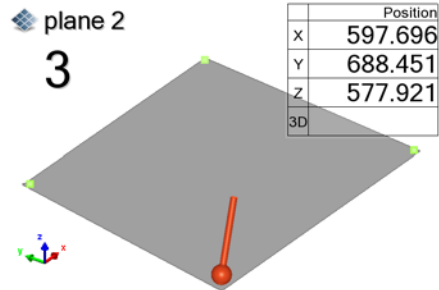
1. In the **Method** list, select **Probe**.

2. Click **Probe**.

A probing device must be selected in the probing plug-ins list. The device must be ready to use.

It should be the same device that will be used to scan the part.

3. Probe the plane, which is typically the table on which the part is resting.

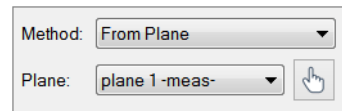


From Plane

This method is used to define the clipping plane from a plane feature that is already defined in the Tree View.

1. In the **Method** list, select **From Plane**.

Note that the selected plane must be aligned to the part that will be scanned.



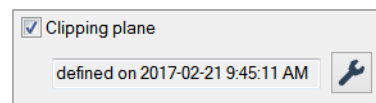
2. Select a **Plane** from the list.

Alternatively, click the **Pick Plane** button to the right of the list box to pick a plane primitive directly in the 3D scene.

3. Click **Define**.

Results

The clipping plane is defined, and automatically saved to the user configuration.



Notes

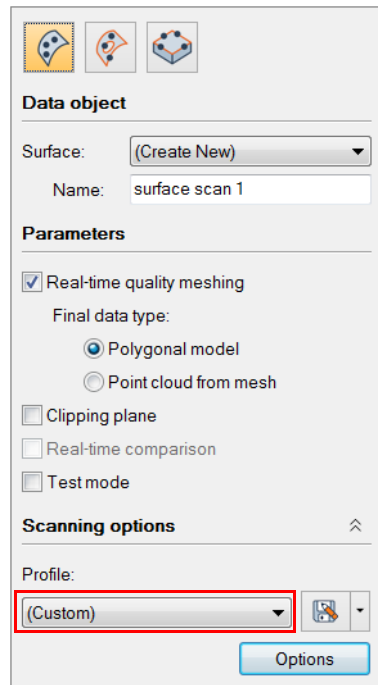
- A clipping plane defined using the **Probe** method does not appear in the Tree View nor is it displayed in the 3D scene.
- The data points filtered by the clipping plane while scanning are permanently deleted.

Scanning parameters

Scanning profiles are offered in the plug-in dialog box. On choosing a profile, the value of scanning parameters changes accordingly, simplifying the setup of a new laser scanning session. Five predefined profiles are available, which are adapted to the size of the smallest details that will be captured:

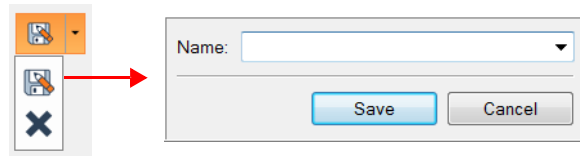
- Coarse Resolution: for details greater than 2.0 mm in size
- Standard Resolution: for details greater than 1.0 mm in size
- Fine Resolution: for details greater than 0.5 mm in size
- Extra Fine Resolution: for details less than 0.5 mm in size
- Large Standoff: for scanners with a standoff distance over 500.0 mm

When the plug-in dialog box is opened, the latest active profile is loaded. If a laser scanning parameter is modified, the profile in the list box is set to **(Custom)**.



The screenshot shows a software dialog box for configuring scanning parameters. At the top, there are three icons representing different scanning modes. Below them is the 'Data object' section, which includes a 'Surface' dropdown menu set to '(Create New)' and a 'Name' text field containing 'surface scan 1'. The 'Parameters' section contains several options: 'Real-time quality meshing' is checked, 'Final data type' is set to 'Polygonal model', and 'Clipping plane', 'Real-time comparison', and 'Test mode' are unchecked. The 'Scanning options' section is expanded, showing a 'Profile' dropdown menu with '(Custom)' selected, highlighted by a red rectangular box. To the right of the dropdown is a small icon. At the bottom right of the dialog is an 'Options' button.

A drop-down menu, to the right of the list box, allows saving the current scanning parameters to a user scanning profile or deleting the selected user scanning profile.



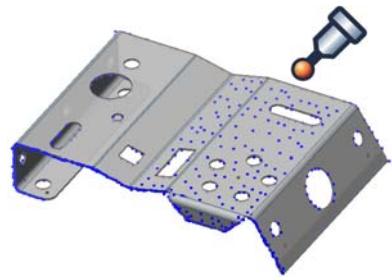
If not saved, **(Custom)** profiles are lost when another profile is selected.

Once the scan operation is launched, a scanning toolbar appears in the 3D scene. For information, see [Connecting to a scanning device](#) on page 42.

Data acquisition by probing

A probing device (such as a laser tracker or an articulated arm) can be used directly within the PolyWorks | Inspector user interface to digitize parts. Point clouds can be probed on the part surface, or along its boundaries (edges):

- When probing on the part surface:
 - If the probing device is aligned to the Reference object, the points can be compensated toward the closest surface.
 - If the probe tip has an orientation, that orientation can be used to assign point normals.
- When probing along the part edges, a boundary point cloud is created. This method can be used to measure the perimeter shape of a thin-walled part, such as a sheet metal part.



For information on the user interface when probing, see [Connecting to a probing device](#) on page 45.

Acquiring surface and boundary Data objects

Scanning a part using a point cloud digitizer plug-in

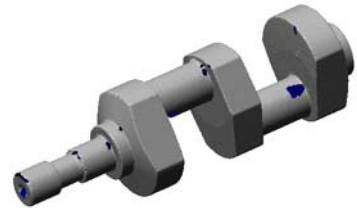
A point cloud digitizer plug-in is used to scan the surfaces and boundaries of a part and obtain a point cloud Data object or a polygonal Data object.

Meshing point cloud data in real time while scanning

The surfaces of a part can be meshed in real time while scanning. Creating a mesh while scanning can be done to obtain optimal surface data quality and highlight areas where data is missing.

Objective: Scan the surfaces of a part using a laser line scanner in PolyWorks | Inspector, automatically producing a polygonal model from scan passes in real time.

Required: A line scanning device is available and ready to use. The required plug-in is loaded in PolyWorks | Inspector.



Steps

1. Choose *Tools > Plug-ins > [device name]*.
The plug-in dialog box opens.
2. Select the **Surface Scanning** scan type.



3. In the **Surface** list, select **(Create New)** to create a new Data object or specify an existing Data object to which data will be added. See [Adding data to a scanned Data object](#) on page 127.
4. Specify the **Name** of the new surface Data object.
5. Select **Real-time quality meshing**.
6. Select **Polygonal model** as the final data type.
7. If desired, enable the **Clipping plane**.
8. Under **Scanning options**, select the appropriate **Profile** based on the size of the smallest details on the part.

Data object

Surface: (Create New)

Name: surface scan 1

Parameters

Real-time quality meshing

Final data type:

Polygonal model

Point cloud from mesh

Clipping plane

Real-time comparison

Test mode

Scanning options ⌵

Profile:

Standard Resolution 🔄

Options

9. In the plug-in dialog box, click **Start Scan**.

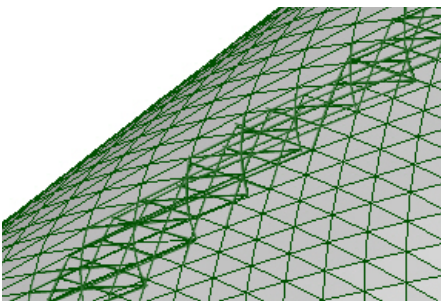
The scanning toolbar appears in the 3D scene, providing quick access to the quality metrics. See [Analyzing the quality of the digitized polygonal model using metrics](#) on page 93.



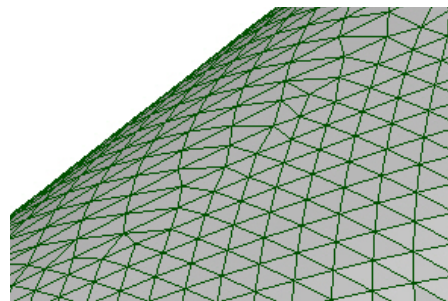
10. In the plug-in dialog box, click **End Scan** when scanning is completed.
11. Click **Finalize Mesh** to complete the polygonal model.

Once scanning is completed, the mesh must be finalized. The seams that are left visible in the mesh that is under construction are automatically stitched to produce a high-quality mesh.

Before finalization

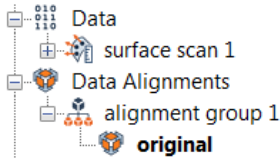


After finalization

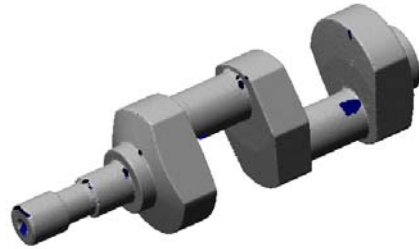


Results

The polygonal model is added to the Tree View, under the Data branch, as a surface Data object.



A polygonal model is displayed in the 3D scene.



Notes

When specifying **Point cloud from mesh** as the final data type, the point cloud has the same quality and precision as the polygonal model output of the real-time quality meshing engine, without the extra computing time required by the mesh finalization phase. The mesh-based point cloud can be converted into a polygonal model at any time by selecting it in the Tree View and choosing *Tools > Data Objects > Convert Mesh-Based Point Clouds to Polygonal Models*.

Analyzing the quality of the digitized polygonal model using metrics

When using real-time quality meshing, the scan quality is monitored and low-quality areas are detected using four quality metrics: High scanner-to-surface-normal angle, Low scan density for mesh curvature, High noise level, Detect misaligned scan passes.

The activation state and the parameters for each quality metric can be configured through the **Surface Scan > Real-Time Quality Meshing > Quality Metrics** page of the Line Scan Options dialog box.

The scanning toolbar, displayed at the bottom of the 3D scene when the scanning mode is launched, provides quick access to the three quality metrics that are computed directly on the polygonal model as well as to their associated parameters. Click the arrow to the right of the buttons to modify the default values for each quality metric. All quality metrics can be enabled simultaneously.



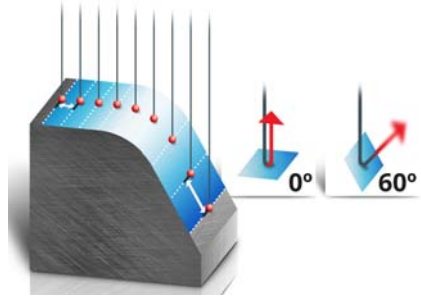
When a quality metric is activated in the scanning toolbar, areas of the scanned polygonal model that are identified as problematic are highlighted using a color specific to each metric. The four quality metrics are described in the following subsections:



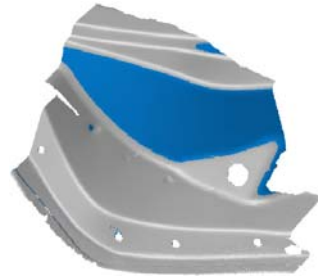
High scanner-to-surface-normal angle

The **High scanner-to-surface-normal angle** quality metric controls the angle between the digitizing vector (direction along which the laser light is projected) and the normal vector of the scanned surface. This quality metric is enabled by default.

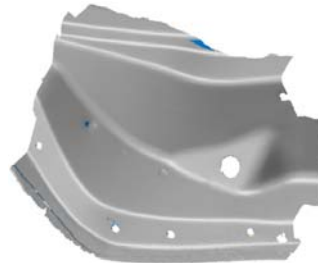
The best quality data is acquired when the laser is normal to the scanned surface.



When the angle between the digitizing vector and the surface normal vector is greater than the specified value, the area is highlighted.



When new scan passes are added covering the same area with an acceptable angle, the highlighting is automatically removed.

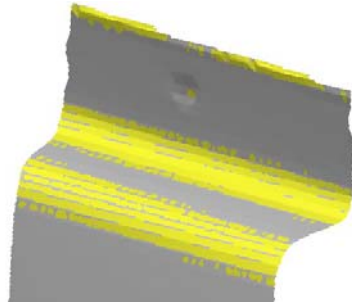
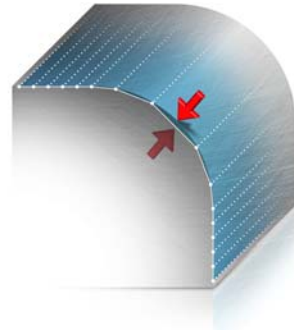




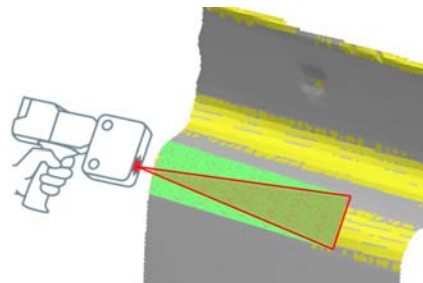
Low scan density for mesh curvature

The **Low scan density for mesh curvature** quality metric, enabled by default, identifies problematic areas where the local resolution of data points is insufficient to properly obtain a smooth curvature transition.

In low-curvature areas, a low density of scanned data points is acceptable while in high-curvature areas a higher density is required. Otherwise, the problematic area is highlighted.



To increase the density in areas of curvature, scan at a slower speed, keeping the laser line perpendicular to any fillet or sharp edge, to ensure that the laser line wraps fillets or radii when possible.





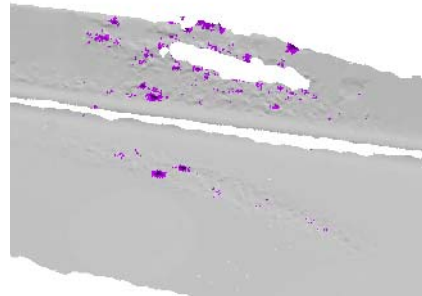
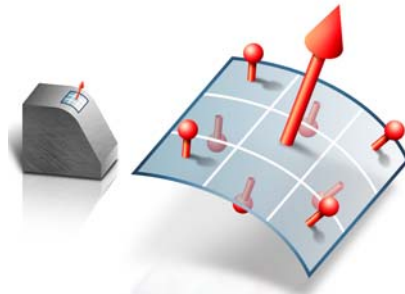
High noise level

The **High noise level** quality metric detects high noise levels on scanned data, often resulting from specular reflection (shiny material), dark materials (less reflected light), or textured surfaces (output from machining).

Local areas where high deviations are detected compared to the surroundings are highlighted indicating the presence of a high noise level.

Note that this quality metric is deactivated by default because its computational time is more significant. It is recommended to activate this metric every few scan passes.

To enable this quality metric, click the **High Noise Level** button on the scanning toolbar. Click the arrow to the right of the button to modify the **Standard deviation threshold** value.



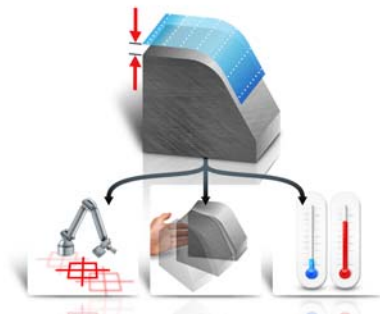
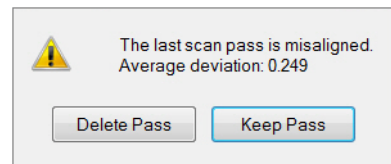
Detect misaligned scan passes

The **Detect misaligned scan passes** quality metric, enabled by default, analyzes each scan pass as it is captured.

Certain circumstances, such as incorrect arm calibrations, part displacements, or temperature changes can cause the misalignment of incoming scan passes, becoming unacceptable based on a specified **Max average deviation**.

If a scan pass is misaligned, a message appears. Based on the scanning accuracy requirements, choose to delete or to keep the scan pass.

In the **Behavior when misaligned** list, select **Confirm Pass Deletion** if a user confirmation is required prior to deleting misaligned scan passes.



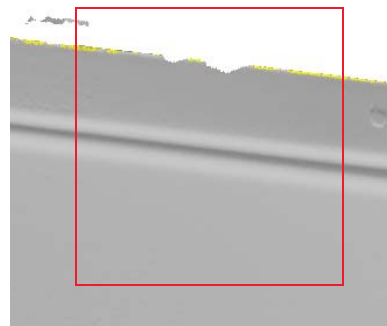
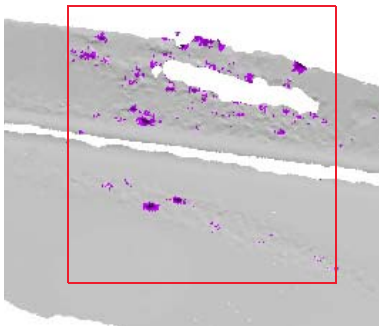
Enhancing the quality of the mesh

Once the issues have been identified using the quality metrics, the operator can rescan until the low-quality areas are no longer highlighted and the polygonal model meets the required quality criteria. This visual feedback ensures systematic high-quality measurements and reduces the variability of scanning results.

According to the identified issues, up to four corrective actions can be taken:

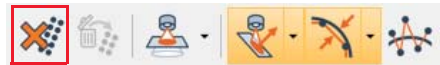
- Add new scans using a better technique
 - Scanning at a moderate speed.
 - Maintaining a good angle between the scanner and the surface.
 - Keeping the laser line perpendicular to any fillets or sharp edges.
- Delete scans and rescan

In the following example, a problematic area caused by specular reflection has been corrected by deleting the area and rescanning it.



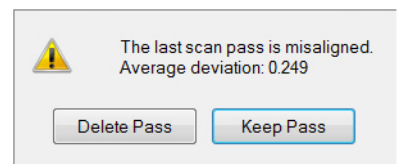
- Delete last scan pass

If the last scan pass captured is of low quality, click the **Delete Last Scan Pass** button on the scanning toolbar to delete it and rescan.



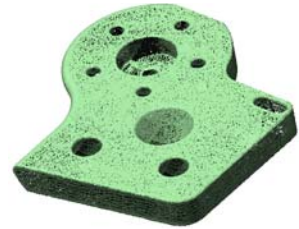
- Reject misaligned scans

Reject misaligned scans even before they are integrated into the polygonal model that is under construction. Click **Delete Pass** in the message that appears.



Scanning surfaces of a part to obtain a raw point cloud Data object

Surface scanning is the most common scanning technique. It gathers data points from all material within the digitizer acquisition range. As data is captured from different angles, point density increases in overlapping areas. Each data point location (x, y, z) is recorded and the point neighborhood is analyzed, allowing point normals (i, j, k) to be calculated.



Objective: Scan the surfaces of a part using a plug-in to obtain a raw point cloud directly in PolyWorks|Inspector.

Required: A line-scanning device is available and ready to use. The required plug-in is loaded in PolyWorks|Inspector.

Steps

1. Choose *Tools > Plug-ins > [device name]*.

The plug-in dialog box opens.



2. Select the **Surface Scanning** scan type.



3. In the **Surface** list, select **(Create New)** to create a new surface Data object or specify an existing surface Data object to which data will be added.
4. Specify the **Name** of the new surface Data object.
5. Clear **Real-time quality meshing**.
6. If desired, enable the **Clipping plane**.
7. Under **Scanning options**, select the appropriate **Profile** based on the size of the smallest details on the part.

Data object

Surface: ((Create New)) ▼

Name: surface scan 1

Parameters

Real-time quality meshing

Clipping plane

Real-time comparison

Test mode

Scanning options ⌵

Profile: Standard Resolution ▼ ▼

Options

8. If required, click **Options** to set the different options for the specific device.
9. In the plug-in dialog box, click **Start Scan** to scan the surfaces of the part.

Typical scanning session

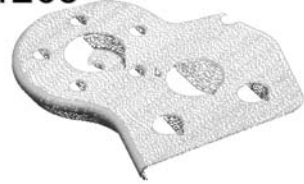
A scanning session involves scanning one or more passes with the same parameters.

All scan passes are added to the same Data object in the Tree View (**surface scan 1** in this example).

The number of acquired data points is displayed in the top left corner of the 3D scene.

surface scan 1

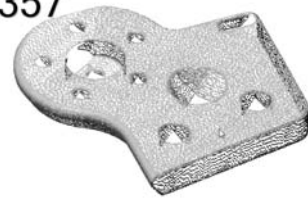
21265



Second scan pass.

surface scan 1

40357



After several scan passes.



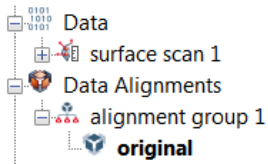
Remove unwanted data

To remove unwanted data points at any time while scanning, use the interactive element selection mode (choose *Select > Elements > Interactively*), then press DELETE. This can be useful, for example, in removing data from a fixture holding the part. See [Selecting elements](#) on page 63.

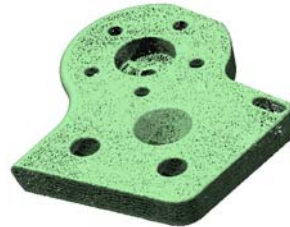


Results

A surface Data object is added to the Tree View. All scan passes are added to one Data object.



The point cloud is displayed in the 3D scene.



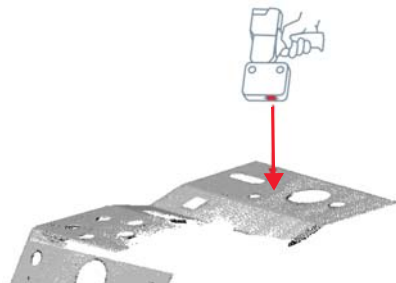
Best practices

Depending on the scanning device used, various settings and quality parameters may be available in the plug-in or in the manufacturer's software. In general, follow the manufacturer's recommendations to apply the optimal settings for the specific device and application. It is recommended, when available, to adjust the scanner intensity as necessary to capture high-quality measurements on a particular material.

When using a line scanner, the tips below also help maximize point cloud quality.

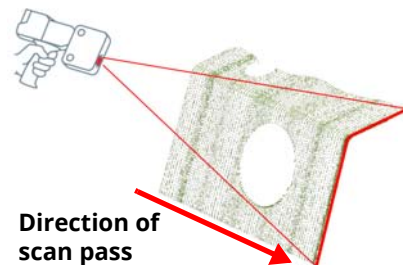
Scanner-to-surface angle

A surface is best captured when the viewpoint is normal to the surface.



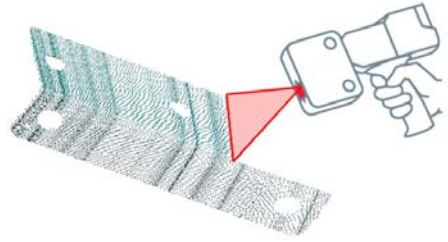
Orientation of line scan

To best capture part details with a line scanner, orient each scan pass so that it runs along edges. Aim directly at the edge detail. This allows more points to be captured along the edge, capturing small features better.



Scan passes

Focus on one simple scan pass at a time. Rather than scanning around the part at multiple angles in one pass, stop each pass after a straight pass, change the position or angle, and begin a new scan pass. This facilitates point cloud organization and point normal calculations.

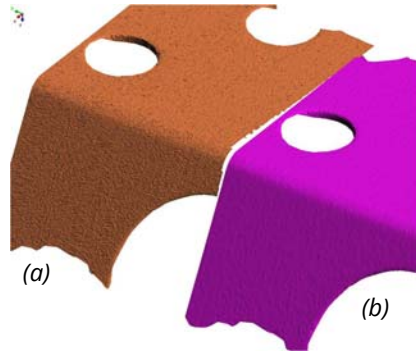


Filtering scan lines

Unnecessarily dense and noisy point clouds can be prevented by using the **Filter scan lines** point filtering option, activated by default, to remove data point outliers during scanning.

The example shown to the right represents the difference between unfiltered data points (a) and filtered data points (b).

The Filter scan lines option is found on the **Surface Scan > Filtering** page of the Line Scan Options dialog box.



Scanning surfaces and boundaries of thin parts

Surface and Boundary scanning combines the surface scanning behavior with the capability to delineate the boundaries of a part and of holes. Surface scanning captures data points across the part. Boundary scanning detects large gaps between surface points as part holes and saves the innermost data points as a boundary scan Data object. It also detects the end of scanned lines and saves the end points in the boundary scan Data object as well. These boundary points can then be used for data color maps.



Objective: Scan the surfaces of a part using a plug-in to obtain both a surface point cloud and a boundary point cloud, delineating internal and external boundaries.

Required: A line-scanning device is available and ready to use. The required plug-in is loaded in PolyWorks | Inspector.

Steps

1. Choose *Tools > Plug-ins > [device name]*.
The plug-in dialog box opens.
2. Select the **Surface and Boundary Scanning** scan type.
3. In the **Surface** list, select **(Create New)** to create a new surface Data object, or specify an existing surface Data object to which data will be added.
4. Specify the **Name** of the new surface Data object.
5. In the **Boundary** list, select **(Create New)** to create a new Data object, or specify an existing boundary Data object to which data will be added.
6. Specify the **Name** of the new boundary Data object.
7. Select **Real-time quality meshing**.
8. Select the **Polygonal model** as the final data type.
9. If desired, enable the **Clipping plane**.
10. Under **Scanning options**, select a **Profile** based on the size of the smallest details on the part.
11. Click **Start Scan**.



Data objects

Surface: (Create New) ▼
Name: surface scan 1

Boundary: (Create New) ▼
Name: boundary scan 1

Parameters

Real-time quality meshing
Final data type:
 Polygonal model
 Point cloud from mesh

Clipping plane
 Real-time comparison
 Test mode

Scanning options ⌵

Profile:
Standard Resolution ▼ ▼

Typical scanning session

A scanning session involves performing one or more scan passes with the same parameters.

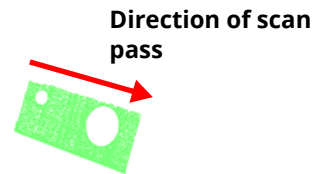
All scan passes are added to the same Data objects in the Tree View (**surface scan 1** and **boundary scan 1** in this example).

The number of acquired data points is displayed in the top left corner of the 3D scene.

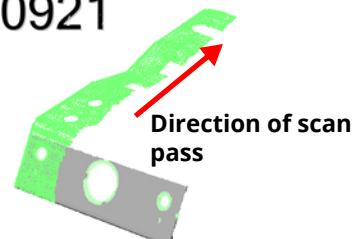
Second scan pass.

After several scan passes.

surface scan 1, boundary scan 1
5971



surface scan 1, boundary scan 1
70921

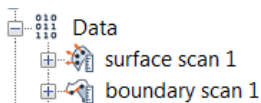


surface scan 1, boundary scan 1
327493

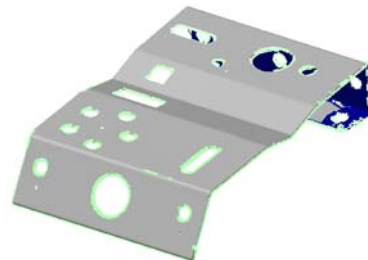


Results

A surface Data object and a boundary Data object are added to the Tree View, under the Data branch.



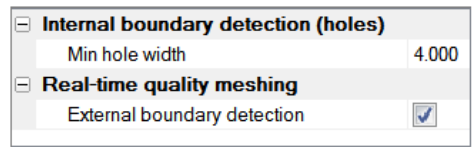
The polygonal model and point cloud are displayed in the 3D scene.



Notes

The **Surface and Boundary Scan** page of the Line Scan Options dialog box offers:

- an option to control the minimum hole width.
- an option to turn the external boundary detection on or off. It is enabled by default.



Scanning sharp edges of thin parts

The sharp edge scanning method is especially designed for sheet metal applications. It records data points by calculating the theoretical sharp edge of any hole or part boundary, which is obtained by extrapolating edge points from laser readings on the part surface and sidewall. Boundary data points can then be used, for example, for part boundary color maps.



Objective: Scan the surfaces of a part using a plug-in to obtain a boundary point cloud, delineating holes and part edges.

Required: A line-scanning device is available and ready to use. The required plug-in is loaded in PolyWorks|Inspector.

Steps

1. Choose *Tools > Plug-ins > [device name]*.

The plug-in dialog box opens.



2. Select the **Sharp Edge Scanning** scan type.



3. In the **Boundary** list, select **(Create New)** to create a new boundary Data object, or specify an existing boundary Data object to which data will be added.
4. Specify the **Name** of the new boundary Data object.
5. Under **Scanning options**, select a **Profile** based on the size of the smallest details on the part.

Boundary:	(Create New) ▼
Name:	boundary scan 1
Scanning options ⌵	
Profile:	Standard Resolution ▼ [Icon] ▼
Options	

Note that the **Sharp Edge Scan** page of the Line Scan Options dialog box offers an option to control the maximum point-to-point distance for points along a scan line as well as the maximum thickness of the part.

Scan line	
Max point-to-point distance	4.000
Maximum thickness	(Automatic) ▼

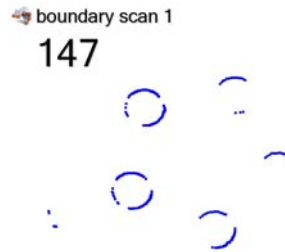
6. In the plug-in dialog box, click **Start Scan**.

Typical scanning session

A scanning session involves scanning one or more passes with the same parameters.

All scan passes are added to the same Data object in the Tree View (**boundary scan 1** in this example).

The number of acquired data points is displayed in the top left corner of the 3D scene.

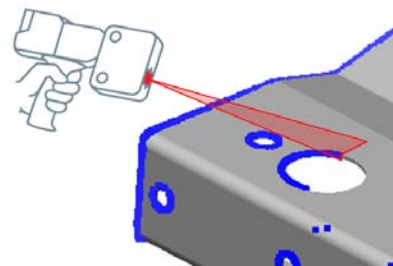


After several scan passes.

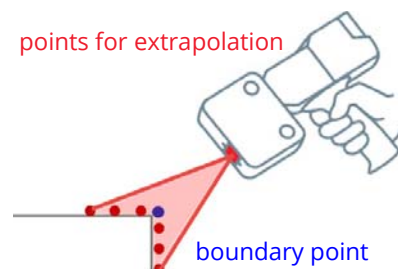


Since the data points are calculated by extrapolating between the part surface and sidewall, the scanner must be angled to adequately measure both surfaces.

The laser line should intersect the part edge at a 45 degree angle in order to gather data both on the material surface and the sidewall.



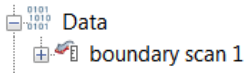
The part must be thick enough, and the scanner resolution small enough, so that at least three points can be measured on the part surface and sidewall.



Results

A boundary Data object is added to the Tree View. All scan passes are added to one Data object.

The boundary point cloud is displayed in the 3D scene.



Guidance while scanning features

When scanning, sometimes there is not enough scanned data to extract quality features or to even extract features at all.

Using guidance when scanning features guides the operator to acquire enough data, based on the feature coverage, to perform the extraction of the features downstream in the inspection process.

Guidance is available for all features whose measurement method is set to **Undefined** or **Extract**.

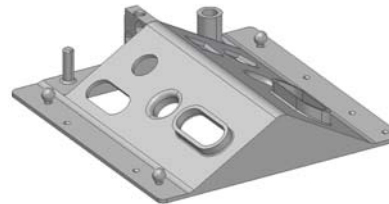
When set to **Extract**, the guidance takes the extraction parameters of each feature into account. When set to **Undefined**, the default extraction parameters are used.

An inspection workflow, using feature scanning guidance to ensure the extraction of features, is illustrated below.



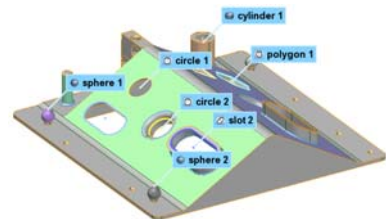
1. Import Reference object

Import the nominal information of the part (CAD model) into the inspection project.



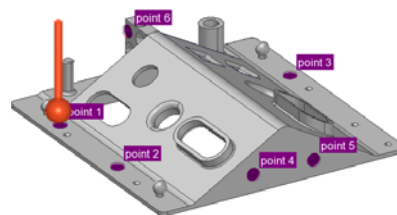
2. Create features

Create the nominal components of features and set their parameters before starting the inspection.



3. Align part to Reference object

Align the part to the CAD model to take advantage of the visual guidance tools.



4. Acquire Data object by scanning

Acquire point cloud or polygonal model data from the actual part needing inspection. Use the feature scanning guidance function to capture enough data and to ensure that the extraction of features is of the highest quality.



Objective: Use the guidance function when scanning features.

Required: The required plug-in is loaded in PolyWorks | Inspector. A line-scanning device is available, ready to use, and prealigned to the CAD model. The project contains nominal features.

Steps

1. Choose *Tools > Plug-ins > [device name]*.

The plug-in dialog box opens.



2. Select the **Surface Scanning** scan type.



3. In the **Surface** list, select **(Create New)** to create a new Data object, or specify an existing Data object to which data will be added.
4. Specify the **Name** of the new Data object.
5. Select **Real-time quality meshing**.
6. Select **Polygonal model** as the final data type.
7. If desired, enable the **Clipping plane**.
8. Under **Scanning options**, select a **Profile** based on the size of the smallest details on the part.
9. Click **Start Scan**.

Data object

Surface: (Create New) ▾

Name: surface scan 1

Parameters

Real-time quality meshing

Final data type:

Polygonal model

Point cloud from mesh


Clipping plane

Real-time comparison

Test mode

Scanning options ⌆

Profile:

Standard Resolution ▾  ▾

Options

The scanning toolbar appears in the 3D scene, providing quick access to feature scanning guidance.



10. Scan the part, making sure that the data acquired around the features is sufficient to obtain the measured component.

Results

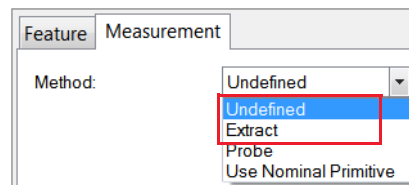
In the 3D scene, pins and guidance zones are displayed for existing features based on their extraction parameters. The pins and guidance zones disappear as soon as the extraction requirements are met.

A message in the 3D scene indicates the number of features left to scan.



Notes

- Guidance is offered for nominal features when the measurement method is set to **Undefined** or **Extract**.
- The line-scanning device can be aligned using the **Surface Points Alignment**.



Understanding the visual cues of the guidance function

- Orange pins indicate the location of features. Features are viewed from the front side (normal orientation).
- The orange highlight indicates where additional data is required in the form of a surface Data object (polygonal model or point cloud).
- The size and the location of the highlight are based on the extraction parameters set for each feature. If the measurement method is set to **Undefined**, the default extraction parameters are used.



- A gray pin under an arrow indicates the location of a feature. However, these features are viewed from the back side.

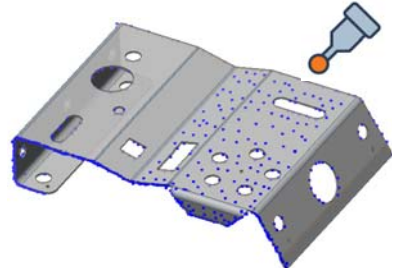


- The white highlight indicates where additional data is required in the form of a boundary Data object.



Probing a part using a probing device plug-in

PolyWorks|Inspector has the capability to use a probing device directly within its user interface to digitize parts. Probing devices include equipment such as laser trackers and articulated arms. Point clouds can be probed on the part surface, or along its boundaries. The point coordinates (x, y, z) and optionally, for arms, the normals (i, j, k) are stored in the current probing session in the Tree View, as either probed surface or probed boundary Data objects.



Using the probing device to collect surface points

A probing device can be used to collect a point cloud on the part. The data points are stored in the current probing session as a probed surface Data object containing the x, y, z point coordinates of each probed point. If the probing device is aligned to the Reference object, the points can be compensated toward the closest surface. If the probe tip has an orientation, that orientation can be used to assign point normals.

Objective: Probe the surfaces of a part using a plug-in to obtain a point cloud directly in PolyWorks|Inspector.

Required: A probing device is available and ready to use, and the required plug-in is loaded in PolyWorks|Inspector. A CAD model is loaded for point compensation, and the device is aligned to it.

Steps

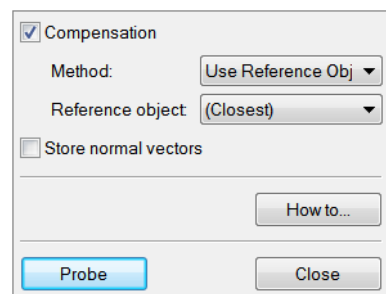
1. Choose *Tools > Data Objects > Probe Surface Point Clouds*.

The Probe Surface Point Clouds dialog box opens.

2. Select **Compensation** and specify the **Use Reference Object** method.

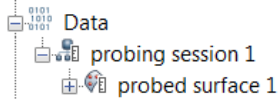
The probed points are translated by the probe radius toward the nearest Reference surface.

3. If it is possible to keep the probe tip orthogonal to the part, and the tip has an orientation (is not only a reflector), point normals can be assigned by selecting **Store normal vectors**.
4. Click **Probe** and acquire data points on the part.

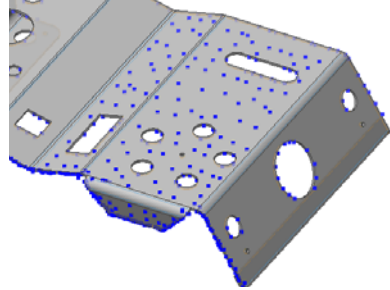


Results

A probed Data object is added to the Tree View. All probed points are added to one Data object.



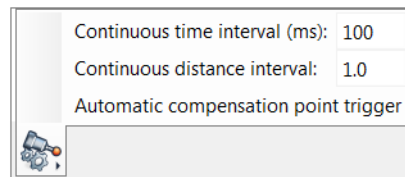
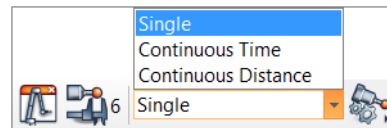
The point cloud is displayed in the 3D scene.



Notes

Many probing modes are available when probing, the most common being:

- The **Single** (or **Stationary**) mode collects one point each time the device is triggered.
- The **Continuous Time** mode collects a series of points when the device is triggered, at equal time increments.
- The **Continuous Distance** mode collects a series of points when the device is triggered, at equal distance increments.
- The increment values can be edited using the **Probing Options** button.



Using the probing device to collect boundary points

A probing device can be used to collect a point cloud on the part edges only, creating a boundary point cloud. This method can be used to measure the perimeter shape of a piece of sheet metal or a thin-walled part. Either the probe tip or shank can be used.

Objective: Probe the surfaces of a part using a plug-in to obtain a point cloud directly in PolyWorks|Inspector.

Required: A probing device is available and ready to use, and the required plug-in is loaded in PolyWorks|Inspector. A CAD file is loaded for point compensation, and the device is aligned to it.

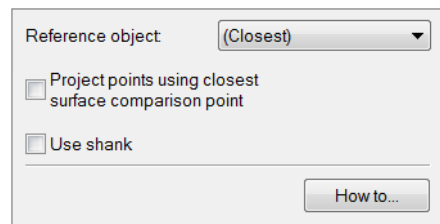
Steps

1. Choose *Tools > Data Objects > Probe Boundary Point Clouds*.

The Probe Boundary Point Clouds dialog box opens.

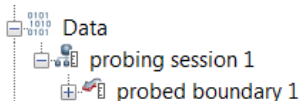


2. In the **Reference object** list, select **(Closest)** to use the closest Reference object, or specify an existing Reference object.
3. Select the **Project points using closest surface comparison point** parameter to offset the probed boundary data points by the measured deviation along the normal of the closest surface comparison point.
4. If the shank of the probing device is calibrated, the shank can be used for probing, rather than the probe tip, by selecting **Use shank**.
5. Click **Probe**.

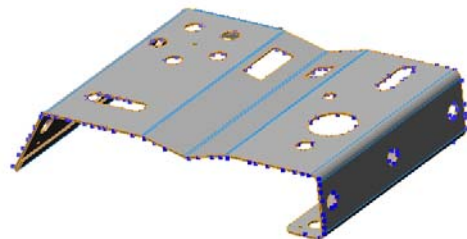


Results

A probed Data object is added to the Tree View. All probed points are added to one Data object.



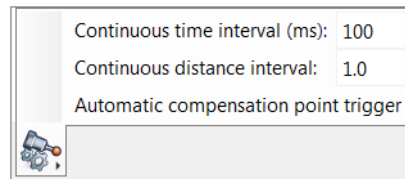
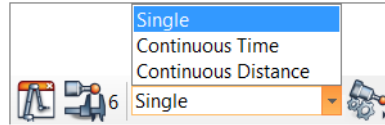
The probed boundary point cloud is displayed in the 3D scene.



Notes

Many probing modes are available when probing, the most common being:

- The **Single** (or **Stationary**) mode collects one point each time the device is triggered.
- The **Continuous Time** mode collects a series of points when the device is triggered, at equal time increments.
- The **Continuous Distance** mode collects a series of points when the device is triggered, at equal distance increments.
- The increment values can be edited using the **Probing Options** button.

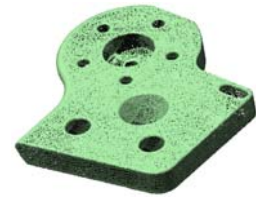


Importing surface and boundary point cloud Data objects

Data objects can be imported from point cloud or polygonal model files, or from other PolyWorks Metrology Suite projects such as another PolyWorks|Inspector project.

Importing surface Data objects

Surface Data object files contain a set of data points of the actual part, acquired by a digitizing device. Usually, the points cover the part surface but sometimes points are collected only around the hole and edge boundaries of the part. Most point cloud files can only be imported as Data objects.



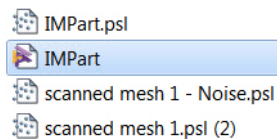
Importing IMAlign projects

Objective: Import the acquired data of an actual part as an IMAlign project.

Required: An IMAlign project.

Steps

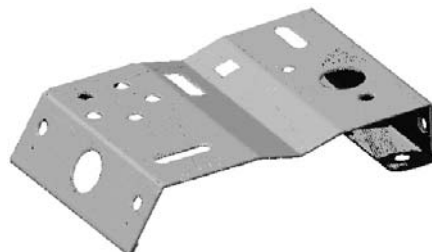
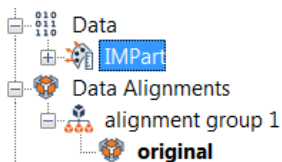
1. Choose *File > Import > Point Clouds*.
2. Browse for the IMAlign project that will be imported and click **Open**.



Results

The IMAlign project is added to the Tree View, under the Data branch.

The Data object is displayed in the 3D scene.



Meshing an IMAlign project

When the data cannot be acquired through a PolyWorks|Inspector digitizer plug-in, or when the data has previously been acquired through an IMAlign plug-in, it is possible to create a polygonal Data object from IMAlign projects.

Objective: Create a polygonal Data object from an IMAlign project.

Required: An open PolyWorks|Inspector project.

Steps

1. Choose *Tools > Data Objects > Create Polygonal Models*.

The Create Polygonal Data Objects dialog box opens.

2. Specify the **Name** of the new polygonal Data object.
3. In the **Method** list, select **From IMAlign Project**.
4. Browse for the IMAlign project.
5. Specify a **Sampling step**.

*The default value is set to **Automatic**, which means the sampling step is obtained from the IMAlign project.*

*If desired, specify a **Custom** sampling step.*

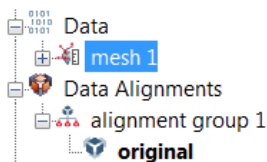
6. Click **Create**.



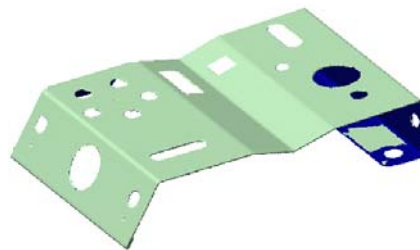
Name:	mesh 1
Creation method	
Method:	From IMAlign Project
Parameters	
Project:	Untitled (X:\Support_new\Proje ...)
Sampling step:	
<input checked="" type="radio"/> Automatic:	0.400
<input type="radio"/> Custom:	0.250

Results

A new polygonal Data object is added to the Tree View, under the Data branch.



The new polygonal Data object is displayed in the 3D scene.



Importing surface Data objects from files

Objective: Import the acquired data of the actual part in the form of a surface point cloud.

Required: A data file in a PolyWorks format (PSL or PIF files), a native format, or a neutral format (such as IGES or text files).

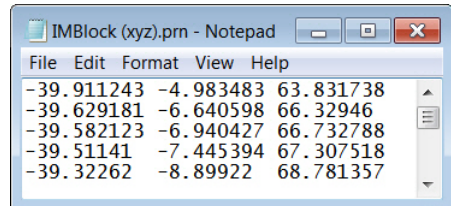
Steps

1. Choose *File > Import > Point Clouds*.



2. Browse for the file that will be imported.

To the right is an example of a text file containing x, y, and z coordinates for each data point.



The Import Parameters dialog box opens.

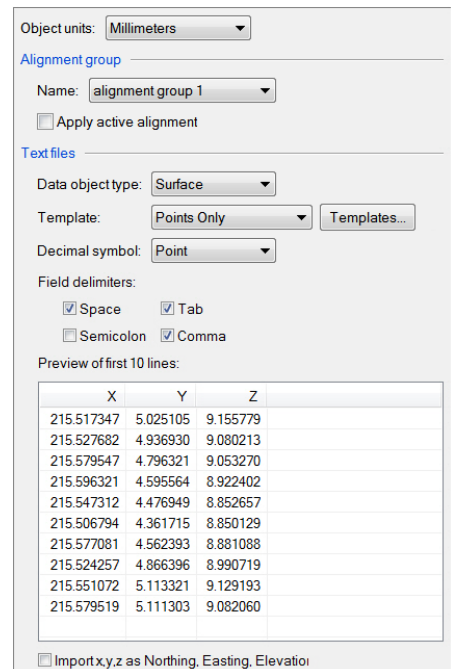
3. Specify the required parameters.

The parameters offered depend on the format of the imported file and the options set in the PolyWorks | Inspector Options (available by choosing *Tools > Options*). For example, for a *.txt* file:

- In the **Object units** list, select the native units of the file.
- In the **Data object type** list, select **Surface**.
- Specify the decimal symbol and field delimiters of the file.

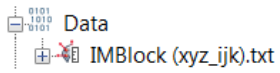
A preview of how the file is interpreted is shown in the form of a grid.

4. Click **OK**.

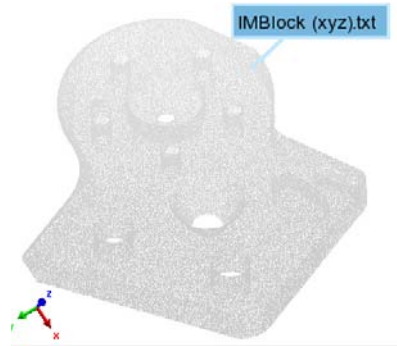


Results

The point cloud is added to the Tree View, under the Data branch, as a surface Data object.



The object is displayed in the 3D scene.



Importing boundary Data objects

Objective: Import the acquired data of the actual part edges in the form of a boundary point cloud.

Required: A data file in a neutral format (such as IGES or text files).

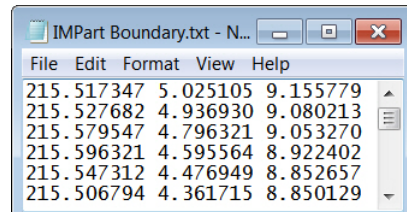
Steps

1. Choose *File > Import > Point Clouds*.



2. Browse for the file that will be imported.

To the right is an example of a text file containing x, y, and z coordinates for each data point.



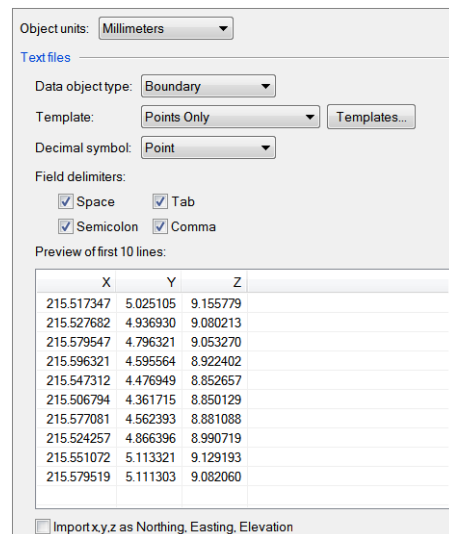
The Import Parameters dialog box opens.

3. Specify the required parameters.

The parameters offered depend on the format of the imported file, and the options set in the PolyWorks | Inspector Options (available by choosing *Tools > Options*). For example, for a .txt file:

- In the **Object units** list, select the native units of the file.
- In the **Data object type** list, select **Boundary**.
- Specify the decimal symbol and field delimiters of the file.

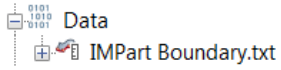
A preview of how the file is interpreted is shown in the form of a grid.



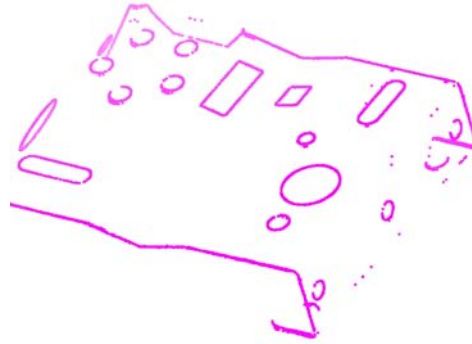
4. Click **OK**.

Results

The Data point cloud is added to the Tree View, under the Data branch, as a boundary Data object.



The object is displayed in the 3D scene.



Importing polygonal models

A polygonal model is a mesh composed of triangles and vertices obtained from data points scanned on the actual part. Polygonal models are generally imported as Data objects, but can also be imported as Reference objects.



Objective: Import the acquired data of the actual part in the form of a polygonal model.

Required: A polygonal model file in a PolyWorks format (such as POL or PQK), or a neutral format (such as STL or OBJ).

Steps

1. Choose *File > Import > Polygonal Models*.

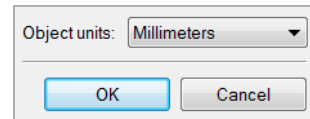


2. Browse for the file that will be imported.

The Import Parameters dialog box opens.

3. In the **Object units** list, select the native units of the file.

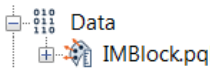
By default, polygonal models are imported as Data objects. The default behavior can be changed in the PolyWorks | Inspector Options (**Objects > Reference/Data > Polygonal Models** page). The PolyWorks | Inspector Options are available by choosing *Tools > Options*.



4. Click **OK**.

Results

The polygonal model is added to the Tree View, under the Data branch, as a surface Data object.



The polygonal model is displayed in the 3D scene.



Editing Data objects

It may sometimes be useful to clean up Data objects by removing unwanted data points or by dividing the dataset into multiple objects. For example, data points collected on a part fixture or inspection fixture can be removed from the overall dataset, or particular regions of a part can be split from the original Data object into a new Data object. Another common operation consists in adding more data to an existing scanned Data object.

Cleaning scan data

It is possible to select and delete individual data points from a Data object. This can be done interactively, in the 3D scene, or by using criteria such as the proximity to reference surfaces. Data points deleted in the project can always be recovered, if necessary.



Removing data points interactively

Objective: Remove data points from a Data object by manually selecting and deleting them.

Required: The project contains a Data object.

Steps

1. Choose *Select > Elements > Interactively*.

Alternatively, press SPACEBAR to enable the interactive element selection mode.

The selection mode is activated.



2. Use the interactive selection tool to select the data points that will be removed.

To select point cloud data points, see [Selecting elements](#) on page 63.

3. Press DELETE.

Alternatively, choose Edit > Delete > Elements to delete the selected data points.



Results

The selected points are removed from the Data object and are not used for analysis.



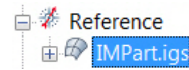
Removing data points using a CAD model

Objective: Remove selected data points from the Data object. The data points are selected using a Reference object (CAD model).

Required: A Reference object and a Data object are available in the project. In addition, the Data object is aligned to the Reference object.

Steps

1. Select the Reference object in the Tree View.



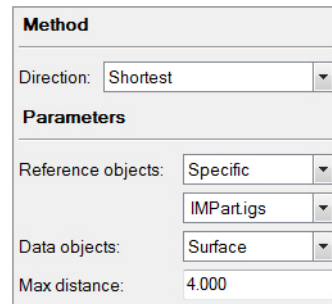
2. Choose *Select > Data Points > Using Reference Object Surfaces*.

The Select Data Points using Surfaces dialog box opens.

3. In the **Direction** list, select **Shortest**.

Since a Reference object has been preselected, **Specific** is displayed in the **Reference objects** list as well as the Reference object name.

It is possible to change the Reference object for another one, or choose **All** in the **Reference objects** list to use all Reference objects.



4. Specify the **Max distance** that will be considered between data points and reference surfaces.
5. Click the **Select** button to select the data points.

The selected data points are highlighted red in the 3D scene.



6. Select the Data object in the Tree View.
7. Right-click and choose *Select > Elements > Invert* to select only data points that do not meet the selection criteria above. New selected data points are highlighted red.

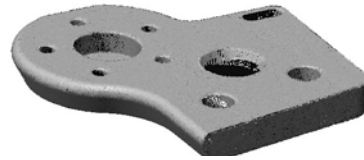


8. Choose *Edit > Delete > Elements* to delete the selected data points.

Alternatively, press DELETE.

Results

The selected points are removed from the Data object and are not used for analysis.



Recovering deleted data points

When data points are deleted, they are not deleted permanently. It is possible to recover deleted data points at any time, which makes deleting and editing operations safer and reversible.

Objective: Recover deleted data points.

Required: The project contains a Data object. Data points have been deleted from that object.

Steps

1. Select the Data object in the Tree View.
2. Choose *Edit > Recover Deleted Elements*.

Results

Previously deleted data points will be added to the Data object. All deleted data points are recovered, not just the last data points deleted.

Notes

Recovering deleted data points can be a long operation due to the automatic project update.

Purging deleted data points

It is possible to purge deleted data points for surface Data objects of a piece in order to reduce RAM requirements and accelerate software operations, as well as to reduce project size on disk.

Objective: Purge deleted data points.

Required: The project contains a surface Data object. Data points have been deleted from that object.

Steps

1. Choose *Edit > Purge Deleted Elements*.

Results

Previously deleted data points are permanently deleted.

Notes

- Only surface Data objects (polygonal models or point clouds) are supported by this tool. This tool does not support the following Data objects:
 - Using real-time quality meshing; polygonal models and point clouds from mesh
 - Boundary point clouds
 - Probed point clouds
- The *Edit > Recover Deleted Elements* command cannot be used to recover data points that have been purged.

Adding data to a scanned Data object

A common editing operation for Data objects consists in adding more data to an existing scanned Data object.

Objective: Add data to an existing Data object acquired using a scanning plug-in.

Required: A Data object that was scanned directly in PolyWorks| Inspector and a scanning device is ready to use and is aligned to the Data object that will be modified.

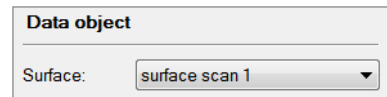
Steps

1. Choose *Tools > Plug-ins > [device name]*.

The plug-in dialog box opens.



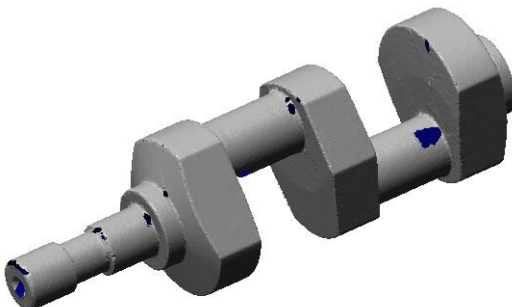
2. In the **Surface** list, select the Data object to which data will be added.



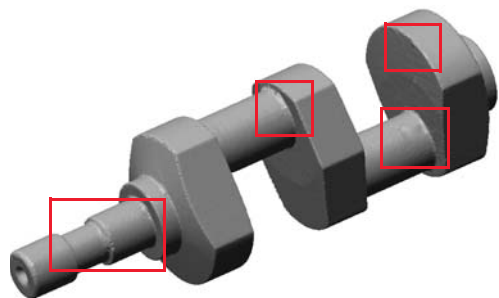
3. Click **Start Scan** and scan the required areas.
4. Click **End Scan** to end the scanning session.
5. If meshing point cloud data in real time, click **Finalize Mesh** to complete the polygonal Data object.

Results

The Data object before adding new data.



The Data object after acquiring new data. Holes have been covered with new data.



Splitting a Data object into two objects

Selected data points can be moved or copied from one Data object to a new object using the Split command. This is useful in separating scans containing multiple parts. Selection can be done interactively in the 3D scene, or using criteria such as the proximity to Reference surfaces.



Objective: Move data points to a new Data object.

Required: The project contains a Data object.

Steps

1. Select data points, using one of these methods for example:
 - The interactive element selection tool. See [Selecting elements](#) on page 63.
 - Using a Reference object; choose *Select > Data Points > Using Reference Object Surfaces*.

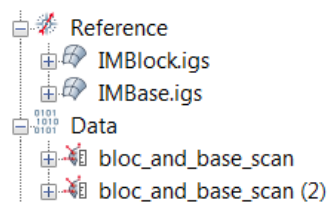


2. Choose *Edit > Split Objects > Move Elements* to remove the selected data points from the current Data object and create a new Data object containing only those data points.

Choose Edit > Split Objects > Copy Elements to preserve the data points in the original Data object.

Results

A new Data object is added to the Tree View, under the Data branch, containing only the selected data points.

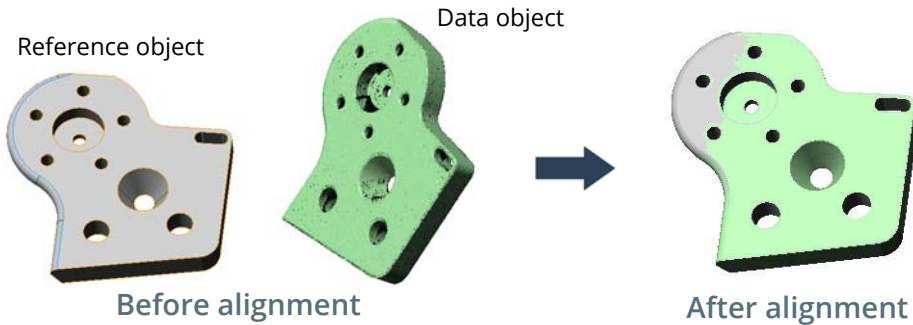


Aligning Data objects to Reference objects



Introducing alignments

An alignment is an object transformation operation that changes the position and orientation of a Data object to bring it into the coordinate system of a Reference object.



PolyWorks | Inspector offers a variety of alignment tools to meet a wide variety of scenarios. Each alignment is represented in the Tree View as a data alignment.

Alignments are available on the *Align* menu or on the **Alignments** toolbar.



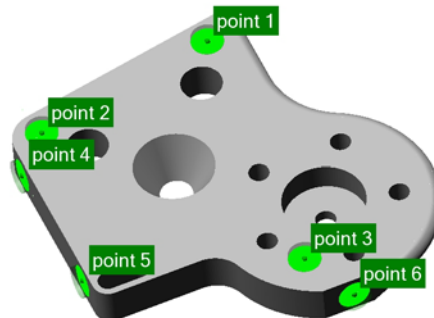
Alignment tools

Here is a list of common tools offered by PolyWorks | Inspector that are used to align Data objects to Reference objects:



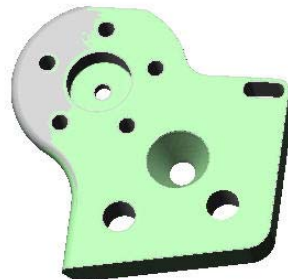
Surface Points

Match source points probed on physical objects to matching destination points picked on Reference objects or specified numerically.



Best-Fit Data to Reference Objects

Best-fit the surface of selected Data objects to the surface of available Reference objects.





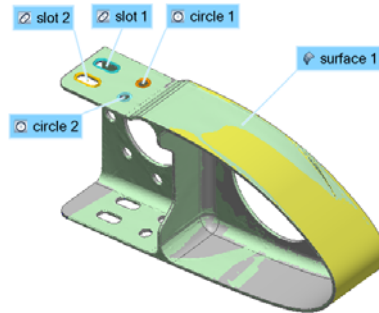
Best-Fit Data to Data Objects

Aligns Data objects to fixed Data objects. This tool is useful when Data objects are acquired in multiple device positions that are not aligned to each other.



Best-Fit Measurement Objects

Align a Data object to a Reference object by matching the points of point-reducible measurement objects (features and comparison points) and surface features to the matched nominal component.

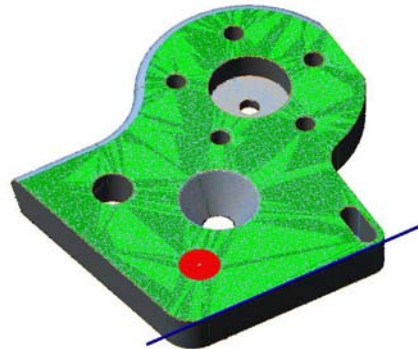


Plane, Axis, Center Point and



Perpendicular Planes

Align Data objects to Reference objects using pairs of planar, direction-based, and center-point-based features (Plane, Axis, Center Point), or by matching three plane features (Perpendicular Planes).





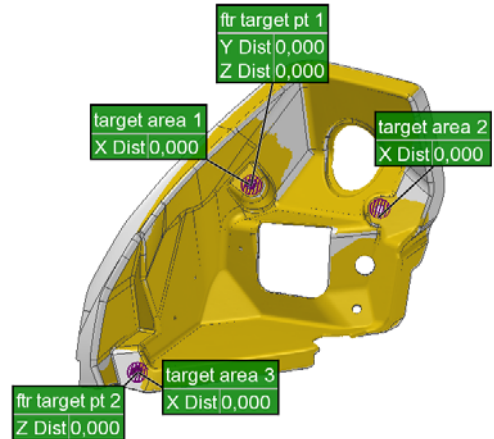
Reference Targets

and



Reference Targets by Probing

Match components extracted from Data objects, or points obtained on Data objects, to reference targets created on Reference objects.



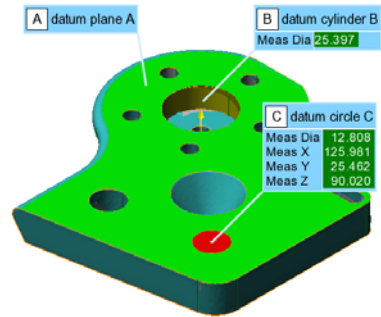
Datum Reference Frame

and



Datum Reference Frame by Probing

Align Data objects (using the measured components of datum features) to Reference objects (using the nominal components of datum features).

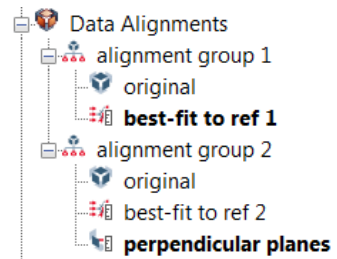


Introducing data alignments

Each transformation of a Data object is automatically recorded in the Tree View as a data alignment. While the latest data alignment is usually the active data alignment, a previous data alignment can be activated manually. This may be useful in examining inspection results that use a variety of alignments.

Each **data alignment** is stored chronologically within an **alignment group**. When created, the data alignment is called **original**. Data alignments within an alignment group form an alignment history. The name of each data alignment states the type of alignment performed.

The active data alignment, often the last one, is shown in bold. Any data alignment can be activated, and project contents are updated automatically. Unnecessary data alignments can be deleted.



Data objects with the same transformations can be contained within the same alignment group, whereas Data objects requiring different alignments are automatically placed in separate alignment groups. The latter case is common for assemblies of multiple Data objects. An alignment performed on any Data object applies to all the Data objects.

The creation of alignment groups and data alignments is automatically managed by PolyWorks | Inspector.

Activating a data alignment

An alignment group can contain multiple data alignments, such as a Best-Fit alignment followed by a Feature-Based alignment. It is possible to activate any alignment in an alignment group and results are automatically updated.

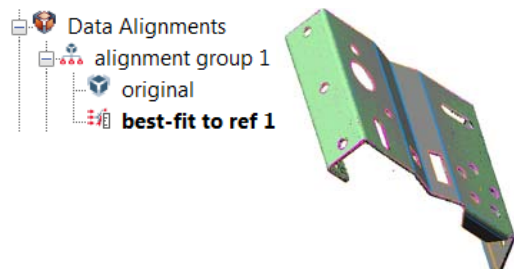
Objective: Activate a previous data alignment in an alignment group.

Required: A Data object with at least two data alignments in an alignment group.

Steps

Here, there are two data alignments: *original* and *best-fit to ref 1*.

The *best-fit to ref 1* alignment is currently the active alignment and is shown in bold.



1. Select a data alignment in the Tree View and choose *Align > Data Alignments > Activate*.

Alternatively, right-click the selection and choose **Activate**.

Measurement objects and results are updated automatically.

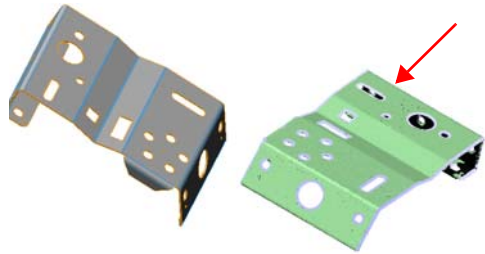


Results

The selected data alignment is activated and shown in bold in the Tree View.



The Data objects are transformed by the new active data alignment.



Notes

When a data alignment is performed incorrectly, it is best to delete it immediately. First, activate the previous data alignment, then select the last data alignment in the alignment group, and choose *Edit > Delete > Objects*. Deleting an intermediate data alignment may have an effect on subsequent data alignments.



Aligning using object surfaces

An alignment is the operation that brings a Data object into the coordinate system of a Reference object. The Best-Fit operation is a surface-based alignment tool that iteratively transforms the position and orientation of a Data object to minimize the deviations of the data points with respect to the surface of a Reference object. The Data object must first be prealigned to the Reference object.



Best-fitting Data objects to Reference objects using an automatic prealignment

For the **Automatic** prealignment function to work well, the Data object must cover most of the Reference object, and the Reference object must not have a symmetrical form.

Objective: Align a Data object to a Reference object using a Best-Fit alignment with an Automatic prealignment.

Required: The project contains a Data object and a Reference object.

Steps

1. Choose *Align > Best-Fit Data Objects > Data to Reference Objects*.




The Best-Fit Data to Reference Objects dialog box opens.

2. In the **Method** list, click **Fit to Reference Object Surfaces**.
3. The **Prealignment** parameter is enabled by default and set to **Automatic**.
4. In the **Reference objects** list, select **All** to consider all the Reference objects.
5. In the **Data objects** list, select **Surface** to consider all the surface Data objects.

Specific considers specific objects.

6. Specify the **Max distance** used to match data points to the surface of the Reference object.

*If the Data object is highly deviated from the Reference object, increase the **Max distance** to match data points to the surface of the Reference object.*

Name:	best-fit to ref 1
Method:	
Parameters ⌵	
<input checked="" type="checkbox"/> Prealignment:	Automatic ⌵
Reference objects:	All ⌵
Data objects:	Surface ⌵
Max distance:	4.000
Subsampling:	1/1 ⌵
Advanced ⌵	
Constraints ⌵	
Statistics ⌶	
Iteration:	<input type="text"/>
Convergence:	<input type="text"/>

7. In the **Subsampling** list, select **1/1** to refine the alignment, but with a longer computing time.

By default, the alignment operation uses one point out of every four points to speed up calculations.

8. Click **Start**.

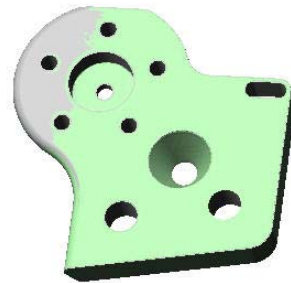
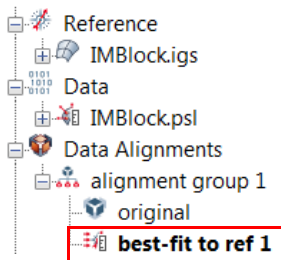
*As the alignment converges to a solution, statistics are displayed in the **Statistics** section.*

Statistics	
Iteration:	13
Convergence:	0.000000839485019

Results

The new data alignment is added to the Data alignment group of the Data object.


The Data object is aligned to the Reference object using an automatic prealignment and is then best-fitted using object surfaces.



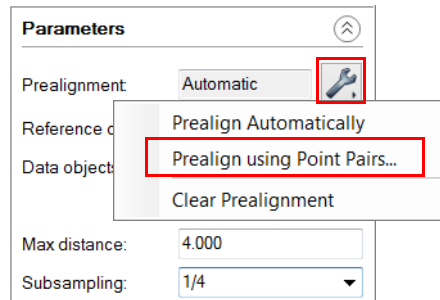
Notes

It is possible to edit a data alignment using its property sheet. Select the object in the Tree View and choose *Edit > Object Properties*, and then change the parameters.

For example, edit the objects used, the **Max distance**, or the **Subsampling** factor to refine the alignment result. Then, click **Apply** and click **Close**.

Parameters	
Prealignment:	Automatic 
Reference objects:	All
Data objects:	Specific
	IMBlock.pol
Max distance:	4.000
Subsampling:	1/4

If the alignment fails, a manual prealignment may be necessary. In the object properties, click the **Set Prealignment** button and choose **Prealign using Point Pairs**. Click **Close**. See the next section for instructions on using point pairs as a prealignment method.



Best-fitting Data objects to Reference objects using a manual prealignment

If a part is partially scanned, or shaped differently from the Reference object, the automatic prealignment tool may not position the Data object in the optimal position for best-fitting. The Point Pairs manual prealignment tool (1 Point Pair and N Point Pairs methods) should be used instead.

Objective: Align a Data object to a Reference object using a Best-Fit alignment with a manual prealignment.

Required: The project contains a Data object and a Reference object.

Steps

1. Choose *Align > Best-Fit Data Objects > Data to Reference Objects*.

The Best-Fit Data to Reference Objects dialog box opens.



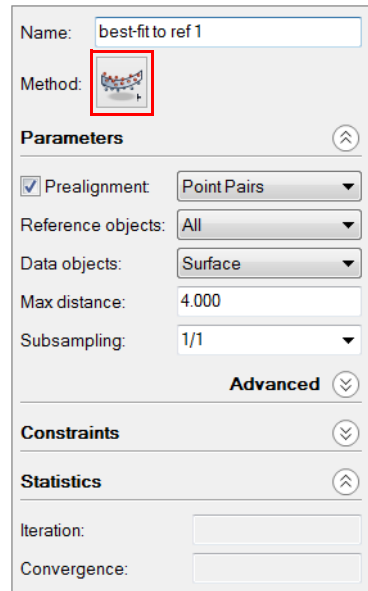
2. In the **Method** list, click **Fit to Reference Object Surfaces**.
3. In the **Prealignment** list, select **Point Pairs**.
4. In the **Reference objects** list, select **All** to consider all the Reference objects.
5. In the **Data objects** list, select **Surface** to consider all the surface Data objects.

***Specific** considers specific objects.*

6. Specify the **Max distance** used to match data points to the surface of the Reference object.
*If the Data object is highly deviated from the Reference object, increase the **Max distance** to match data points to the surface of the Reference object.*
7. In the **Subsampling** list, select **1/1** to refine the alignment, but with a longer computing time.
By default, the alignment operation uses one point out of every four points to speed up calculations.
8. Click **Start**.

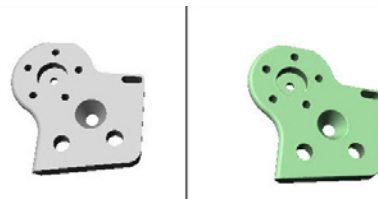
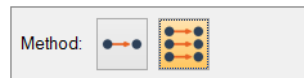
The Prealign using Point Pairs dialog box opens.

The Reference object and the Data object are displayed in separate viewports.

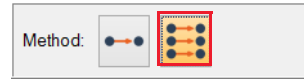


9. Move both the Reference object (left) and the Data object (right) to give them a similar orientation. This facilitates the picking of pairs of points in similar areas.

Just moving the objects does not operate any transformation of the objects. If the mode is exited before picking points and aligning, the objects are still in their original position.



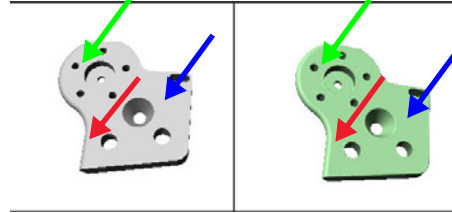
10. Click **N Point Pairs**.



11. Click **Pick**.

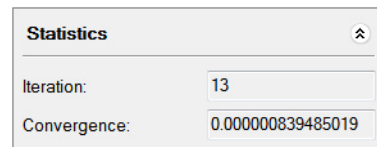
12. Anchor matching points on both objects, using the same order. A minimum of three pairs of points is required.

The points are displayed using the same color and the same number in subscript.



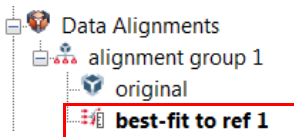
13. Click **Align** to confirm the alignment and exit the picking mode.

*The Best-Fit alignment is automatically performed after the manual prealignment. As the alignment converges to a solution, statistics are displayed in the **Statistics** section.*

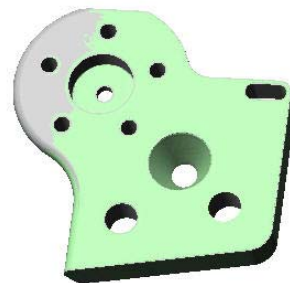


Results

The new data alignment is added to the data alignment group of the Data object.



The Data object is aligned to the Reference object using a manual prealignment and is then best-fitted using object surfaces.



Notes

In the Prealign using Point Pairs dialog box, another method is available: the **1 Point Pair** method. This method requires picking only one pair of points. A Best-Fit alignment is automatically performed after a successful prealignment.

Aligning by probing surface points

The **Surface Points** alignment method is used to align probed points to the same locations on Reference objects. This prealignment tool greatly facilitates visualization and guidance for subsequent operations.

Objective: Align by probing surface points.

Required: A Reference object, a physical part, and a probing device.

Steps

1. Choose *Align > Surface Points*.

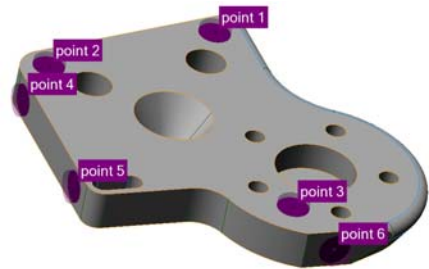
The Create Surface Points dialog box opens.



2. In the **Method** list, specify **Anchor**.
3. Click **Anchor**.
4. Anchor six points on the Reference object that will be used for the alignment.

All six degrees of freedom should be constrained using a 3-2-1 alignment method, where the alignment direction of three surface points is along one reference axis (X axis), the alignment direction of two other surface points is along a second reference axis (Y axis), and the alignment direction of the last surface point is along the remaining reference axis (Z axis). If all the degrees of freedom are not constrained, then only a final N Point Pairs alignment is performed.

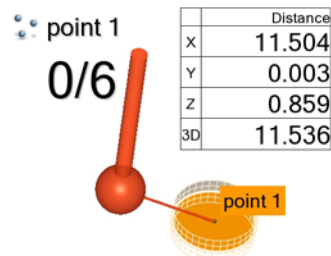
The anchored points are listed in the Align Surface Points pane. A radial tolerance determines how closely the probed point must match the reference location.



No.	Name	Direction	Tol Radial	Dev Radial	Test Radial
1	point 1	Y	5.000		
2	point 2	Y	5.000		
3	point 3	Y	5.000		
4	point 4	X	5.000		
5	point 5	X	5.000		
6	point 6	Z	5.000		

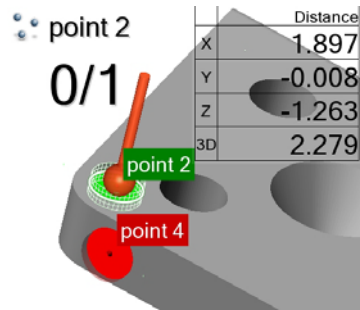
5. If a probing device is connected, the probing dialog box opens automatically. Otherwise, click **Probe Source Points** in the pane.
6. On the physical part, probe the same six points using the same order.

As the points are probed, the probing device is more closely aligned to the Reference object providing a better visual context.



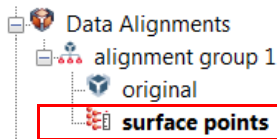
- If the probed points are not precise enough to fall within the radial tolerance of the reference location, a message box offers the possibility of reprobing the points that are out of tolerance.

Visual and audio guidance are provided to remeasure those points. The points can be reprobbed in any order.

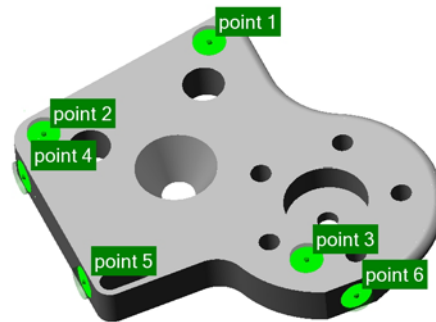


Results

The new data alignment is added to the alignment group of the Data object.



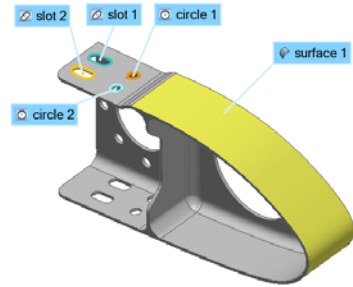
The probed data points are aligned to the same locations on the Reference object.



Best-fitting measurement objects

The Best-Fit Measurement Objects alignment method is used to improve workflows in the aerospace and automotive industries. This method aligns the measured component of a measurement object to the corresponding nominal component.

The following table indicates the possible alignment techniques based on the types of measurement objects used.



Measurement objects	Alignment technique
Point-reducible features: Circles, slots, ellipses, spheres, arcs duplicated in 3D, polygons, rectangles, points	Aligns objects by matching the center point of a measured component to the center point of the corresponding nominal component.
Surface features Plane features Cylinder features Cone features	Aligns objects by matching the measured data points to the corresponding nominal component.
Pattern features	Adds all features of the pattern to the alignment pane and aligns the objects based on the children feature types.
Comparison points	Aligns the data points to the nominal component along the surface normal (surface comparison points) or along the edge vector (trimmed edge or hemmed edge comparison points).

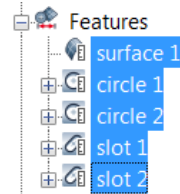
Point-reducible features can also be paired. A number of combinations are possible. They can be paired with another point-reducible feature, a line feature, a cylinder axis, a cone axis, or a plane feature.

Objective: Align a Data object to a Reference object using a Best-Fit Measurement Objects alignment.

Required: The Data object should be prealigned to the Reference object. The objects used for the alignment must contain nominal and measured components.

Steps

1. In the Tree View, select the objects that will be used for the alignment.



2. Choose *Align > Best-Fit Measurement Objects*.

The Best-Fit Measurement Objects dialog box opens, as well as the Best-Fit Measurement Objects - Setup and Results pane.

The selected objects are automatically added in the pane.



3. In the pane, specify the alignment direction (**X, Y, Z**) for each object.

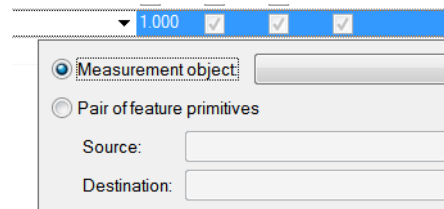
Object	Use X	Use Y	Use Z	Dev	Max Dev
surface 1	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	-----	-----
circle 1	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	-----	-----
circle 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	-----	-----
slot 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	-----	-----
slot 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	-----	-----

4. If necessary, click **Add Objects** and choose **Add**.

An empty line is added, and the edit dialog opens automatically.



5. In the **Measurement object** list, choose the object that will be added.



6. Click elsewhere in the interface.

Alternatively, drag and drop measured measurement objects from the Tree View into the pane to add them to the alignment.

In the Best-Fit Measurement Objects dialog box:

The **Automatic prealignment** is enabled by default. It allows roughly aligning measured geometry to nominal geometry. The alignment will continue to refine the match until the **Max number of iterations** is reached, or when all objects move less than the **Max displacement target** between iterations.

7. Specify the **Max number of iterations**.
8. Specify the **Max displacement target**.
9. Click **Start**.

*As the alignment converges to a solution, statistics are displayed in the **Statistics** section.*

Parameters	
<input checked="" type="checkbox"/> Automatic prealignment	
Max number of iterations:	500
Max displacement target:	0,000100

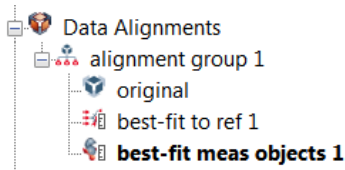
Statistics	
Iteration:	9
Max displacement:	0.000

As long as the dialog box is not closed, it is possible to make modifications to the alignment parameters and objects. The results are updated automatically.

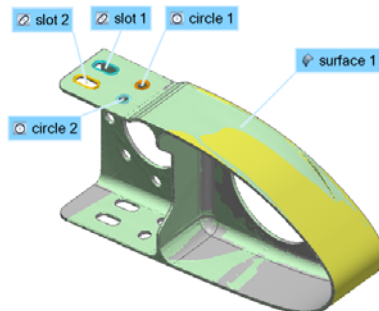
Once the dialog box is closed, the alignment can be edited by right-clicking the alignment in the Tree View, and choosing **Edit Alignment** (see [Editing the alignment parameters](#) on page 146).

Results

The new data alignment is added to the alignment group of the Data object.



The Data object is aligned to the Reference object using the specified measurement objects.



Statistics for each object are displayed in the pane.

Object	Use X	Use Y	Use Z	Weight	Dev	Min Dev	Max Dev
surface 1	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	-0.203	-0.597	0.367
circle 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	0.028		
circle 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	0.133		
slot 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.000	0.032		
slot 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.000	0.009		

Notes

- Surface features with non compensated points can be used in this alignment.
- Point-reducible cross-section features can be used in this alignment if they are duplicated in 3D.
- In the **More** section, the **Scale to minimize 3D deviations** parameter enables the scaling of all available Data objects and the measured components in order to minimize the 3D deviations between the nominal components and the measured components used for alignment. This is useful in compensating for temperature variations when measuring Data objects that span over 10 meters.

Understanding the Best-Fit Measurement Objects - Setup and Results pane

The Best-Fit Measurement Objects pane is used to specify the objects that will be used for the alignment and to set the alignment parameters. It features a vertical toolbar and a table area to configure the alignment.

Column header

Table area

Vertical toolbar

Object	Use X	Use Y	Use Z	Weight	Dev	Min Dev	Max Dev
surface 1	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	-0.203	-0.597	0.367
circle 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	0.028		
circle 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	0.133		
slot 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.000	0.032		
slot 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.000	0.009		

Column header

Names of the different columns displayed in the pane.

Vertical toolbar



Add Objects

Menu button that allows adding objects to the alignment or adding objects already selected in the Tree View to the alignment.



Edit Objects

Menu button that allows modifying the **Use axes**, **Weight**, and **Use to compute** scaling factor parameters of the selected objects.



Remove Objects

Removes the selected object from the alignment.



Automatic Update

The alignment automatically updates when a change is made in the pane. This menu button allows disabling the automatic update. It also allows manually launching the update of the alignment.



Report

Allows creating a report table of the alignment.

Table area

The table area presents the list of objects used for the alignment, their alignment directions and their statistics (once the alignment is created).

Note

It is possible to customize the pane. For example, it is possible to show more columns and to reorder them.

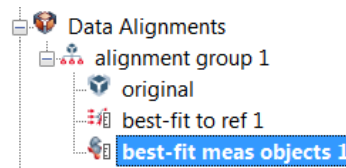
Editing the alignment parameters

Objective: Edit the Best-Fit Measurement Objects alignment.

Required: The project contains a Best-Fit Measurement Objects alignment.

Steps

1. In the Tree View, select the data alignment.



- Right-click and choose *Edit Alignment*.

The Best-Fit Measurement Objects dialog box and the Best-Fit Measurement Objects - Setup and Results pane open.

- Make the necessary modifications, such as modifying the axes used for the alignment and the weight of an object.
- Click **Apply**.

Results

The results are updated automatically and displayed in the Best-Fit Measurement Objects - Setup and Results pane.

Before

Object	Use X	Use Y	Use Z	Weight	Dev	Min Dev	Max Dev
surface 1	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	-0.203	-0.597	0.367
circle 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	0.028		
circle 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	0.133		
slot 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.000	0.032		
slot 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.000	0.009		

After

Object	Use X	Use Y	Use Z	Weight	Dev	Min Dev	Max Dev
surface 1	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	100.000	-0.100	-0.533	0.202
circle 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	0.519		
circle 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	0.629		
slot 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.000	0.071		
slot 2	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.000	0.047		

Changing the objects used for the alignment

It is possible to change the objects used for the alignment. Objects can be deleted, added, or changed.

In the Best-Fit Measurement Objects - Setup and Results pane:

- Select an object and remove it using the **Remove Objects** button.



- Add an object using the **Add Objects** button. Alternatively, drag and drop measured objects from the Tree View into the pane.
- Replace an object by clicking the edit shortcut arrow that corresponds to the object to be modified.



Object	Use X	Use Y	Use Z	Dev
circle-src 1	✓	✓	✓	0.026

Measurement object circle-src 1

Pair of feature primitives

Source:

Destination:

Changing the pairing of feature components

By default, this alignment method matches measured components to their nominal components. It is possible to customize the pairing and align a measured component to another measured component. This is especially useful for virtual assemblies of parts.

Steps

In the Best-Fit Measurement Objects - Setup and Results pane:

1. Click the edit shortcut arrow adjacent to the object to be modified.

The contextual dialog box opens.

Object	Use X	Use Y	Use Z	Dev
circle-src 1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0.026

Measurement object: circle-src 1

 Pair of feature primitives

 Source:

 Destination:

2. Select **Pair of feature primitives**.
3. In the **Source** list, specify the source component.
4. In the **Destination** list, specify the destination component.

Measurement object: circle-src 1

 Pair of feature primitives

 Source: circle-src 1 - meas-

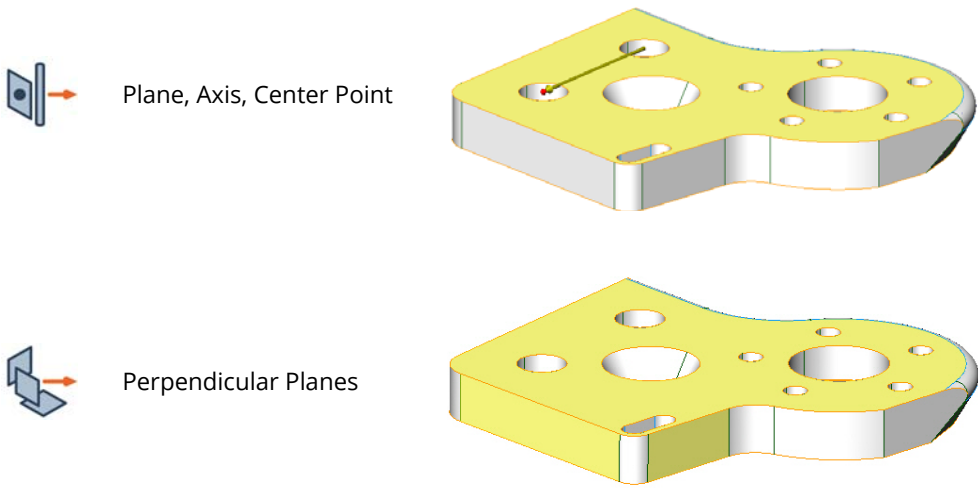
 Destination: circle-dest2 - meas-

Source components are moved to destination components.

Aligning with features

This approach consists in creating features whose nominal primitives are created on the Reference object, and whose measured primitives are created on the Data object. Then, the measured primitives are aligned to the nominal primitives, transforming the position and the in-space orientation of the Data object.

There are several feature-based alignment methods:



Alternatively, the **Plane, Axis, Center Point** and the **Perpendicular Planes** methods can also align the Data object to standard axes. Measured primitives created on the Data object are aligned to standard planes, standard axes, and a coordinate along the axes.

Distinct characteristics of planar features, direction-based features, and center-point-based features allow using them for alignment methods:

- **Planar features:** planes, circles, arcs, ellipses, polygons, rectangles, slots, and slabs.
- **Direction-based features:** circles, arcs, cones, cylinders, ellipses, polygons, rectangles, slots, and lines, but not planes.
- **Center-point-based features:** circles, arcs, ellipses, points, polygons, rectangles, slots, and spheres. They are also called point-reducible features.

Aligning using a plane-axis-point approach

The **Plane, Axis, Center Point** alignment method is used to align Data objects to Reference objects using pairs of planar, direction-based, and center-point-based features. Generally, the measured component of a feature is assigned as the source object while its nominal component is assigned as the destination object.

Objective: Align a Data object to a Reference object using a Plane, Axis, Center Point alignment.

Required: The project contains a Data object and a Reference object, as well as a planar feature, a direction-based feature (except planes), and a center-point-based feature, each containing a nominal and a measured component.

Steps

1. Choose *Align > Feature-Based > Plane, Axis, Center Point*.



The Align Plane, Axis, Center Point dialog box opens.







2. In the **Sequence** list, select **Plane, Axis, Center Point**.

3. Specify a planar feature, a direction-based feature, and a center-point-based feature that will be used for the alignment.

*The **Source** object is the measured feature component, while the **Destination** is the nominal feature component. The measured components are moved to the nominal locations.*

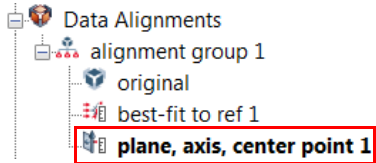
*Click the **Pick** button to the right of each list box to pick the source and destination objects directly in the 3D scene.*

4. Click **Confirm**.

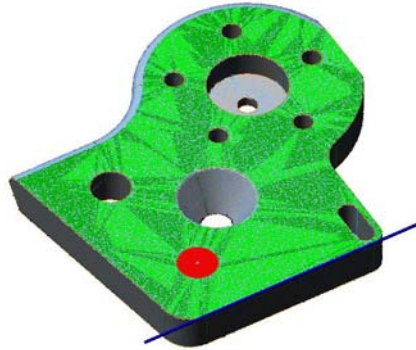
Name:	plane, axis, center point 1	
Sequence:	Plane, Axis, Center Point ▼	
Parameters		
Plane		
Source:	plane 1 -meas- ▼	
Destination:	plane 1 -nom- ▼	
Axis		
Source:	line 1 -meas- ▼	
Destination:	line 1 -nom- ▼	
Center point		
Source:	circle 1 -meas- ▼	
Destination:	circle 1 -nom- ▼	

Results

The new data alignment is added to the alignment group of the Data object.



The Data object is aligned to the Reference object using the features.



Notes

- If no nominal feature component has been defined, standard system planes, axes, and coordinates can be specified in the **Destination** list box.
- Sometimes, a partial alignment can be performed. If only one or two features are used for the alignment, the remaining parameters can be left blank. Make sure that the **Sequence** chosen begins with the type of feature used for the partial alignment.

Aligning using perpendicular planes

The **Perpendicular Planes** alignment method aligns the Data object to the Reference object by matching three plane features. The planes must intersect each other in space.

Objective: Align a Data object to a Reference object using a Perpendicular Planes alignment.

Required: A Reference object, a Data object and three perpendicular plane features with nominal and measured components.



Steps

1. Choose *Align > Feature-Based > Perpendicular Planes*.



The Align Perpendicular Planes dialog box opens.

2. In the lists, select the planes that will be used for the alignment. The measured components should be in the **Source** column.

Source		Weight
1st plane:	plane 1 -meas	1.000
2nd plane:	plane 2 -meas	1.000
3rd plane:	plane 3 -meas	1.000

*When a measured component is selected in the **Source** column, the corresponding nominal component is automatically specified in the **Destination** list. The user can select another component in the **Destination** list, if desired.*

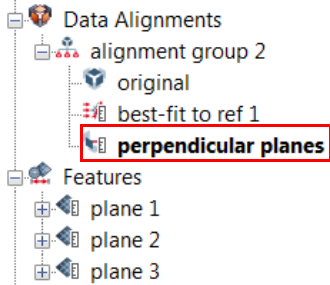
The order in which planes are specified is important. The first plane is the primary plane and has precedence over the second and third planes when aligning.

Destination			Normal	Position
<input checked="" type="radio"/>	plane 1 -nom	<input type="radio"/>	+X	0.000
<input checked="" type="radio"/>	plane 2 -nom	<input type="radio"/>	+Y	0.000
<input checked="" type="radio"/>	plane 3 -nom	<input type="radio"/>	+Z	0.000

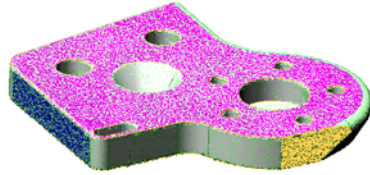
3. Click **Align**, then click **Done**.

Results

The new data alignment is added to the alignment group of the Data object.

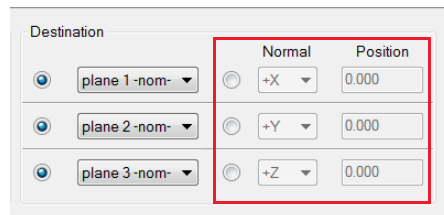


The Data object is aligned to the Reference object using three perpendicular planes.



Notes

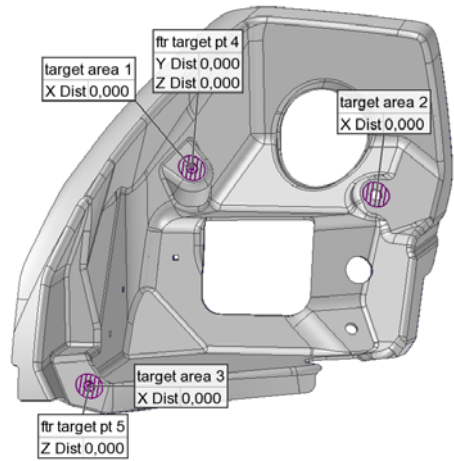
When no **Destination** components exist, the alignment can be performed using planes obtained from standard axes and positions.



Aligning with reference targets

Reference targets are points, lines, or areas that can be used to constrain an alignment along a specified direction. They are commonly found in sheet metal part drawings as well as mold and die drawings, in which specific coordinates on the part are established and directions along the standard axes are defined to align the part.

Typically, reference targets are created on the Reference object. Once they are in the desired position, measurements for the reference targets are obtained and a reference target alignment can be performed.

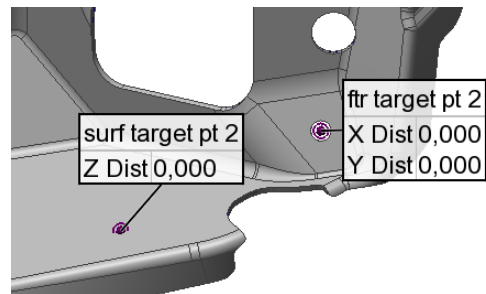


Introducing reference targets

There are three types of reference targets: points, lines, and areas.

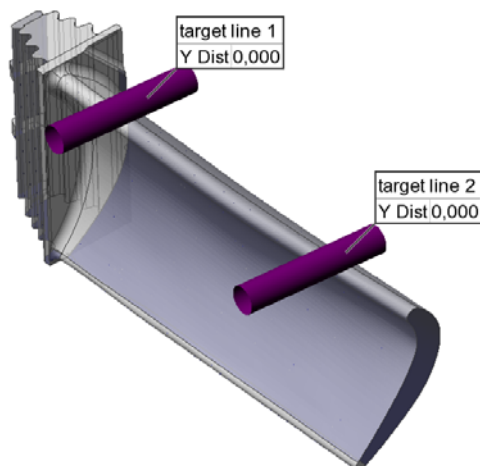
Reference target points

They are positioned at precise coordinates, and align the Data object along the alignment directions.



Reference target lines

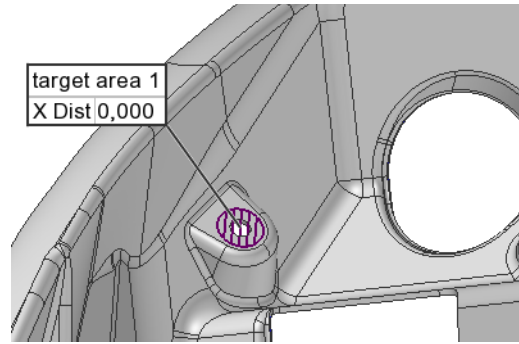
They are created from line features, and constrain the alignment along a pin or shank edge.





Reference target areas

They align the Data object according to the specified area on the Reference object. Two types can be created: circular and rectangular.

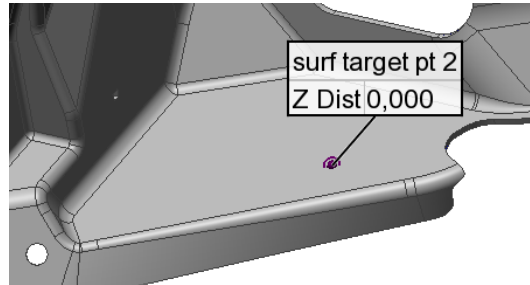


There are three subtypes of reference target points: surface, feature, and cross-section.



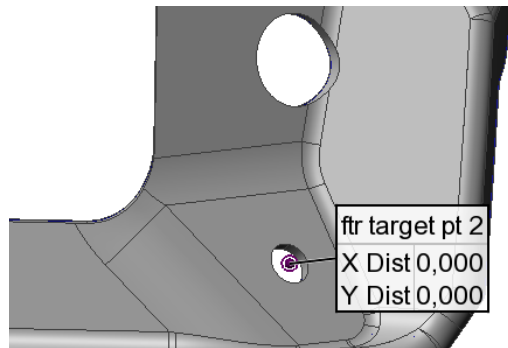
Surface reference target points

They are positioned at precise coordinates on Reference object surfaces. When performing the alignment, the distance from the measured point on the Data object to the target is minimized according to the specified alignment directions.



Feature reference target points

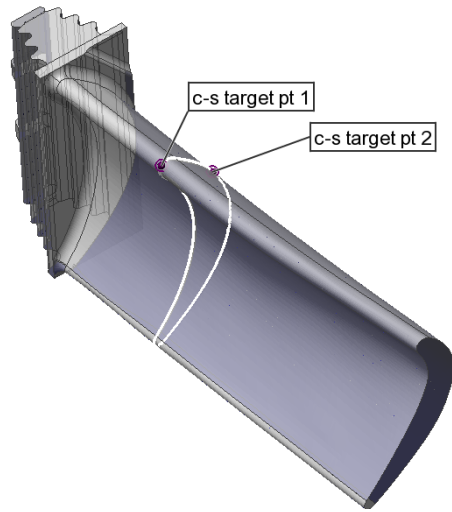
They are created from precise coordinates of the Reference object, and are associated with individual features, such as a circle, a slot, or a plane. When performing the alignment, the distance to the feature is minimized according to the specified alignment directions.





Cross-section reference target points

They are positioned at precise coordinates on Reference object surfaces, and are associated with cross-sections. When performing the alignment, the distance from a measured point on the cross-section to the target is minimized.



Creating reference target areas

Objective: Create reference target areas using specific coordinates on the part.

Required: A Data object, a Reference object, and coordinates for target locations.

Steps

1. Choose *Measure > Reference Targets > Create > Areas*.

The Create Reference Target Areas dialog box opens.

2. In the **Method** list, select **Numerically** to specify coordinates.

By default, the reference target area is circular. However, it is also possible to create a rectangular reference target area.

3. Use the coordinates provided on the part drawing to define the **Center** of the reference target area.

The target can be projected onto the Reference object surface along an axis. Instead of entering the third coordinate, select **Project** and choose the projection axis.

4. Set the **Axis orientation** of the reference target area. If the **Automatic** option is enabled, the axis orientation of the area is calculated automatically by projecting the center point of the reference target area onto the Reference object at the nearest location. The normal of the Reference object is then used to determine the axis orientation.

5. Set the **Diameter** of the reference target area.

6. Set the **Alignment direction** of the target point. It can be set to:

- **Automatic:** The standard axis that is closest to the axis orientation of the reference target area is used to determine the alignment direction.
- **Axis orientation:** The axis orientation of the reference target area is used to determine the alignment direction.
- One of the standard axis: **+X, -X, +Y, -Y, +Z** or **-Z**.

7. Click **Create**.



Creation method

Method: Numerically

Parameters

Shape: Circle Rectangle

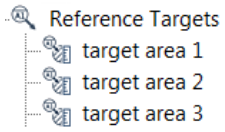
Center		Axis orientation	
X:	531.823	I:	0.000
Y:	-810.392	J:	0.000
Z:	546.366	K:	1.000
<input type="checkbox"/> Project		<input checked="" type="checkbox"/> Automatic	
Z			

Diameter: 35,000

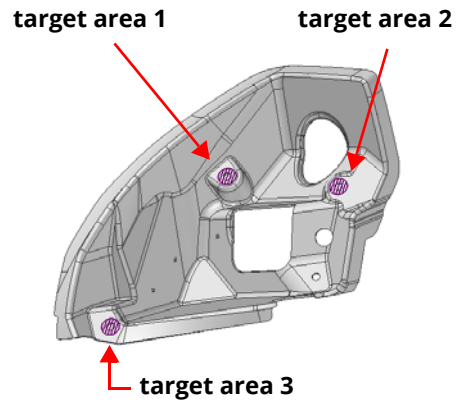
Alignment direction: Automatic

Results

The new reference target areas are added to the Tree View, under the Reference Targets branch.



The reference target areas are displayed in the 3D scene as lined plane areas.



Creating reference target points

Creating surface reference target points

Objective: Create surface reference target points using specific coordinates on the part.

Required: A Data object, a Reference object, and coordinates for target locations.

Steps

1. Choose *Measure > Reference Targets > Create > Surface Points*.

The Create Surface Reference Target Points dialog box opens.



2. In the **Method** list, select **Numerically** to specify coordinates.
3. Use the coordinates provided on the part drawing to define the target **Point** location.

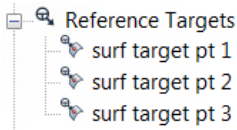
The target can be projected onto the Reference object surface along an axis. Instead of entering the third coordinate, select **Project** and choose the projection axis.

4. Set the **Normal** direction of the point. If the **Automatic** option is enabled, the normal direction of the surface point is calculated automatically from the surface normal of the Reference object.
5. Set the **Alignment direction** of the target point. It can be set to **Automatic**, **Point Normal**, **+X**, **-X**, **+Y**, **-Y**, **+Z**, **-Z**, or **XYZ** direction.
6. Click **Create**.

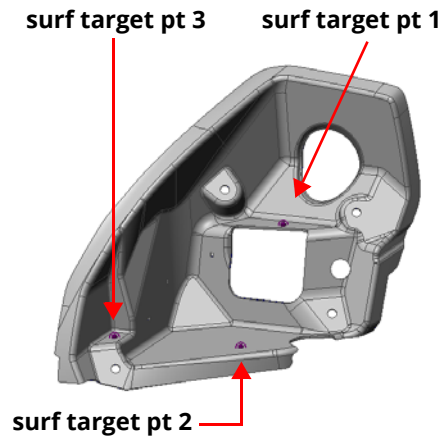
 A screenshot of the 'Creation method' dialog box. The 'Method' dropdown is set to 'Numerically'. Under the 'Parameters' section, there are two columns: 'Point' and 'Normal'. The 'Point' column has input fields for x (0.000), y (0.000), and z (0.000), a 'Project' checkbox (unchecked), and a dropdown menu currently showing 'Z'. The 'Normal' column has input fields for i (0.000), j (0.000), and k (1.000), and an 'Automatic' checkbox (checked). At the bottom, the 'Alignment direction' dropdown is set to 'Automatic'.

Results

The new surface reference targets are added to the Tree View, under the Reference Targets branch.



The reference targets are displayed in the 3D scene.



Notes

The surface reference targets can be set as datum targets and be used as part of a Datum Reference Frame of a GD&T control. For more information on datum targets, see [Defining datum targets](#) on page 511.

Creating feature reference target points

Objective: Create feature reference target points using features on the part.

Required: A Data object and a Reference object. The part must have measurable features.

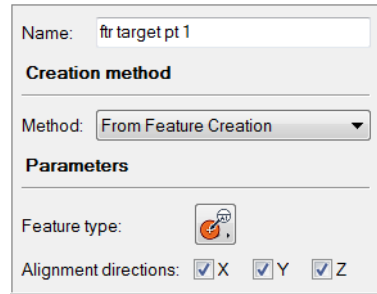
Steps

1. Choose *Measure > Reference Targets > Create > Feature Points*.

The Create Feature Reference Target Points dialog box opens.

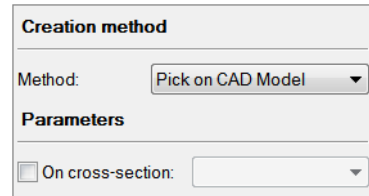


- In the **Method** list, select **From Feature Creation** to create a new feature for the reference target.
- Choose a **Feature type**, such as a circle feature.
- Specify the **Alignment directions** to be constrained by the feature.



It can be any combination of X, Y, and Z.

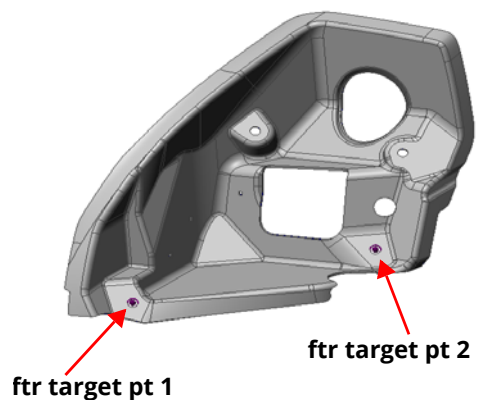
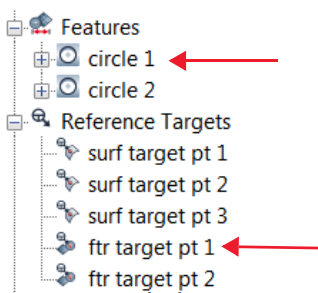
- Click **Create**.
The feature creation dialog box opens.
- In the **Method** list, select a feature creation method such as **Pick on CAD Model**.
- Click **Pick**.
- In the 3D scene, pick the feature on the CAD model.



Result

The new feature reference targets and their associated features are added to the Tree View.

The feature reference targets are displayed in the 3D scene.



Measuring reference targets

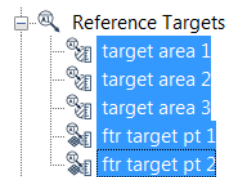
Reference targets can be extracted from scanned data. The Data object must be closely aligned to the Reference object for the reference target measurement to be successful.

Objective: Measure the reference targets by extracting them.

Required: The project contains a Data object, a Reference object, and reference targets (feature/surface reference target points or reference target areas). The Data object must be prealigned to the Reference object.

Steps

1. Select the reference targets in the Tree View.



2. Choose *Measure > Extract Measured*.

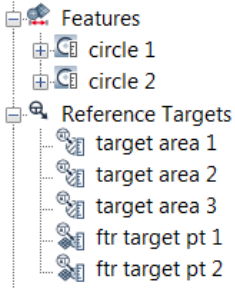
Note the following:

- Feature reference targets are measured by measuring their associated features.
- Reference target areas and surface reference targets are measured, but they will also be remeasured throughout the alignment operation. See [Aligning using reference targets](#) on page 165.

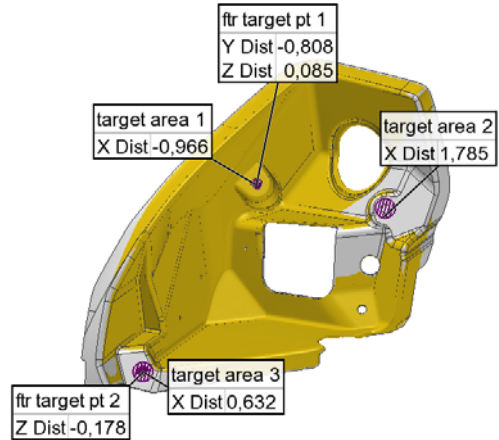


Results

The reference targets are measured. For feature reference targets, the corresponding features are measured as well.



The reference targets show the current deviation between the Data object and the Reference object in the specified alignment directions.



Notes

- Feature reference targets can also be measured by probing the associated features.
- It is also possible to measure surface reference target points, reference target lines, and reference target areas using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Aligning using reference targets

Reference targets can be used in a reference target alignment. A combination of reference targets can be used for the alignment of Data objects.

Two alignment methods are available, which are explained below:

- **Align** (for reference targets measured using the **Extract** measurement method);
- **Align by Probing** (for reference targets measured using the **Probe** measurement method). This method is not available for reference target lines and reference target areas.

Reference targets can be used as datum targets for use in Datum Reference Frame alignments (see [Defining datum targets](#) on page 511).

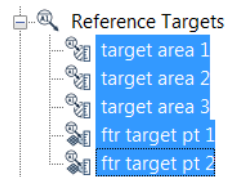
Align using reference targets

Objective: Align the Data object to the Reference object using reference targets.

Required: The project contains a Data object, a Reference object and reference targets. The Data object must be prealigned to the Reference object.

Steps

1. Select the reference targets in the Tree View.

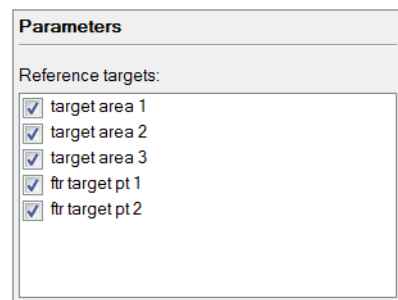


2. Choose *Align > Reference Targets > Align*.

The Align Reference Targets dialog box opens.



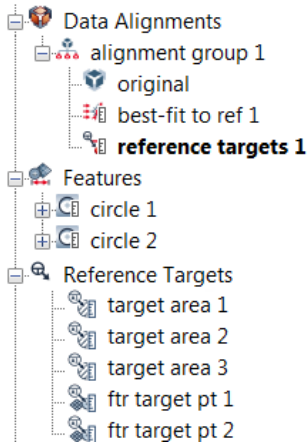
3. The preselected reference targets are automatically selected in the **Reference targets** area.
4. Select or clear reference targets as desired.
Only selected reference targets are used for the alignment.
5. Click **Start**.



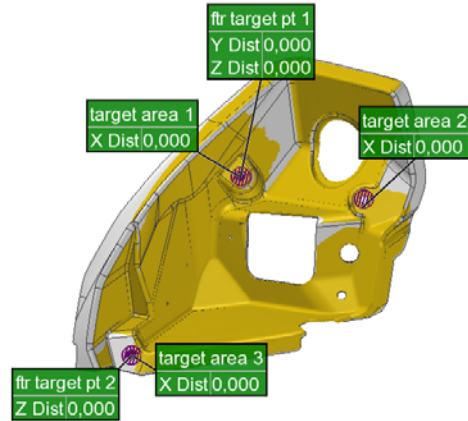
If there are unmeasured targets, they will be measured before the alignment starts. As for reference target areas and surface reference targets, they will be remeasured throughout the alignment operation.

Results

The new data alignment is added to the alignment group of the Data object.



The Data object is aligned to the Reference object using the reference targets.



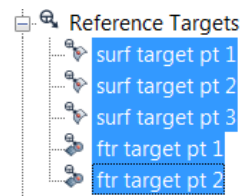
Align using surface and feature reference targets by probing

Objective: Align the Data object to the Reference object by probing using reference targets.

Required: A Reference object, a physical part, surface or feature reference target points, and a probing device.

Steps

1. In the Tree View, select the reference target points.



2. Choose *Align > Reference Targets > Align by Probing*.

The Align Reference Targets by Probing pane opens.



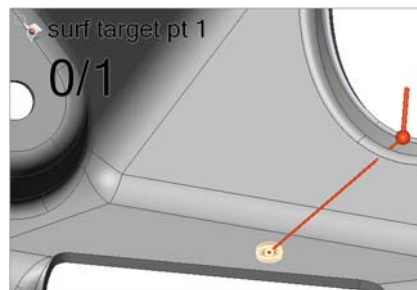
- The selected reference target points are listed in the pane.

Align Reference Targets by Probing			
Name	Feature Type	Feature Name	Tol Rad...
surf target pt 1			5.000
surf target pt 2			5.000
surf target pt 3			5.000
ftt target pt 1	Circle	circle 2	
ftt target pt 2	Circle	circle 1	

- Click the **Probe Measured** button.

Name	Tol Rad...	Dev Radial	TestRadial	Measured
	5.000			
	5.000			

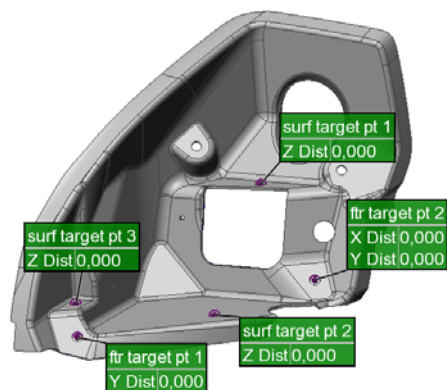
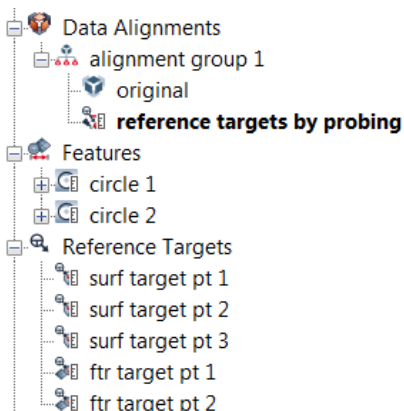
- Probe the surface reference target points and features associated to the feature reference target points, as guided.



Results

The new data alignment is added to the Tree View. The reference targets and the features associated to the feature reference targets are measured.

The Data object is aligned to the Reference object using reference target points.

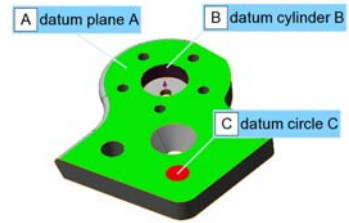


Notes

- For point features, a radial tolerance is used; it is possible to reprobe points that are out of the radial tolerance.
- The feature type can be specified in the Align Reference Targets by Probing pane if not already linked to a feature.

Aligning with Datum Reference Frames

A Datum Reference Frame (DRF) is a reference that serves to orient and locate objects in space. A DRF can be composed of datum features with nominal and measured primitives. The DRF alignment aligns Data objects (using the measured primitives) to Reference objects (using the nominal primitives). It can also be composed of datum targets. A DRF used by a Geometric Dimensioning and Tolerancing (GD&T) control serves to orient and locate tolerance zones.



Align using a Datum Reference Frame

Objective: Align the Data object to the Reference object using a DRF.

Required: The PolyWorks | Inspector project contains a Data object and a Reference object, as well as datum features and/or datum targets, each containing a nominal and a measured component.

Steps

1. Choose *Align > Datum Reference Frame > Align*.

The Align Datum Reference Frame dialog box opens.



2. Choose an existing DRF from the list.

DRFs used in any GD&T controls are listed automatically.

3. To add a new DRF, under **Creation**, select datum features using the lists.

4. Click **Create**.

The new DRF is added to the **Datum reference frames** list.

5. Select a DRF from the list, then click **Align**.

Datum reference frames:

A
A D B

Align
Delete

Creation ^

Datum feature 1: A

Datum feature 2: B

Datum feature 3: C

Create

*To be used in a DRF, a feature must have a datum feature label. The datum feature label can be specified on the **Feature** tab of a feature's property sheet.*

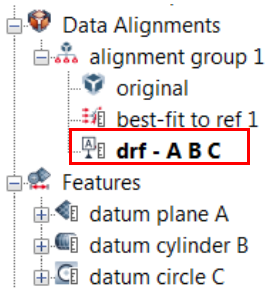
Feature	Measurement	Display
Datum feature label: A		

To be used in a DRF, a reference target must have a datum target label. The datum target label can be specified on the **Reference target** tab of a reference target's property sheet.

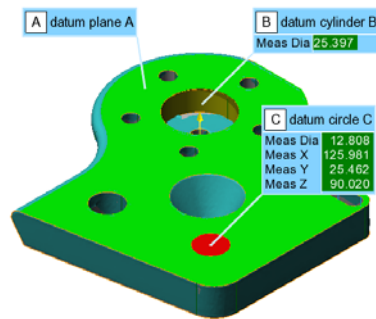
Reference target	Measurement
Datum target label:	A 1

Results

The new data alignment is added to the alignment group of the Data object.



The Data object is aligned to the Reference object using the datum features.



Notes

When datum features are specified in a GD&T control, the Datum Reference Frame alignment is applied virtually for the calculation – the Data object does not need to be actually aligned to the Reference object. However, the alignment used by a GD&T control can be visualized using the Feature Navigator tool.

Align a Datum Reference Frame by probing

Objective: Align the Data object to the Reference object by probing the datum features and the datum targets linked to a Datum Reference Frame.

Required: The PolyWorks | Inspector project contains a Data object and a Reference object, as well as datum features and/or datum targets.

Steps

1. Choose *Align > Datum Reference Frame > Align by Probing*.



The Align Datum Reference Frame by Probing pane opens.

2. Choose an existing DRF from the drop-down list.

DRFs used in any GD&T controls are listed automatically.

3. To add a new DRF, click on the **Create Datum Reference Frame** button located next to the drop-down list.
4. The involved datum features and datum targets are listed in the pane.
5. Click the **Probe Measured** button.

Align Datum Reference Frame by Probing

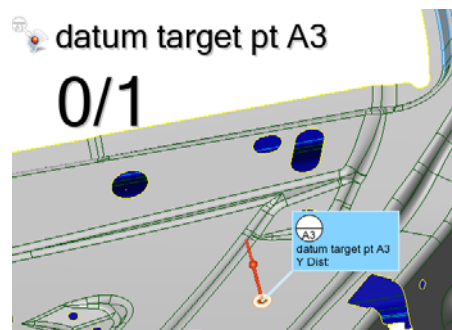
Datum reference frame: ABC +

Datum Label	Name	Type
A	datum target pt A1	Surface Reference
A	datum target pt A2	Surface Reference
A	datum target pt A3	Surface Reference
B	datum circle B	Circle
C	datum slot C	Slot

Feature Name	Tol Radial	Dev Radial	Test Result
le 1			
le 2			
le 1			

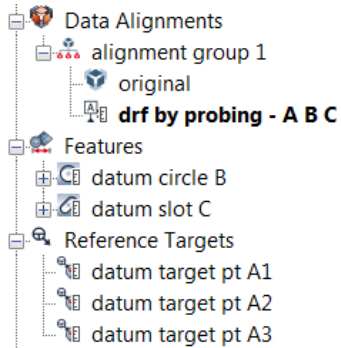
6. Probe the features and targets, as guided.

The order in which the datums are probed is based on their order in the Datum Reference Frame.

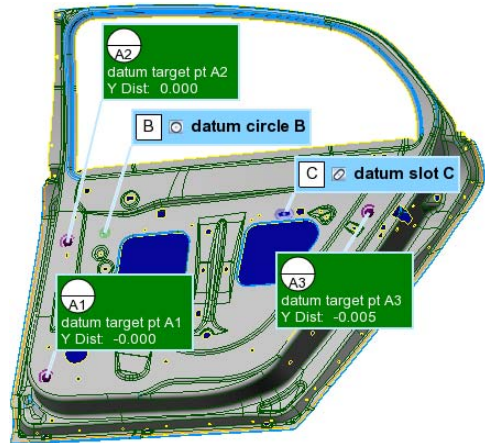


Results

The new data alignment is added to the alignment group of the Data object. The datum features and datum targets are measured.



The Data object is aligned to the Reference object using the A B C Datum Reference Frame.



Notes

- For datum target points, a radial tolerance is used; it is possible to reprobe points that are out of the radial tolerance.

Aligning surface Data objects to each other using surface information

When scanning a part, it is sometimes impossible to capture all the surfaces without moving the scanning device or changing the part position relative to the scanning device. This generates one surface Data object for each different setup used. If the device positions are not already aligned, it is possible to align these surface Data objects to one another. The alignment uses the areas that overlap with neighboring surface Data objects.



Align Data objects to Data objects one at a time

This method aligns one set of Data objects to a fixed set of Data objects. First, a prealignment is done by the user by picking matching points. Then, a best-fit alignment is performed using the surface shape in the overlying regions.

Objective: Align one set of surface Data objects to a fixed set of Data objects.

Required: Two or more surface Data objects with sufficient overlapping data to guarantee a proper alignment.

Steps

1. Choose *Align > Best-Fit Data Objects > Data to Data Objects*.

The Best-Fit Data to Data Objects dialog box opens.



2. Specify a **Name** for the data alignment.
3. Select the **Align Data Objects** method.
4. The **Prealign using point pairs** check box is selected by default.
5. In the **Data objects to align** list, select the Data objects that will be aligned.
6. In the **Fixed data objects** list, select **Other Data Objects**.

Fixed Data objects do not move for the alignment.

7. Specify a **Max distance**.

The maximum distance is used by the best-fit alignment to find overlapping areas.

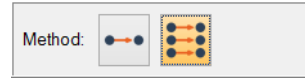
8. In the **Subsampling** list, select **1/1**.

Here, all points are used to refine the best-fit alignment but with a longer computing time. The default value is 1/4.

9. Click **Start**.

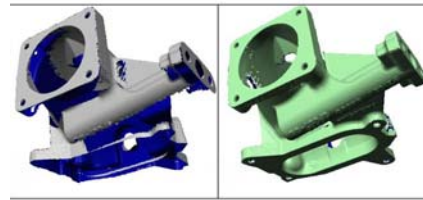
The Prealign using Point Pairs dialog box opens.

Fixed Data objects and the Data objects that will be aligned are displayed in separate viewports.

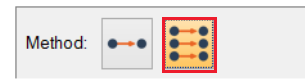


10. Move both the fixed Data object (left) and the Data object that will be aligned (right) to give them a similar orientation.

This facilitates the picking of pairs of points.

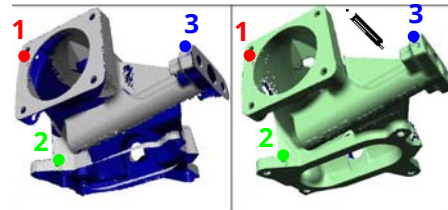


11. Choose the **N Point Pairs** prealignment method.



12. Click **Pick** to enable an interactive picking mode.

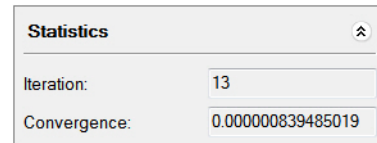
13. Anchor at least three matching points on both objects, using the same order. It is recommended to anchor four or five pairs of matching points. The matching points are displayed using the same color and the same number.



14. Click **Align** to confirm the alignment and exit the picking mode.

The best-fit alignment is automatically performed after the prealignment.

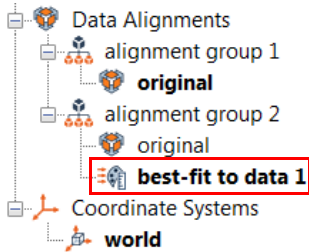
*As the alignment converges to a solution, results are displayed in the **Statistics** section.*



Results

A new data alignment is added to the alignment group of the aligned Data objects.

The surface Data objects are aligned to each other in the 3D scene.



Notes

When there are three Data objects or more, and once they are aligned together using a best-fit Data to Data Objects alignment, it is possible to optimize the alignment between all the Data objects, using surface information, through the **Globally Optimize Alignments** method.

Globally optimizing the alignments of all Data objects

When scanning parts in multiple device positions, a new scanned Data object is produced in each device position and then aligned to the Data objects from previous device positions using the best-fit Data to Data Objects functionality. When three or more Data objects from different device positions are present, the alignment of all Data objects can be optimized using the **Globally Optimize Alignments** method to ensure the best results in terms of alignment between the scanned Data objects. Typically, this method should be used in the last device position.

Objective: Optimize the alignment of all the Data objects.

Required: Three or more Data objects from different device positions are in the project and already aligned using the Best-Fit Data to Data Objects alignment.

Steps

1. Choose *Align > Best-Fit Data Objects > Data to Data Objects*.

The Best-Fit Data to Data Objects dialog box opens.



2. Specify the **Globally Optimize Alignments** method.
3. Specify a **Name** for the data alignment.
4. In the **Data objects to align** list, select **All**.

5. Specify a **Max distance**.

The maximum distance is used by the best-fit alignment to find overlapping areas.

6. In the **Subsampling** list, select **1/1**.

Here, all points are used to refine the best-fit alignment but with a longer computing time.

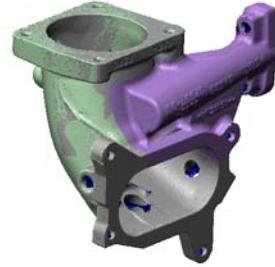
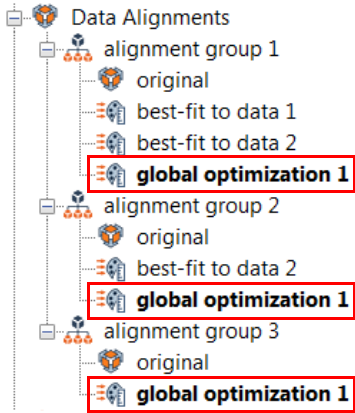
7. Click **Start**.

*As the alignment converges to a solution, results are displayed in the **Statistics** section.*

Results

A new data alignment is added to the alignment group of the aligned Data objects.

The surface Data objects are aligned in the 3D scene, and their alignment is globally optimized.



Coordinate systems



Cartesian



Cylindrical



Spherical

Introducing coordinate systems

A coordinate system describes the origin and orientation of an inspection project.

Three types of coordinate systems are supported by PolyWorks | Inspector:

- Cartesian
- Cylindrical
- Spherical

PolyWorks | Inspector projects come with a default coordinate system called “world”. It corresponds to the standard Cartesian coordinate system (x, y, z), whose origin is located at 0, 0, 0.

Multiple coordinate systems are useful for users who need to work in a specific frame of reference for the purpose of visualizing and reporting information. New coordinate systems can be created. They appear under the Coordinate Systems branch of the Tree View, and are represented in the 3D scene as an origin and three axes. Only one coordinate system can be active at a time.

In the 3D scene

The active coordinate system is displayed using red, green, and blue for the X, Y, and Z axes respectively.

The world coordinate system is easily identifiable since it has a blue cube at its origin.



Coordinate systems that are not active are displayed in gray.



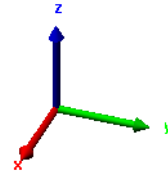
Using a new coordinate system

When a coordinate system is activated, the following occurs:

- All objects are represented in the new coordinate system, regardless of their status.
- All measurement results are transformed in order to be aligned in the new coordinate system. Measurement results remain valid.
- The numerical values of object properties related to location and orientation are expressed in the active coordinate system. Default values, for example in the object creation dialog boxes, do not change.
- Objects are exported in the active coordinate system.
- The bounding box of the 3D scene is recalculated. This can influence the creation of certain primitives, like a plane primitive created numerically, or certain dependent primitives that are created using the bounding box.

Understanding the coordinates of a point

Cartesian coordinates of a point are represented by x , y and z values based on three mutually perpendicular X , Y , Z axes.

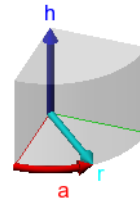


The cylindrical coordinates of a (x, y, z) point are represented by a trio of radius, azimuthal angle, and height values calculated according to the following convention:

The radius value (r) represents the distance from the origin along the X - Y plane.

The azimuthal angle value (a) represents the angle with respect to the positive X direction, calculated within the X - Y plane according to the right-hand rule. This angle ranges from 0 to 360 degrees.

The height value (h) represents the distance from the origin along the cylindrical axis (or Z axis).

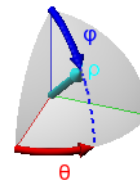


The spherical coordinates of a (x, y, z) point are represented by a trio of radius, azimuthal angle, and inclination angle values calculated according to the following convention:

The radius value (ρ) represents the distance from the origin.

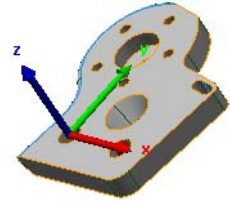
The azimuthal angle value (θ) represents the angle with respect to the positive direction, calculated within the X - Y plane according to the right-hand rule. This angle ranges from 0 to 360 degrees.

The inclination angle value (φ) represents the angle with respect to the vertical axis (or positive Z axis). This angle ranges from 0 to 180 degrees.



Creating and activating coordinate systems

PolyWorks | Inspector projects come with a default coordinate system called "world". New coordinate systems can be created as well from feature primitives, from translations and rotations, or numerically. When a new coordinate system is created, it is automatically activated.



As shown to the right, the active coordinate system is displayed using red, green, and blue for the X, Y, and Z axes respectively.

Objective: Create a new coordinate system using feature primitives to define the origin and orientation.

Required: Features whose geometric properties (ex. center point, normal) can be used to describe the origin and orientation of the coordinate system.

Steps

1. Choose *Tools > Coordinate Systems > Create Cartesian*.



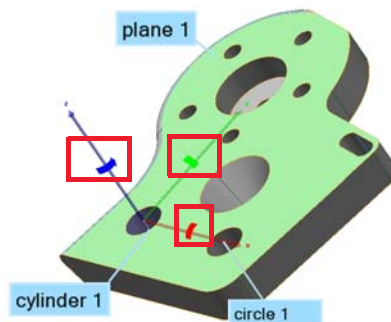
The Create Cartesian Coordinate Systems dialog box opens.

2. In the **Method** list, select **From Primitives**.
3. In the **Primary**, **Secondary**, and **Tertiary** lists, select the primitives that define the coordinate system.

Name:	csys 1		
Method:	From Primitives		
Primary:	plane 1 -nom-	▼	👤
Secondary:	cylinder 1 -nom-	▼	👤
Tertiary:	circle 1 -nom-	▼	👤

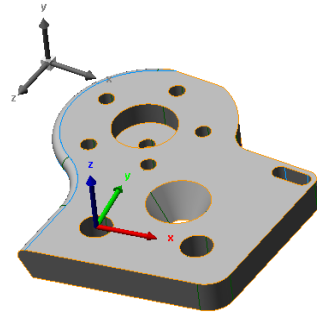
Optionally, click the hand icon and pick the primitives in the 3D scene.

4. In the 3D scene, clicking the colored arrow on an axis rotates the other axes around it by 90 degrees, thereby changing their direction.
5. In the Create Cartesian Coordinate Systems dialog box, click **Create** to confirm the new coordinate system.



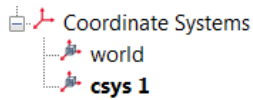
Tips

All coordinate systems are listed as objects in the Tree View. The active coordinate system is displayed in bold in the Tree View, and in color in the 3D scene. To activate a coordinate system, right-click it in the Tree View and choose *Activate*.

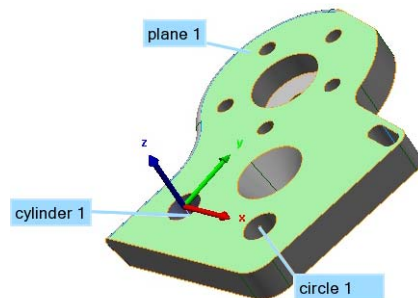


Results

The new coordinate system is added to the Tree View, under the Coordinate Systems branch. It is named **csys 1** by default.



The new coordinate system is displayed in the 3D scene.



Notes

- When the creation method is set to **Numerically**, the exact origin and axis of the coordinate system can be specified. No Reference object or feature primitives are required.
- When the creation method is set to **From Translation and Rotation**, the origin and axes can be offset from the active coordinate system position by entering specific values. No Reference object or feature primitives are required.

Name:

Method:

Origin

x:

y:

z:

Axes

First axis:

i:

j:

k:

Second axis:

i:

j:

k:

Name:

Method:

Translation		Rotation	
x:	<input type="text" value="0.000"/>	X	<input type="text" value="0.000"/>
y:	<input type="text" value="0.000"/>	Y	<input type="text" value="0.000"/>
z:	<input type="text" value="0.000"/>	Z	<input type="text" value="0.000"/>

Measuring dimensions and deviations



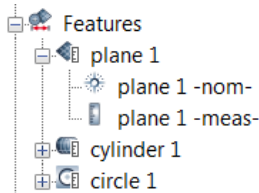
Introducing measurement objects in PolyWorks|Inspector

PolyWorks|Inspector is an object-oriented analysis and verification environment. All items imported or created in PolyWorks|Inspector are objects, and PolyWorks|Inspector preserves the relational correspondence of these objects throughout the inspection process.

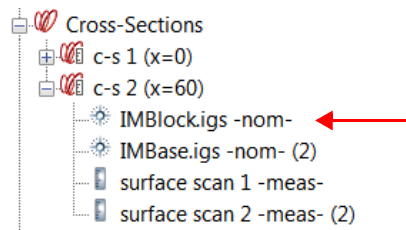
Measurement objects

Measurements are done through the use of measurement objects: data color maps, features, cross-sections, comparison points, and gauges. These objects can be measured, remeasured, updated, and reported. Measurement objects obtain results by calculating the deviation between nominal components, derived from a Reference object, and measured components, derived from a Data object. The components of feature and cross-section measurement objects can be displayed by expanding the objects in the Tree View, while the components of comparison points and gauges are contained within the objects themselves. A measured status icon is displayed in the Tree View for objects that have been successfully measured. For color maps, the measured status icon indicates that the color map is valid.

A simple feature has a nominal and a measured component.



The nominal and the measured component of a cross-section can be represented by several children, depending on the number of Reference and Data objects encountered by its slicing plane.



Measured status icon

A ruler icon indicates the measured status of objects in the Tree View. The color of the icon specifies whether or not the controls associated with the object are within tolerance:



All controls are within tolerance.



At least one control is within the warning limit.



At least one control is out-of-tolerance. This status has priority over the warning status.



The in-tolerance status is unknown for at least one control (for example, the measurement object is not controlled, or one of its controls is in error). This status has priority over all other statuses.

The **Display > Tree View** options page offers the **Show in-tolerance status on measurement objects** option, used to enable or disable the display of the in-tolerance status. When disabled, only the measured status is displayed (white ruler).

Automatic project update

The automatic project update is an option that keeps objects up-to-date by recalculating measured values as related objects are modified. By propagating changes throughout the inspection project, the coherence of the inspection project is always maintained. For more information, see [Automatic project update](#) on page 695.

Data color maps



Surface



Boundary



Primitives



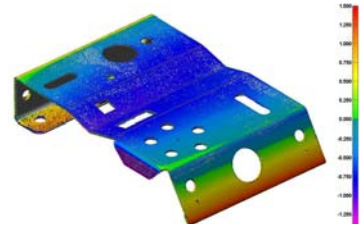
Thickness



Clearance

Measuring deviations using data color maps

Data color maps are measurement tools to measure the deviation of Data objects from the surfaces or boundaries of Reference objects, or from primitives or other objects. A deviation is calculated for each data point, or for just selected data points, of the Data object. The deviations are displayed in the 3D scene using a color map and can also be presented in report form (table). Color map display options are available to configure the color scale and the color mapping to get a specific result, like a pass/fail representation.

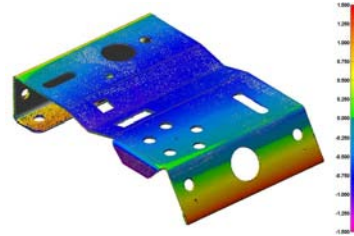


Data color map types

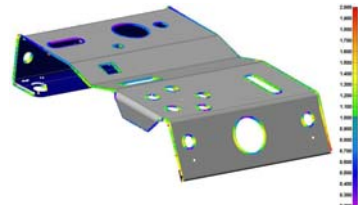
Data color maps are available on the *Measure* menu or on the **Data Color Maps** toolbar:



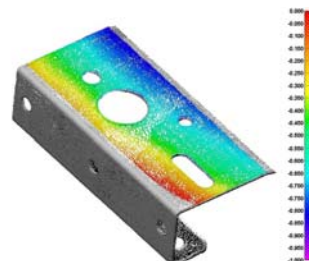
Data object deviations from Reference object surfaces



Data object deviations from Reference object boundaries

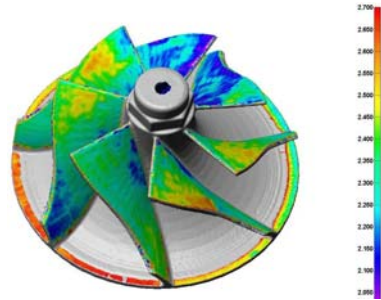


Data object deviations from feature primitives

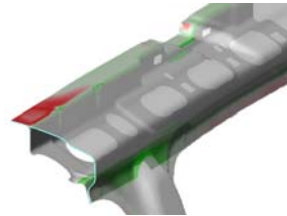




Data object thickness



Data object clearance

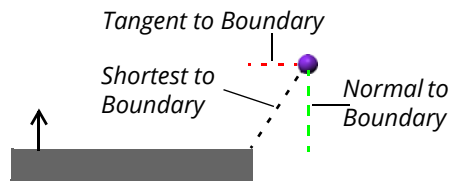
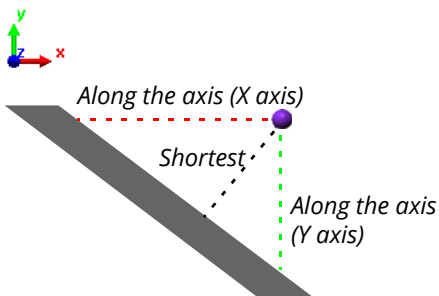


Understanding key parameters

Parameters are offered for the creation of data color maps. Possible values vary according to the type of data color map. Here are some key parameters:

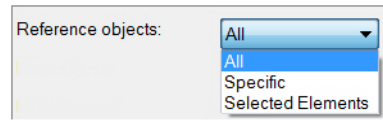
Direction

Select a direction along which deviations are measured. Possible values are illustrated below. They vary according to the type of data color map.



Reference objects

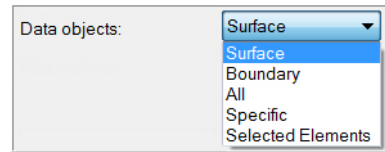
Choose the Reference objects that are taken into consideration for the measurement. Possible values are:



- **All:** Considers all non-ignored Reference objects.
- **Specific:** Considers only specific Reference objects; specify them in the list box offered.
- **Selected Elements:** Considers only the selected elements of Reference objects.

Data objects

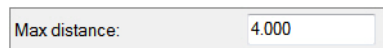
Choose which Data objects are taken into consideration for the measurement. Possible values are:



- **Surface:** Considers all surface Data objects, which includes point clouds and polygonal models.
- **Boundary:** Considers all boundary Data objects, such as boundary point clouds.
- **All:** Considers all non-ignored Data objects.
- **Specific:** Considers only specific Data objects; select them in the list box offered.
- **Selected Elements:** Considers only the selected elements of Data objects.

Max distance

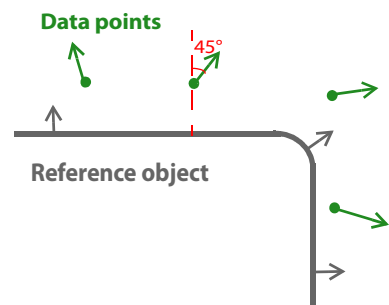
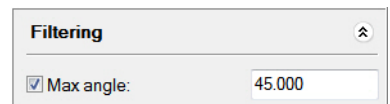
To speed up calculations, a maximum search distance is considered to find the deviations of the Data object from the Reference object. The default value is 4 mm and no deviation larger than 4 mm is detected. If high deviations occur on a part, ensure that the **Max distance** is larger than the expected deviations so as to measure all deviations.



Max angle

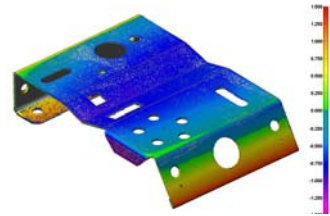
In the **Filtering** section, the **Max angle** parameter is enabled by default. This parameter acts as a filter to ensure that the normal of each considered data element has a normal compatible with the reference surfaces or the feature from which the deviations are measured. The angular deviation between the two normals must be smaller than the specified **Max angle**.

If the part is highly deformed, it may be necessary to disable this parameter which allows data elements to be considered regardless of their normal direction.



Measuring deviations from surfaces

The deviations of Data objects from Reference object surfaces can be measured. The results are displayed in the 3D scene by means of a data color map, as shown to the right.



The procedure used to measure deviations is explained below. For information on the key parameters, see [Understanding key parameters](#) on page 189.

Objective: Obtain the deviations of a Data object from the surface of a Reference object.

Required: The project contains a Data object and a Reference object. The Data object must be aligned to the Reference object.

Steps

1. Choose *Measure > Deviations of Data Objects > From Reference Object Surfaces*.



The **Measure Deviations from Surfaces** dialog box opens.

2. In the **Direction** list, select the measurement direction (**Shortest**, typically).
3. In the **Reference objects** and **Data objects** lists, select the objects that will be measured.
4. Specify a **Max distance**.
5. Click **Measure**.

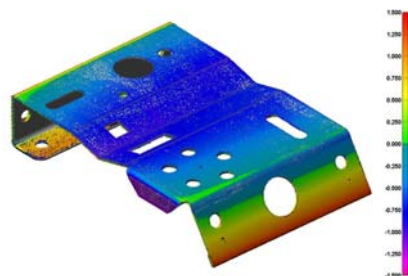
Name:	dev to surface 1
Method	
Direction:	Shortest
Parameters	
Reference objects:	All
Data objects:	Surface
Max distance:	4.000

Results



The data color map is added to the Tree View, under the Data Color Maps branch.



The color-mapped object is displayed in the 3D scene.

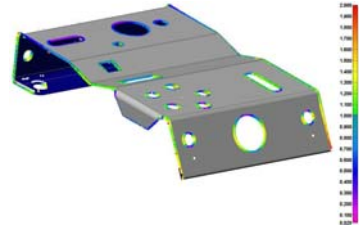


Notes

- To hide the display of the active data color map, click **Hide Color Map**. 
- To edit the color scale, choose *View > Edit Color Scale*. Conversion to colors, scale type, and range limits can be edited in the dialog box. 
- Once created, it is possible to access and edit the color map creation parameters if needed using its property sheet. Select the object in the Tree View and choose *Edit > Object Properties*.

Measuring deviations from boundaries

The perimeter of a Data object (usually boundary Data type point clouds) can be compared to the perimeter of a Reference object by measuring each data point to the open boundary edges of the Reference object. Deviations can be measured along the shortest, normal or tangent distance to the boundary. The results are displayed in the 3D scene by means of a data color map, as shown to the right.



The procedure used to measure deviations is explained below. For information on the key parameters, see [Understanding key parameters](#) on page 189.

Objective: Measure the deviation of the measured data points from the open boundaries of the Reference object, generating a data color map.

Required: The project contains a Data object as well as a Reference object with open boundaries. The Data object must be aligned to the Reference object.

Steps

1. Choose *Measure > Deviations of Data Objects > From Reference Object Boundaries*.



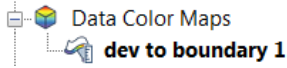
The Measure Deviations from Boundaries dialog box opens.

2. In the **Direction** list, select a direction of measurement (**Shortest to Boundary** typically).
3. In the **Reference objects** and **Data objects** lists, select the objects that will be measured.
4. Specify the **Max distance**.
5. Click **Measure**.

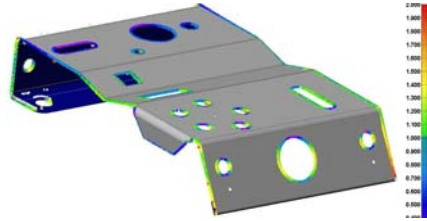
Name:	dev to boundary 1
Method	
Direction:	Shortest to Boundary ▼
Parameters	
Deviation sign:	Unsigned ▼
Reference objects:	All ▼
Data objects:	Boundary ▼
Max distance:	4.000

Results

The new data color map is added to the Tree View, under the Data Color Maps branch.



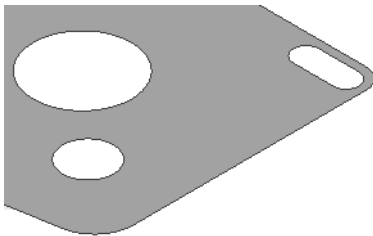
The color-mapped object is displayed in the 3D scene.



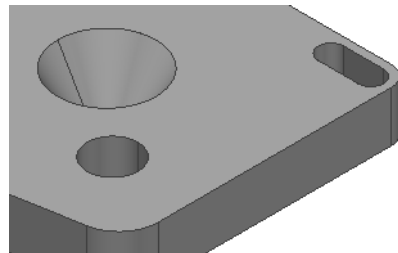
Notes

- A Reference object with open boundaries implies the presence of surface discontinuities in which case the model does not define a closed volume. In the following examples, a Reference object is illustrated with both types of boundaries: open and closed.

Open boundaries



Closed boundaries

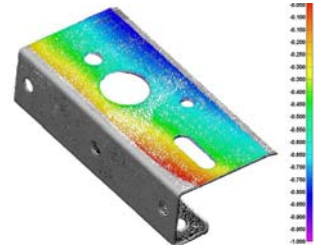


- To hide the display of the active data color map, click **Hide Color Map**.
- To edit the color scale, choose *View > Edit Color Scale*. Conversion to colors, scale type, and range limits can be edited in the dialog box.
- Once created, it is possible to access and edit the color map creation parameters if needed using its property sheet. Select the object in the Tree View and choose *Edit > Object Properties*.



Measuring deviations from feature primitives

The shape of a Data object can be compared to a geometric feature, such as a plane or a cylinder, by measuring each data point to a feature primitive. The results are displayed in the 3D scene by means of a data color map, as shown to the right.



The procedure used to measure deviations is explained below. For information on the key parameters, see [Understanding key parameters](#) on page 189.

Objective: Measure the deviations of the measured data points from feature primitives, which generates a data color map.

Required: The project contains a Data object and a feature primitive, which are aligned.

Steps

1. Choose *Measure > Deviations of Data Objects > From Feature Primitives*.



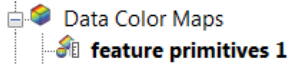
The Measure Deviations from Feature Primitives dialog box opens.

2. In the **Direction** list, select the measurement direction.
3. In the **Primitives** list, select the primitives that will be used.
4. In the **Data objects** list, select the Data objects that will be used.
5. Specify a **Max distance**.
6. Click **Measure**.

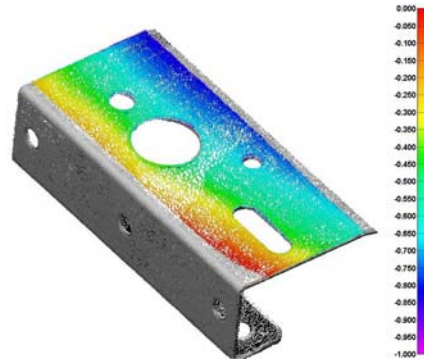
Name:	feature primitives 1
Method	
Direction:	Shortest
Parameters	
Primitives:	Specific
	plane 1 -nom-
Data objects:	Surface
Max distance:	4.000

Results

The data color map is added to the Tree View under the Data Color Maps branch.



The color-mapped object is displayed in the 3D scene.



Notes

- To hide the display of the active data color map, click **Hide Color Map**.
- To edit the color scale, choose *View > Edit Color Scale*. Conversion to colors, scale type, and range limits can be edited in the interface.
- Once created, it is possible to access and edit the color map creation parameters if needed using its property sheet. Select the object in the Tree View and choose *Edit > Object Properties*.



Measuring the thickness of a part

The thickness of thin-walled parts can be measured by matching each data point to an opposing surface that has an opposite orientation. A data color map is generated on the Data object to illustrate the deviations. If the project contains a Reference object, it is also possible to measure the deviation between Data object thicknesses and Reference object thicknesses at similar locations. The Data object must be aligned to the Reference object.

The procedure used to measure the thickness of a part is explained below. For information on the key parameters, see [Understanding key parameters](#) on page 189.

Objective: Calculate the thickness of a Data object.

Required: The project contains a Data object.

Steps

1. Choose *Measure > Thickness of Data Objects*.

The Measure Thickness of Data Objects dialog box opens.



2. In the **Data objects** list, select the Data objects.
3. Specify a **Max distance**.

The distance should be at least as large as the width of the thickest wall that will be measured.

4. If the Data object is a point cloud, select **Sampling step** which ensures smooth, continuous shading.

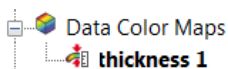
A smaller sampling step results in finer measurement details, while a larger sampling step speeds up the operation.

5. Click **Measure**.

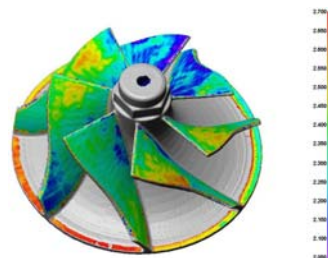
Name:	thickness 1
Parameters	
Data objects:	Surface
Max distance:	10.000
Point clouds:	
<input checked="" type="checkbox"/> Sampling step:	1.000
<input type="checkbox"/> Compute nominal and deviation	

Results

The data color map is added to the Tree View, under the Data Color Maps branch.



The color-mapped Data object is displayed in the 3D scene.



Notes

- To hide the display of the active data color map, click **Hide Color Map**.
- To edit the color scale, choose *View > Edit Color Scale*. Conversion to colors, scale type, and range limits can be edited in the dialog box.
- Once created, it is possible to access and edit the color map creation parameters if needed using its property sheet. Select the object in the Tree View and choose *Edit > Object Properties*.
- Optionally, a Reference object can be used for thickness comparison purposes. In this case, the data color map displays either the measured thickness, the nominal thickness, or the deviation between the two thicknesses. In all cases, the data color map is displayed on the Data object.



- In the Measure Thickness of Data Objects dialog box, select **Compute nominal and deviation**.
- Specify a value for the **Reference objects**, **Max distance**, and **Max angle** parameters.

<input checked="" type="checkbox"/>	Compute nominal and deviation
Advanced ▾	
Nominal and deviation ▲	
Reference objects:	All ▾
Max distance:	4.000
<input checked="" type="checkbox"/>	Max angle: 45.000

- Click the **Object Display Options** menu button on the 3D scene toolbar.

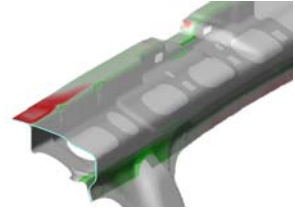


- In the **Thickness color mode** list, under **Data color maps**, select **Nominal**, **Measured**, or **Deviation**.

Data color maps	
Thickness color mode:	Measured ▾
Texture filling radius:	4.0

Measuring the clearance between parts

When two or more Data objects have been aligned to each other, the clearance between these objects can then be measured. The resulting data color map illustrates gaps or interference between the objects. This information can be used to adjust their position.



The procedure used to measure the clearance between parts is explained below. For information on the key parameters, see [Understanding key parameters](#) on page 189.

Objective: Measure the clearance between several Data objects.

Required: The project contains two or more Data objects that have been previously aligned to each other.

Steps

1. Choose *Measure > Clearance between Data Objects*.



The Measure Clearance between Data Objects dialog box opens.

All nonignored polygonal Data objects are considered for the clearance operation.

2. Choose the measurement method, either for measuring objects between themselves or to master objects only.

If a master object is used, it becomes the focus of the measurement and all other Data objects are measured only to it.

3. When the thickness of a part is known and only one side of the part was scanned, specify a **Thickness** to take into account the other side of the part.

4. Specify a **Max distance** and a **Max angle**.

5. Click **Measure**.

Method

Measure objects between themselves

Measure objects to master objects only

Parameters

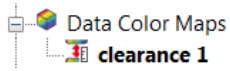
Name	Thickness
Part_1_1mm.stl	1.100
Part_1_0mm.stl	1.000
Part_1_7mm.stl	1.700

Max distance:

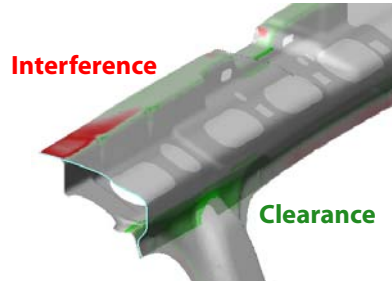
Max angle:

Results

The data color map is added to the Tree View, under the Data Color Maps branch.



The color-mapped object is displayed in the 3D scene.



Notes

- To hide the display of the active data color map, click **Hide Color Map**.
- To edit the color scale, choose *View > Edit Color Scale*. Conversion to colors, scale type, and range limits can be edited in the interface.
- Once created, it is possible to access and edit the color map creation parameters if needed using its property sheet. Select the object in the Tree View and choose *Edit > Object Properties*.



Changing the color map display

The color scale controls the way measured deviations are converted to colors in color maps. PolyWorks | Inspector provides many predefined color scales, and an interface that allows customizing them.

Objective: View a color map using different color scales.

Required: A project contains an active data color map.

Steps

1. Choose *View > Edit Color Scale*.

The Edit Color Scale dialog box opens.



2. In the **Conversion to colors** list, select a method. The following methods are available:

- **Auto Range** assigns a predefined range of colors to the range of color map values.
- **Use Customized Scale** assigns colors based on a specific range of values.
- **Use Object Tolerances** assigns colors based on the pass/fail status of data points.

3. In the **Scale type** list, select a predefined type.

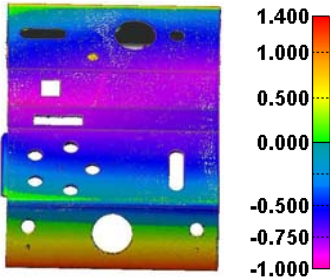
4. To create a custom color scale, click **Add** to create a completely new color scale, or click **Edit** to modify an existing color scale.

5. Edit the **Range limits** parameters, if using the Auto Range conversion to colors, to display the desired range of deviation values.

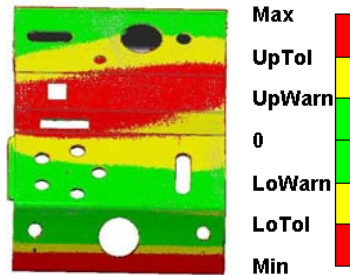
6. Click **Apply**.

Results

If the **Auto Range** method is specified, the range of deviations is displayed as a continuous map, with minimum and maximum values defined in the **Range limits**.

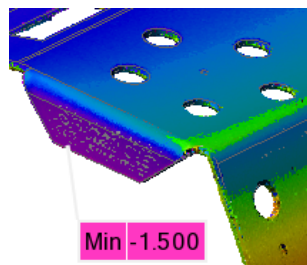
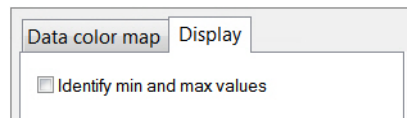


If the **Use Object Tolerances** method is specified, the deviations are displayed as passing or failing the tolerance test, based on the tolerance associated with the Reference object.



Notes

- To hide the display of the active data color map, click **Hide Color Map**.
- Choose *Tools > Options > Display > Color Maps* for additional color map display settings, such as the use of deviation vectors and how color mapping can be applied to both Data and Reference objects.
- Minimum and maximum deviations can be displayed on a data color map. To view them, select **Identify min and max values**, available on the **Display** tab of the property sheet of the data color map.



Displaying data point deviations on a data color map

The deviation value of any data point on a data color map can be displayed using an annotation. Annotations can be created interactively by clicking the model and then repositioning them for better viewing. Annotation values can also be reported in a table.

Objective: Place annotations on the data color map, showing individual data point deviations, and report them.

Required: The project contains a Data object aligned to a Reference object, and an active data color map.

Steps

1. Choose *Tools > Annotations > Create > Color Map Point Annotations*.

An interactive picking mode is launched.

2. Hover the pointer over the color map in the 3D scene to temporarily display the deviation of the closest data point.

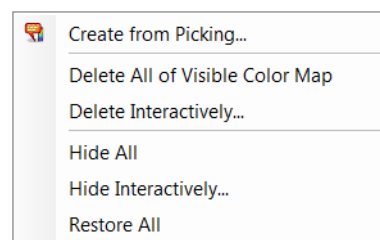
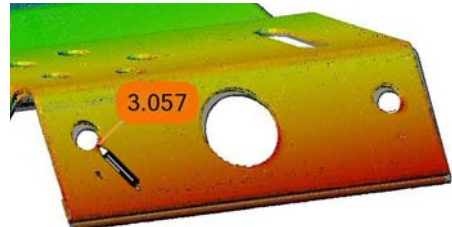
3. Click the color map to create the annotation, which shows the deviation of the data point. Right-click to exit the mode.

4. For a clearer display, or for snapshot purposes, the annotations can be dragged. Choose *Tools > Annotations > Position*.

- Press Spacebar to temporarily exit the mode and perform rotations/translations.
- Press ESC to exit the mode.

5. Choose *View > Hide > All Color Map Point Annotations* to control the visibility of the annotations.

6. Choose *Report > Create Tables > From All Color Map Point Annotations of Active Data Color Map* to report the annotations.



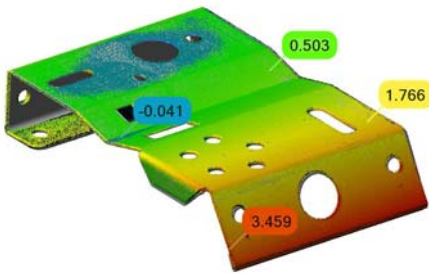
Tip

Tools to delete, hide, and restore annotations are also accessible using the down arrow next to the Create Color Map Point Annotations button.



Results

Individual data point deviations are displayed using annotations in the 3D scene.









Data point annotations are added to a table for reporting.

Table Type Color Map Point (Data Deviation)

CSYS	world
Data Alignments	perpendicular planes

Index	Nom (x)	Nom (y)	Nom (z)	Dev(x)	Dev(y)	Dev(z)	Dev(3D)
1	127.237	131.275	12.186	0.000	0.075	0.000	0.075
2	0.007	67.145	20.533	0.007	-0.003	-0.002	-0.008
3	83.555	120.414	5.848	0.010	0.000	0.040	-0.041
4	71.808	12.957	3.439	0.122	0.000	-0.488	0.503
5	185.169	128.986	24.482	2.453	1.986	-1.416	3.459
6	149.546	10.633	-1.766	0.000	0.000	-1.766	1.766

Features

	<u>Points</u>		<u>Cylinders</u>
	<u>Lines</u>		<u>Cones</u>
	<u>Planes</u>		<u>Spheres</u>
	<u>Circles</u>		<u>Surfaces</u>
	<u>Arcs</u>		<u>Slabs</u>
	<u>Slots</u>		<u>Polylines</u>
	<u>Rectangles</u>		<u>Patterns</u>
	<u>Polygons</u>		<u>Distances</u>
	<u>Ellipses</u>		<u>Angles</u>

Defining part geometry using features

Features define geometric characteristics of a part, such as holes and planar surfaces, and are typically used for part alignment and dimensional inspection purposes.

A feature typically contains a nominal and/or a measured component, also referred to as primitives. The nominal primitive represents the theoretical or ideal measurement, such as that found on a CAD model. The measured primitive is created from data points measured on the actual part being inspected.

Features are available on the *Measure > Features* menu or on the **Features** toolbar.



Feature types

Features can be categorized as follows:

Individual features

Features containing a nominal and/or a measured primitive. Individual features can be categorized by their geometrical properties. The most common categories are:

Surface-based:

Features that have a surface.



Plane



Cylinder



Cone



Sphere

Curve-based:

Planar curves with a center point (x, y, z) and a directional vector (i, j, k) .



Circle



Arc



Slot



Rectangle



Polygon



Ellipse

Other:



Point



Line



Polyline



Slab*

*Feature composed of two existing parallel plane features with opposite orientations and containing a nominal and/or a measured primitive.

Composite features

Features composed of two or more existing features.



Distance



Angle



Pattern



Surface**

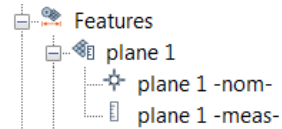
**Feature composed of CAD surfaces (nominal component) and data points (measured component).

Feature primitives

A feature typically contains a nominal and/or a measured primitive:

Nominal primitives represent the theoretical or ideal measurement, such as that found on a CAD model.

Measured primitives are the geometrical entities found on the actual part being inspected.



The feature creation methodology

A standard dialog box with common and contextual parameters is used for the creation of individual features and their primitives. Other features (surface, slab, angle, distance, and pattern) are created using their own creation dialog box, described respectively in each subsection.

The subsections that follow describe the feature creation methods and their parameters.

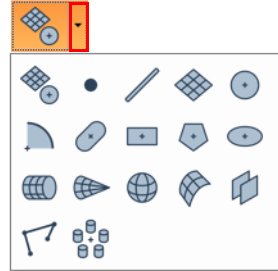
The feature creation dialog box

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose which type of feature to create.



3. In the **Name** text box, specify a name, if desired.
4. In the **Datum feature label** list, assign a datum label, if desired.
5. In the **Method** list, select a creation method.
6. In the **Submethod** list, select a creation submethod, if available.
7. Specify the required **Parameters** according to the creation method.
8. Click the appropriate button to launch the creation.

Key feature creation parameters

The feature creation dialog box offers the following parameters, depending on the feature type:

Name

A default name is automatically entered in the **Name** text box. It can be edited, if desired.

The same name is assigned to the primitive with the -nom- (nominal) or the -meas- (measured) suffix according to the primitive status.

Datum feature label

A datum label can be specified for all individual features, except polylines, to create a Datum feature. Datum features are used in GD&T controls ([Adding and encoding GD&T controls](#) on page 508).

Choose a datum letter in the **Datum feature label** combo box, or type in a customized datum label, consisting of one or more letters. Note that datum labels cannot contain the letters I, O, Q, X, Y, and Z. Once created, the feature is assigned the specified datum label.

Method

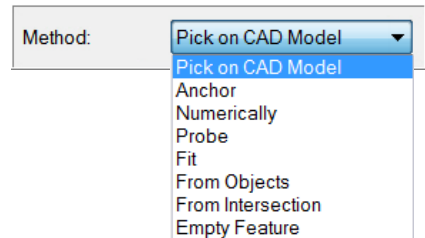
The creation methods available vary according to the type of feature selected.

The **Pick on CAD Model**, **Anchor**, **Numerically**, **Probe**, and **Fit** creation methods follow standard creation processes which are described later, and result in a feature object with a nominal or a measured primitive.

Construction methods, such as **From Objects**, **From Intersection**, and **From Projection**, create features using existing objects.

The **From Elements** construction method creates points from selected elements.

The **Empty Feature** creation method allows creating an empty geometric feature of a specific type, to which a nominal and/or measured object can eventually be added.



Parameters

The parameters available in the dialog box vary according to the creation method specified in the **Method** list box.

Creation methods explained

Pick on CAD model

Features (except arcs, points, polylines, and lines) can be created by picking the CAD geometry. For instance, a cylinder on the CAD model can be picked to create a nominal cylinder feature.

The **Pick on CAD Model** creation method offers two submethods for surface-based features:

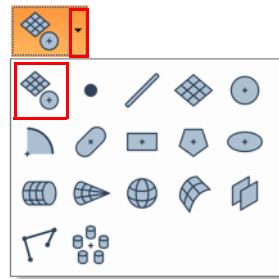
- **Single Surface:** Creates a feature by picking a single CAD surface in the 3D scene.
- **Multiple Surfaces:** Creates a feature by picking multiple CAD surfaces in the 3D scene.

To create different types of features without opening separate tools, choose *Measure > Features > Create*.



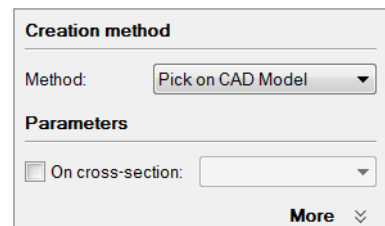
The Create Features dialog box opens.

Choose **Multiple types** from the feature type drop-down menu.



Most standard features can be picked using this interface.

- To create curve-based features, such as circles and slots, two approaches can be used:
 - Move the mouse cursor on a CAD model surface to select the closest curve-based feature and pick the one desired when it is highlighted.
 - Move the mouse cursor in the void within a hole to select the first found primitive based on the current 3D scene view and pick.



Anchor

Features can be created by anchoring their primitives on visible Data and Reference objects.

The **Anchor** creation method enables an interactive mode used to anchor primitives.

Each type of feature is created using a specific anchoring method. The **How To Window** button of the interactive mode guidance toolbar displays a diagram illustrating how to anchor points to create the specified feature type.

When finished anchoring the required number of points, a primitive passing through the picked points is automatically created and added to the Tree View.

Numerically

Features can be created numerically using mathematical definitions.

The **Numerically** creation method offers parameters specific to the type of primitive being created.

Point, Center, Origin, or Apex coordinates can be provided by manually entering the (x, y, z) coordinates or by dragging a center-point-based primitive (arc, circle, ellipse, point, polygon, rectangle, slot, and sphere) and dropping it over the **x, y, or z** text box.

A **Radius** value can be provided manually or by dragging a primitive with a radius (arc, circle, polygon, rectangle, slot, cylinder, and sphere) and dropping it over the text box.

Orientation or Normal coordinates can be provided by manually entering the (i, j, k) coordinates or by dragging a direction-based primitive (arc, circle, cone, cylinder, ellipse, plane, polygon, rectangle, slot, and line) and dropping it over the **i, j, or k** text box.

Probe

Features can be created by probing a primitive directly using a measuring device.

The **Probe** method offers several parameters that allow users to customize the probing session.

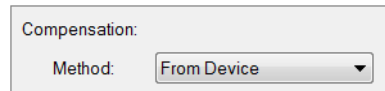
Thin material

It is recommended to enable this parameter when probing curved-based features on an object made of a thin material like sheet metal. It takes the thickness of the material and the constraining plane into consideration to calculate the compensation correctly, even when the probe is in contact with the edge instead of the sidewall.

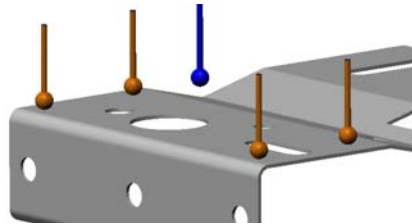


Compensation method

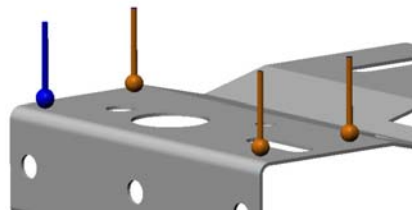
By default, the **Compensation Method** is generally set to **From Device**. This method uses the **Compensation Method** value of the probing device specified on the **Probing Device > Compensation Method** page of the PolyWorks|Inspector Options dialog box. The following values are available:



Compensation Point: using this method, a final probed point is used to identify the compensation direction. The probe is positioned away from the part material, and the probing session is ended.

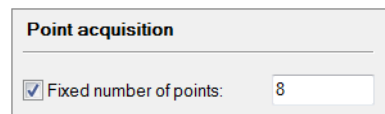


Probe Orientation: the probe orientation is used to identify the compensation direction. The first point must be gathered with the probe oriented normal to the contact surface.



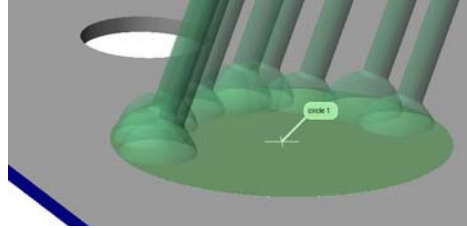
Fixed number of points

In order to more strictly control the way each feature is probed, it is possible to set the number of probed points required to define it. When **Fixed number of points** is selected, any number of points greater than or equal to the mathematical minimum can be specified. The fixed number of points can also be set for constraining planes.

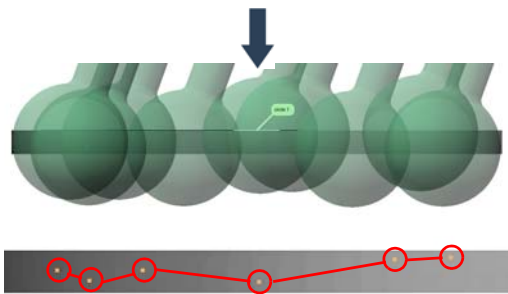


Constraining planes

A constraining plane helps control the orientation of two-dimensional curve-based features like circles and slots. Since points probed around the feature perimeter may have different heights, the constraining plane prevents the feature from fitting at a tilt.



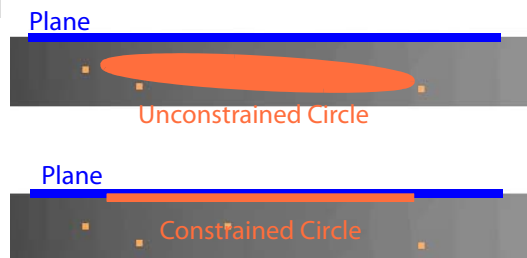
When a curve-based feature, such as a circle, is probed, the exact height of each probed point will vary slightly.



If a constraining plane is specified, the probed points will be projected onto that plane, removing the height variation.



While an unconstrained circle could be tipped compared to the part surface, a constrained circle will lie exactly on the surface plane.



Fit

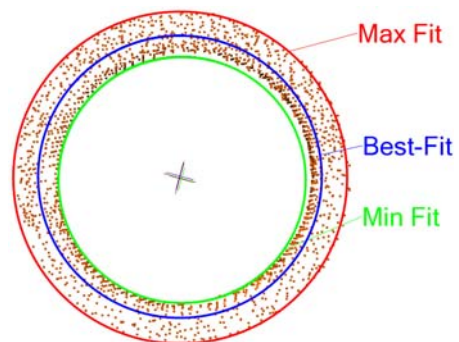
When creating or extracting features from a point cloud or a polygonal model, the user can specify how the feature should be fit. Three items are available: Best-fit, Min, and Max. The method specified will determine where, within the data elements with the qualifying distance and angle, the feature primitive will be positioned. The **Fit** creation method allows creating features by fitting their primitives to selected Data or Reference object elements or to picked points.

If the **Best-fit** fit type is selected, a standard best-fit algorithm is applied, using a pure least-squares approach to generate a feature that is an average fit within the eligible data elements. **Best-fit** is the default fit type for many feature types, such as planes. It is a reasonable choice in regions where the point cloud is expected to generally define a smooth surface and any digitizer noise is equally distributed above and below the surface.

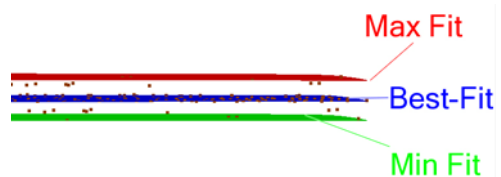
If the **Min** fit type is selected, the feature is fit such that no points are enclosed by the primitive. For a circular hole, the circle feature would fit to the innermost eligible data points, while for a planar surface, the plane would fit to the bottom eligible data points. It is a reasonable choice when defining a hole in the piece, because it describes the largest mating diameter possible without interference.

If the **Max** fit type is selected, the fitting algorithm applied creates the smallest primitive that encloses all considered data points. For a cylindrical pin, the cylinder would fit to the outermost eligible data points, while for a planar surface, the plane would fit to the top eligible points. The **Max** fit type is a reasonable choice when defining a physical pin or a diffuse surface like a honeycomb, because it describes the actual mating surface without interference.

A Best-fit, Min, and Max circle feature, fit to a selection of data points.



A Best-fit, Min, and Max plane, fit to a selection of data points.



Constructing features

When creating features using construction methods, such as **From objects** or **From Intersection**, the creation method section of the creation dialog box offers a submethod selection list box. If the objects required for the construction are preselected in the Tree View, a corresponding construction submethod is automatically specified according to the type of objects selected.

When constructing features using existing features, the primitives of the selected features are divided into two separate groups, nominal and measured, and the desired construction operation is performed on each group. Note the following:

- If all the source features contain a nominal primitive, the resulting features also contain a nominal primitive.
- If all the source features contain a measured primitive, the resulting features also contain a measured primitive.
- The nominal primitive and the measured primitive created using the same source feature belong to the same resulting feature.

If both the nominal and the measured primitives could not be created, and the **Create dependencies** check box was selected in the **More** subsection of the creation dialog box, corresponding empty features will be created. Otherwise, no feature is created.

Creating features from files

Commands on the *Measure > Features > Create from Files* submenu can be used to import same-type primitives from text files as well as to import one or more types of primitives from specific file types (for example, IGES files).

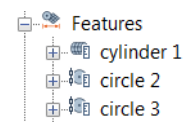
Each new primitive is assigned to a new feature of the primitive type. For example, importing a circle results in the creation of a circle feature that contains the circle primitive. The status of the new primitives (nominal or measured) can be specified when imported.

Feature dependencies

When a feature is constructed using existing objects in the project, for example, when using the **From Objects** method, the newly created features are dependent on the parent objects used to create them. This means that any changes made to the parent objects will result in the recalculation of the dependent features.

Dependent features can easily be identified in the Tree View, by a chain icon next to the feature icon.

To remove the dependencies of a feature, select it in the Tree View and choose *Measure > Features > Remove Dependencies*.



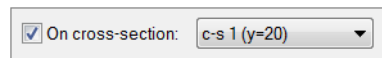
Creating features on cross-sections for 2D inspection

Inspecting in 2D along cross-sections is useful for parts that are too flexible or deformed, such as plastic parts and castings, or for measuring geometry types that are difficult to measure in 3D, like fillet radius and wall thickness. Points, lines, circles, arcs, slots, angles, and distances can be directly created on nominal cross-section components and then extracted on measured cross-section components. A cross-section object must exist and be specified upon creation of these features.

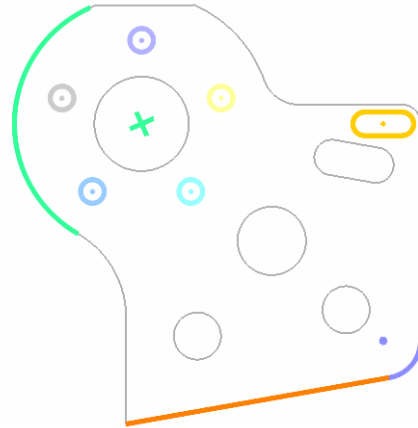
The following methods can be used to create features on cross-sections:

- **Pick on CAD Model:** Circles, arcs, slots, and lines can be created on cross-sections using this creation method. This method enables an interactive mode that has the same behavior as the existing mode for creating features. The only difference is that features are created by picking on the nominal cross-section components extracted from a CAD model. A preview of the feature that will be created is displayed.
- **Anchor:** Circles, arcs, slots, lines, and points can be created on cross-sections using this creation method. This method enables an interactive mode used to anchor primitives on nominal or measured cross-section components.
- **Numerically:** Circles, arcs, slots, lines, and points can be created on cross-sections using this creation method. Features that are created numerically are automatically projected onto the cross-section plane.
- **From Objects, From Intersection, and From Projection:** Circles, slots, lines, and points can be created on cross-sections using a construction method. Features created from other features are automatically projected onto the cross-section plane, while always preserving their form. For example, when creating a cross-section feature from the intersection of a plane and a cylinder, the circle resulting from the intersection is first computed in space. Then its center point is projected onto the cross-section plane and its axis orientation is forced to be the same as the cross-section normal vector while keeping the same radius.
- **Empty Feature**

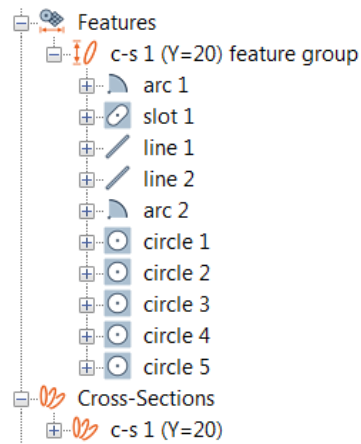
To create features on cross-sections, the **On cross-section** parameter, located under the parameters section of the feature creation dialog box, must be selected and a cross-section must be specified.



Once the feature creation is launched, a special display mode is enabled. Only the specified cross-section is displayed in the 3D scene. As the cross-section features are created, they are also highlighted in the 3D scene.



A link associates the features with the specified cross-section by automatically placing them in a cross-section feature group named after the cross-section object and displaying them in the Tree View, under the Features branch.

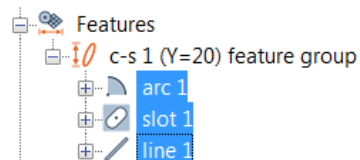


Duplicating cross-section features in 3D

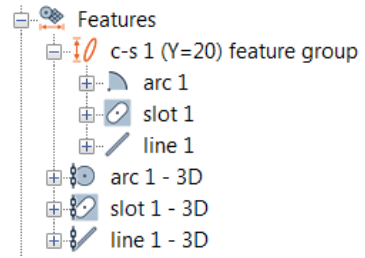
Features created on a cross-section for 2D inspection can be duplicated in 3D for other purposes.

The resulting duplicated feature is dependent on its cross-section feature counterpart: the nominal primitive, on the one hand, is identical to the cross-section feature, except for arc features, which are duplicated as circles; the measured primitive, on the other hand, can deviate from the cross-section feature, since it is extracted in the active alignment instead of the alignment used to extract the cross-section.

To duplicate cross-section features in 3D, select the feature in the Tree View and choose *Edit > Duplicate Cross-Sections Features in 3D*.



Duplicated features are created outside of the cross-section feature group but are associated to their corresponding cross-section feature by a dependency link.



Analyzing measured point deviations

Deviation information of measured points for a feature can be displayed in the 3D scene at any time, without having to create a data color map. This information is available when features are measured and the appropriate option is enabled.

Objective: Display the measured point deviations for a feature.

Required: The project contains a measured feature.

Steps

1. Select a feature in the Tree View.



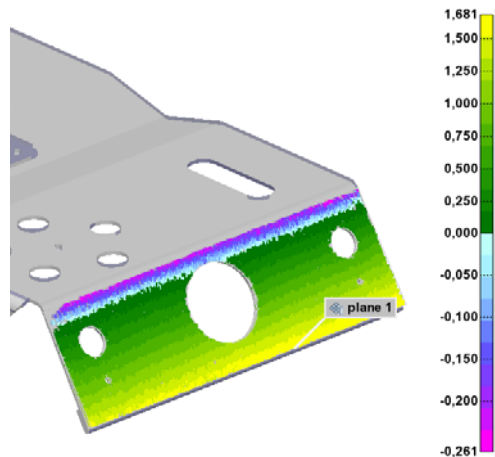
2. Choose *View > Restore > Feature Deviations*.

Alternatively, right-click the selected feature in the Tree View and choose *View > Deviations > Restore*.

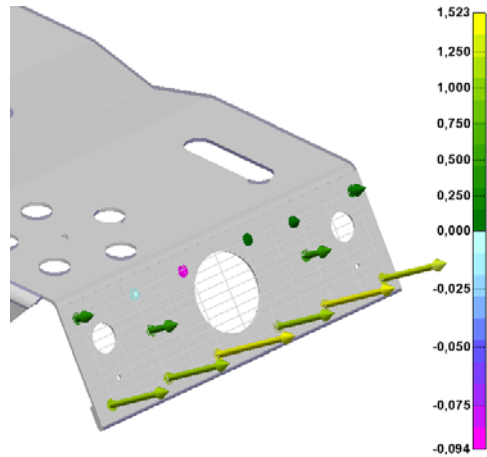
Results

The deviations are displayed according to predefined options (see [Feature measured point deviation display options](#) on page 221 for information).

- Deviation display for features measured by extraction.

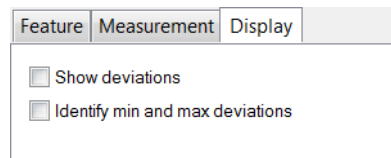


- Deviation display for features measured by probing.

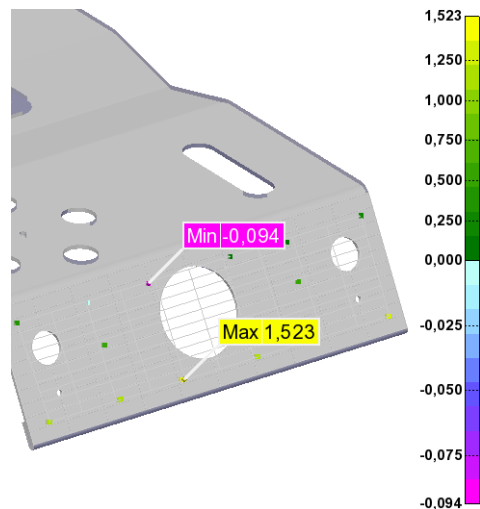


Notes

- The deviation display can also be enabled/ disabled from the property sheet of each feature. Select a feature, choose *Edit > Object Properties*, then click the **Display** tab.



- The **Display** tab also offers the **Identify min and max deviations** parameter. When selected, annotations are displayed in the 3D scene to identify the minimum and maximum measured point deviations for the feature. In this example, the parameter has been enabled for a plane feature.



Feature measured point deviation display options

A number of display options are available to modify the way measured point deviations are represented for features.

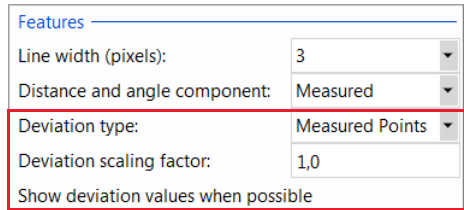
1. Click the **Object Display Options** menu button on the 3D Scene toolbar.



The **Features** section contains the following options:

- **Deviation type:**

By default, **Measured Points** is selected, which means that the deviation of measured points is displayed with respect to the nominal or the measured primitive (see the **Considered feature component** option later on in this section).

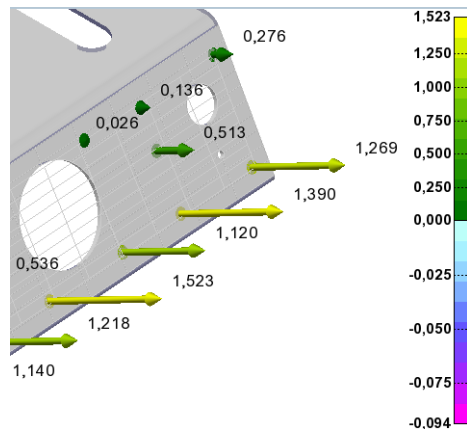


- **Deviation scaling factor:**

Increases the length of deviation vectors for better viewing.

- **Show deviation values when possible:**

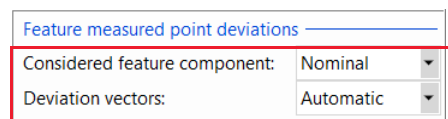
Deviation values can be displayed as 3D text by enabling this option.



The **Feature measured point deviations** section contains the following options:

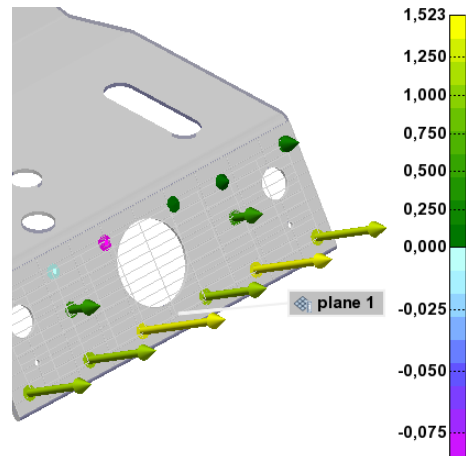
- **Considered feature component:**

The deviations can be displayed from the **Nominal** component or the **Measured** component.

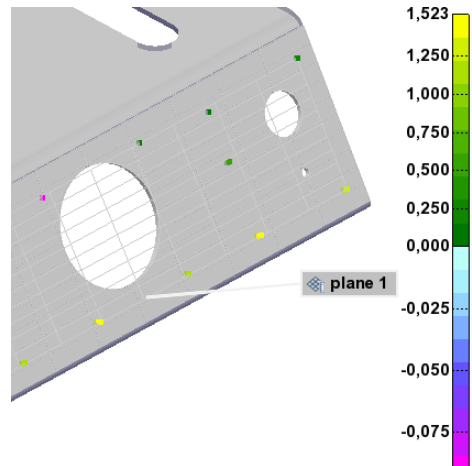


- **Deviation vectors:**

It controls whether deviation vectors are displayed. The default value is set to **Automatic**, which lets the application decide whether the vectors are shown or not based on the number of point deviations that are displayed in the 3D scene. The deviation vector display can also be set to **Show** or **Hide**.



The measured point deviations are displayed as colored points, based on the active color scale.



Notes

Additional options are found on the **Display > Features** page of the PolyWorks | Inspector Options (available by choosing *Tools > Options*).



Creating point features

Point features are typically created with a nominal primitive, using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal point primitives

Objective: Create a nominal point primitive interactively or using mathematical definitions.

Required: The project contains at least a Reference object, a Data object, a surface-based feature, a curve-based feature, or a cross-section-based object, or the information that defines the nominal primitive is available.

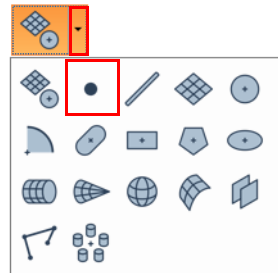
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

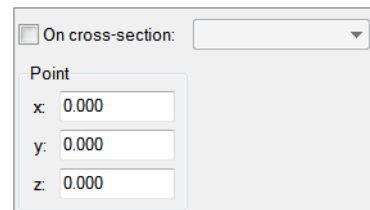


2. Expand the feature type drop-down menu and choose **Point**.
3. Choose one of the following methods.



Numerically

1. In the **Method** list, select **Numerically**.
2. Specify the **Point** coordinates to define the feature.
3. Click **Create**.



Anchor

1. In the **Method** list, select **Anchor**.

The interactive anchoring mode is automatically activated. If not, click **Anchor**.



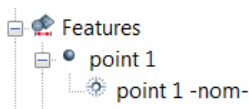
2. In the 3D scene, anchor a point at the desired location.

Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.

3. Right-click to exit the anchoring mode.

Results

The point feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured point primitives

To obtain the measured primitive of a point feature, the most typical method available is **Probe**. The method is explained in the following subsection.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Probe

Objective: Create the measured point primitive by probing it.

Required: The project contains a nominal point primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the features that will be probed.



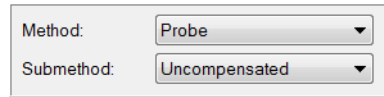
2. Choose *Measure > Features > Define Measured*.

Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.

The Define Measured Feature Components dialog box opens.



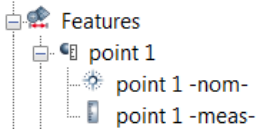
3. In the **Method** list, select **Probe**.
4. In the **Submethod** list, select **Uncompensated**.
5. Click **Probe**.
6. Probe the measured point primitive.



Results

The probed measured primitive is added to the point feature, under the Features branch of the Tree View.

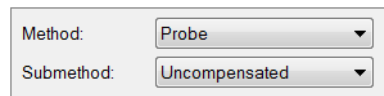
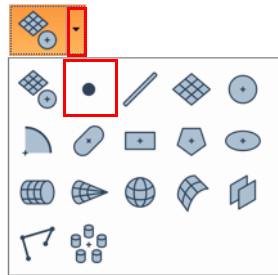
The primitives are displayed in the 3D scene.



Notes

To probe a measured primitive without a nominal primitive:

1. Choose *Measure > Features > Create*.
2. Expand the feature type drop-down menu and choose **Point**.
3. In the **Method** list, select **Probe**.



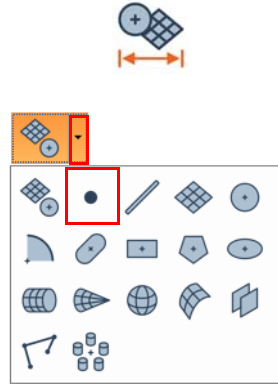
Constructing point features

Objective: Construct a point feature using existing objects.

Required: The project contains objects.

Steps

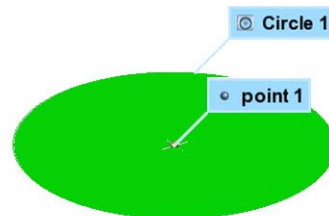
1. In the Tree View, select the objects to use for the construction.
2. Choose *Measure > Features > Create*.
The Create Features dialog box opens.
3. Expand the feature type drop-down menu and choose **Point**.
4. Select one of the following methods.



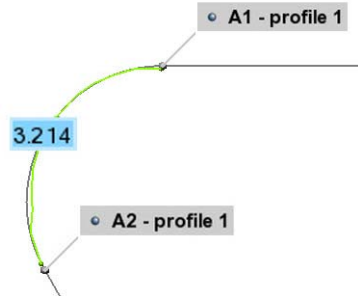
From Objects

1. In the **Method** list, select **From Objects**.
2. In the **Submethod** list, select a submethod.
A corresponding construction submethod is automatically specified according to the type of selected objects.
3. Specify the required parameters, if necessary.
4. Click **Create**.

Example 1: The **Circle Centers** submethod creates a point at the center point of each selected circle. This submethod is available for all curve-based features.



Example 2: The **Profile Gauges** submethod creates characteristic points related to the profile gauge, such as tangency points.



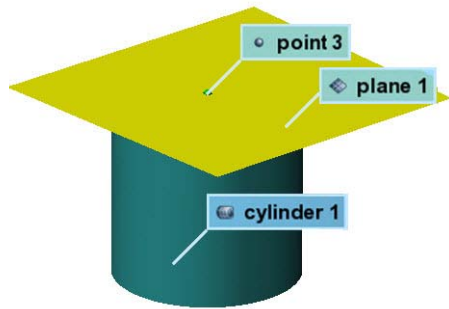
From Intersection

1. In the **Method** list, select **From Intersection**.
2. In the **Submethod** list, select a submethod.

A corresponding construction submethod is automatically specified according to the type of selected objects.

3. Specify the required parameters, if necessary.
4. Click **Create**.

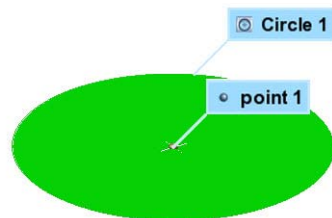
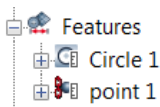
Example: The **Plane and Cylinders** submethod creates a point at the intersection of the selected plane and the axis of each selected cylinder.



Results

The constructed point feature is added to the Tree View, under the Features branch. A chain overlays the icon, indicating its dependent status.

The point feature is displayed in the 3D scene.



Notes

These methods generate dependent point features, which means that if something affects the source objects, it affects the dependent point feature as well. For example, if the source object becomes measured, the dependent point feature becomes measured as well.

Creating point features on cross-sections

Point features can be created on cross-sections for 2D inspection purposes. Nominal point primitives are created on nominal cross-section components. The measured primitives are subsequently extracted on measured cross-section components. A cross-section object must exist and be specified upon creation of these features.

For more information, see [Creating features on cross-sections for 2D inspection](#) on page 216.

The creation procedures are explained in the following subsections.

Creating nominal cross-section point primitives

Objective: Create a nominal cross-section circle primitive.

Required: The project contains a nominal cross-section component created on a CAD Reference object.

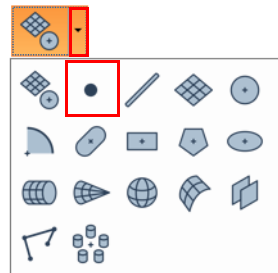
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



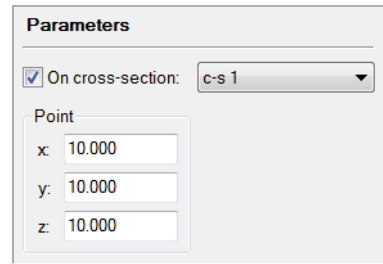
2. Expand the feature type drop-down menu and choose **Point**.
3. Choose one of the following methods.



Numerically

1. In the **Method** list, select **Numerically**.
2. Enable the **On cross-section** parameter and choose a cross-section from the list.
3. Specify the **Point** coordinates to define the feature.
4. Click **Create**.

The point is automatically projected onto the cross-section.



Anchor

1. In the **Method** list, select **Anchor**.
2. Enable the **On cross-section** parameter and choose a cross-section from the list.
3. The interactive anchoring mode is automatically activated. If not, click **Anchor**.
4. In the 3D scene, anchor a point on the cross-section contour at the desired location to create a point.

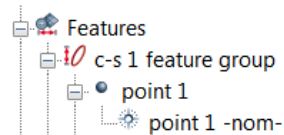
Press and hold SHIFT when anchoring to pick the closest vertex on a cross-section component.

5. Right-click to exit the anchoring mode.



Results

The point feature created on a cross-section, and its nominal primitive, are added to the Tree View, under the Features branch, and also grouped in a cross-section feature group.

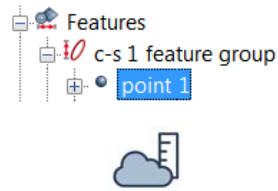


Extracting measured cross-section point primitives

- Objective:** Extract the measured primitive of a cross-section circle using its nominal primitive.
- Required:** The project contains a cross-section point feature with a nominal primitive, and its linked cross-section is measured. For more information about obtaining cross-section measured components, refer to [Measuring the geometry on the part](#) on page 390.

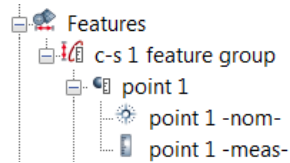
Steps

1. In the Tree View, select the features that will be extracted.
2. Choose *Measure > Extract Measured*.



Results

The measured primitive is added to the cross-section point feature, under the Features branch.



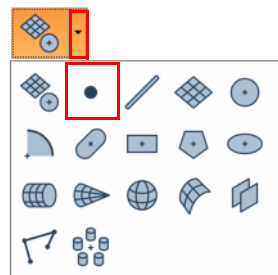
Constructing cross-section point features

Objective: Construct a cross-section point feature using existing objects.

Required: The project contains objects.

Steps

1. In the Tree View, select the objects to use for construction.
2. Choose *Measure > Features > Create*.
The Create Features dialog box opens.
3. Expand the feature type drop-down menu and choose **Point**.
4. Choose one of the following methods.

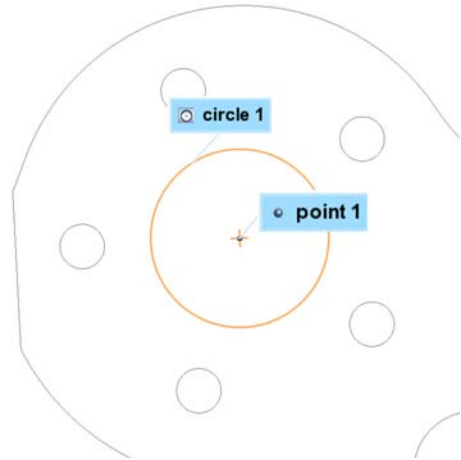


From Objects and From Intersection

1. In the **Method** list, select **From Objects** or **From Intersection**.
2. In the **Submethod** list, select a submethod.
3. Enable the **On cross-section** parameter and choose a cross-section from the list.
4. Specify the required parameters, if necessary.
5. Click **Create**.

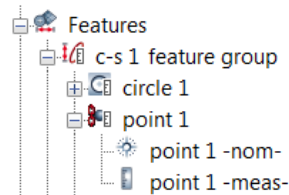
Cross-section point features created from other features are automatically projected onto the specified cross-section. In this example, point 1 was constructed from the center of circle 1.

For more information, see [Constructing point features](#) on page 226.



Results

The cross-section point feature is created and added to the Tree View, under the Features branch. A chain overlays the icon indicating its dependent status.



Notes

These methods generate dependent point features, which means that if something affects the source objects, it affects the dependent point feature as well. For example, if the source object becomes measured, the dependent point feature becomes measured as well.

Creating line features

Line features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal line primitives

Objective: Create a nominal line primitive interactively or using mathematical definitions.

Required: The project contains at least a Reference object, a Data object, a surface-based feature, a curve-based feature, a cross-section-based object, or the information that defines the nominal primitive is available.

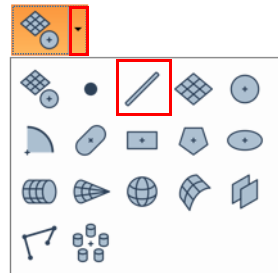
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Line**.
3. Choose one of the following methods.



Numerically

1. In the **Method** list, select **Numerically**.
2. In the **Submethod** list, select **Origin and Orientation**.
3. Specify the required values to define the feature **Origin**, **Orientation**, and **Length**.
4. Click **Create**.

<input type="checkbox"/> On cross-section:		<input type="text"/>
Origin		Orientation
x:	<input type="text"/>	i: <input type="text"/>
y:	<input type="text"/>	j: <input type="text"/>
z:	<input type="text"/>	k: <input type="text"/>
Length:		<input type="text"/>

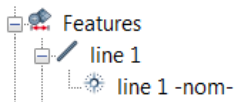
Anchor

1. In the **Method** list, select **Anchor**.
2. In the **Submethod** list, select **Pick 2**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
3. In the 3D scene, anchor two points at the desired location.
Press and hold SHIFT when anchoring to pick the closest vertex on a cross-section component.
4. Right-click to exit the anchoring mode.

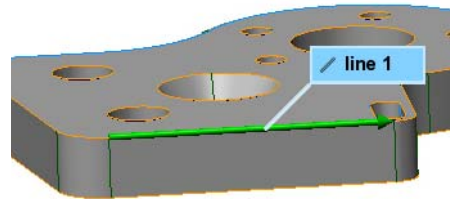


Results

The line feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured line primitives

To obtain the measured primitive of a line feature, the most typical method available is **Probe**.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Probe

Objective: Create the measured line primitive by probing it.

Required: The project contains a nominal line primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the line features to probe.
2. Choose *Measure > Features > Define Measured*.
Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.



The Define Measured Feature Components dialog box opens.



3. In the **Method** list, select **Probe**.
4. In the **Submethod** list, select **Standard**.
5. Constrain the feature to a plane.
 - Select **Use Plane Feature** to use an existing plane in the Tree View, or to probe a new one that will appear in the Tree View. This method is useful when the constraining plane will be used for several objects, for example, thick objects.
 - Select **Probe Local Plane** to probe a new plane. Note that this plane is probed near the feature. It is stored within the feature and does not appear in the Tree View. This method is useful, for example, when probing thin material, such as sheet metal, that could have a significant deviation from a feature to another.
6. Click **Probe**.
7. Probe the constraining plane, if (**Probe New**) or **Probe Local Plane** is specified.
8. Probe the measured line primitive.

Creation method

Method: Probe

Submethod: Standard

Parameters

Thin material

Constraining plane:

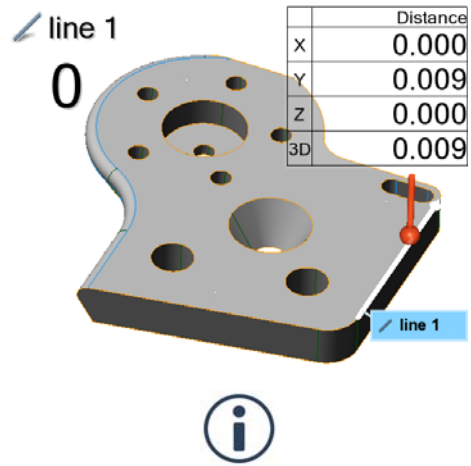
Method: Use Plane Feature

Name: (Probe New)

Fixed number of points: 3

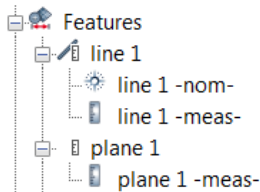
Tips

- If a feature contains a nominal primitive, it is highlighted and centered in the 3D scene for guidance.
- A minimum number of points must be probed to fit a valid primitive.
- The probed primitive is compensated according to the specified compensation method set in the **More** section.
- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the line.

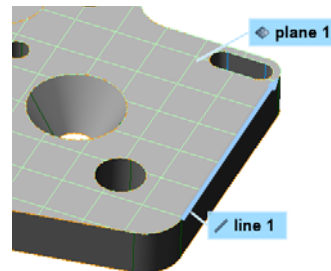


Results

The probed measured primitive is added to the line feature, under the Features branch of the Tree View.



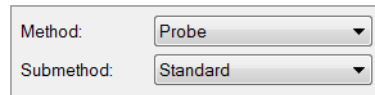
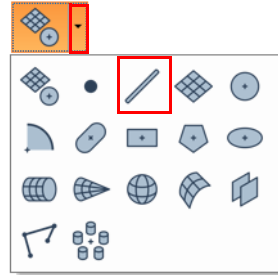
The primitives are displayed in the 3D scene.



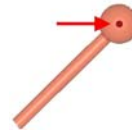
Notes

A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Line**.
- In the **Method** list, select **Probe**.

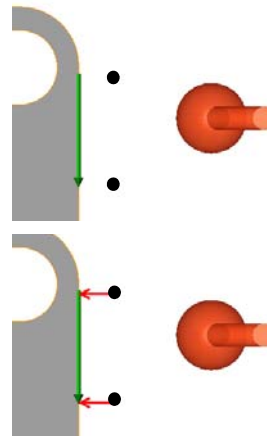


When probing, probed points are collected at the center of the probe tip.



Compensation is necessary to offset the measured points to the point of contact.

When using **Compensation point** as the compensation method, the line is compensated by moving the probe slightly off the surface and ending the probing.



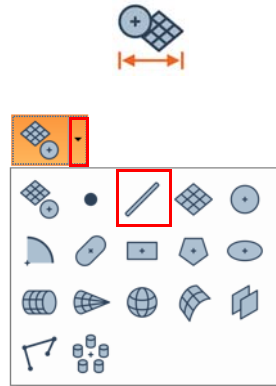
Constructing line features

Objective: Construct a line feature using existing objects.

Required: The project contains existing objects.

Steps

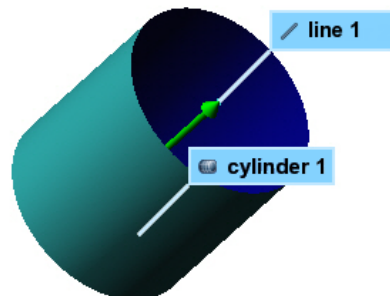
1. In the Tree View, select the objects to use for the construction.
2. Choose *Measure > Features > Create*.
The Create Features dialog box opens.
3. Expand the feature type drop-down menu and choose **Line**.
4. Choose one of the following methods.



From Objects

1. In the **Method** list, select **From Objects**.
2. In the **Submethod** list, select a submethod.
A corresponding construction submethod is automatically specified according to the type of selected objects.
3. Specify the required parameters, if necessary.
4. Click **Create**.

Example: The **Cylinder Axes** submethod creates a line using the axis of each selected cylinder.



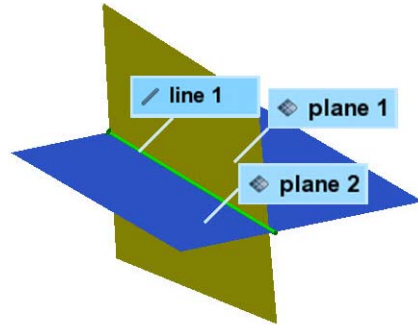
From Intersection

1. In the **Method** list, select **From Intersection**.
2. In the **Submethod** list, select a submethod.

A corresponding construction submethod is automatically specified according to the type of selected objects.

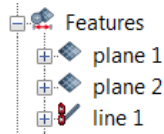
3. Specify the required parameters, if necessary.
4. Click **Create**.

Example: The **2 Planes** submethod creates a line at the intersection of two selected planes.

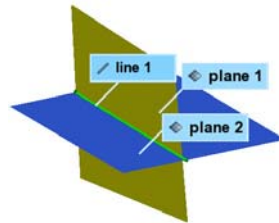


Results

The constructed line feature is added to the Tree View, under the Features branch. A chain overlays the icon indicating its dependent status.



The line feature is displayed in the 3D scene.



Notes

These methods generate dependent line features, which means that if something affects the source objects, it affects the dependent line feature as well. For example, if the source object becomes measured, the dependent line feature becomes measured as well.

To make the line features independent, choose *Measure > Features > Remove Dependencies*.

Creating line features on cross-sections

Line features can be created on cross-sections for 2D inspection purposes. Nominal line primitives are created on nominal cross-section components. The measured primitives are subsequently extracted on measured cross-section components. A cross-section object must exist and be specified upon creation of these features.

For more information, see [Creating features on cross-sections for 2D inspection](#) on page 216.

The creation procedures are explained in the following subsections.

Creating nominal cross-section line primitives

Objective: Create a nominal cross-section line primitive.

Required: The project contains a nominal cross-section component created on a CAD Reference object.

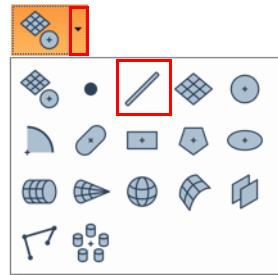
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Line**.
3. Choose one of the following methods.

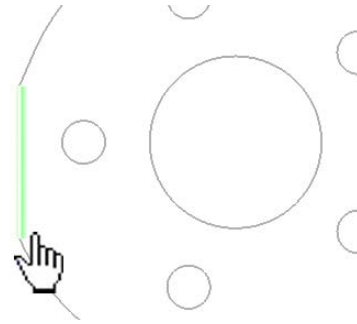


Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. In the **Cross-section** list, select a cross-section.
3. Click **Pick** to launch the interactive picking mode.

Creation method	
Method:	Pick on CAD Model ▼
Parameters	
Cross-section:	c-s 1 (y=20) ▼

- In the 3D scene, move the mouse cursor over the cross-section to select the closest line feature and pick the one desired when it is highlighted.



Numerically

- In the **Method** list, select **Numerically**.
- In the **Submethod** list, select a submethod.
- Enable the **On cross-section** parameter and choose a cross-section from the list.
- Specify the required parameters, if necessary.
- Click **Create**.

The line is automatically projected onto the cross-section.

Creation method	
Method:	Numerically
Submethod:	Origin and Orientation
Parameters	
<input checked="" type="checkbox"/> On cross-section:	c-s 1
Origin	
x:	10.000
y:	10.000
z:	10.000
Orientation	
i:	0.000
j:	0.000
k:	1.000
Length:	1.000

Anchor

- In the **Method** list, select **Anchor**.
- In the **Submethod** list, select **Pick 2**.
- Enable the **On cross-section** parameter and choose a cross-section from the list.
- The interactive anchoring mode is automatically activated. If not, click **Anchor**.
- In the 3D scene, anchor two points on the cross-section contour at the desired location to create a line

Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.

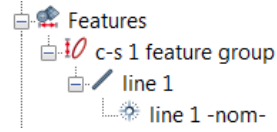
- Right-click to exit the anchoring mode.

Creation method	
Method:	Anchor
Submethod:	Pick 2
Parameters	
<input checked="" type="checkbox"/> On cross-section:	c-s 1



Results

The line features created on cross-sections, and their nominal primitives, are added to the Tree View, under the Features branch, and also grouped in a cross-section feature group.



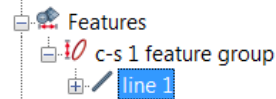
Extracting measured cross-section line primitives

Objective: Extract the measured primitive of a cross-section line using its nominal primitive.

Required: The project contains a cross-section line feature with a nominal primitive, and its linked cross-section is measured. For more information about obtaining cross-section measured components, refer to [Measuring the geometry on the part](#) on page 390.

Steps

1. In the Tree View, select the features that will be extracted.

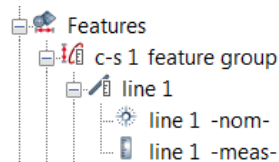


2. Choose *Measure > Extract Measured*.



Results

The measured primitive is added to the cross-section line feature, under the Features branch.



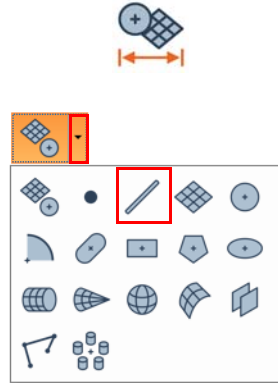
Constructing cross-section line features

Objective: Construct a cross-section line feature using existing objects.

Required: The project contains existing objects.

Steps

1. In the Tree View, select the objects to use for the construction.
2. Choose *Measure > Features > Create*.
The Create Features dialog box opens.
3. Expand the feature type drop-down menu and choose **Line**.
4. Choose one of the following methods.

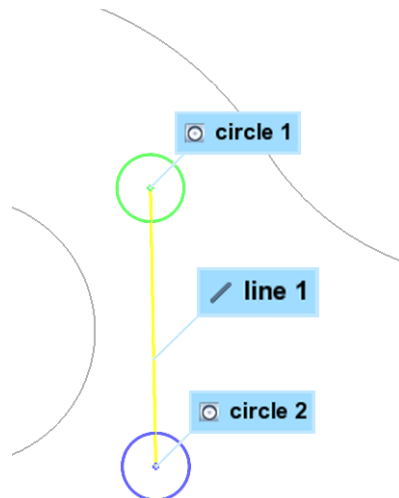


From Objects and From Intersection

1. In the **Method** list, select **From Objects** or **From Intersection**.
2. In the **Submethod** list, select a submethod.
3. Enable the **On cross-section** parameter and choose a cross-section from the list.
4. Specify the required parameters, if necessary.
5. Click **Create**.

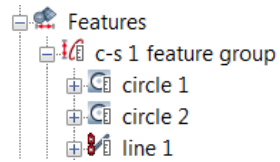
Cross-section line features created from other features are automatically projected onto the specified cross-section, while always preserving their form. In this example, line 1 was constructed from the center points of circle 1 and circle 2.

For more information, see [Constructing line features](#) on page 237.



Results

The constructed line feature is added to the Tree View, under the Features branch, and grouped in the cross-section feature group. A chain overlays the icon indicating its dependent status.



Notes

These methods generate dependent line features, which means that if something affects the source objects, it affects the dependent line feature as well. For example, if the source object becomes measured, the dependent line feature becomes measured as well.

Creating plane features

Plane features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal plane primitives

Objective: Create a nominal plane primitive interactively using Reference object elements or a mathematical definition.

Required: The project contains a Reference object or the information that defines the nominal primitive is available.

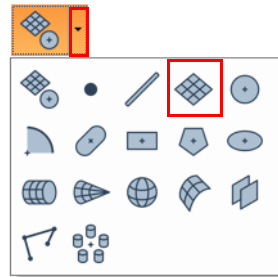
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Plane**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. Click **Pick** to launch the interactive picking mode.
3. In the 3D scene, pick a CAD surface to create the plane.
4. Right-click to exit the picking mode.



Numerically

1. In the **Method** list, select **Numerically**.
2. In the **Submethod** list, select **Point and Normal**.
3. Specify the required values to define the feature **Point** and **Normal**.
4. Click **Create**.

Point		Normal	
x:	<input type="text" value="0.000"/>	i:	<input type="text" value="0.000"/>
y:	<input type="text" value="0.000"/>	j:	<input type="text" value="0.000"/>
z:	<input type="text" value="0.000"/>	k:	<input type="text" value="1.000"/>

Anchor

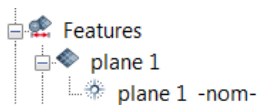
1. In the **Method** list, select **Anchor**.
2. In the **Submethod** list, select **Pick 3**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
3. In the 3D scene, anchor three points on the surface to create a plane.



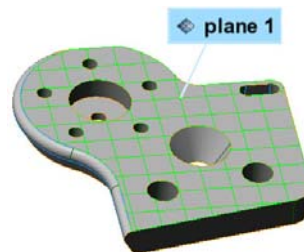
Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.

Results

The plane feature, and its nominal primitive, are added to the Tree View under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured plane primitives

To obtain the measured primitive of a plane feature, the most common methods used are **Extract**, **Probe**, **Extract from Polygonal Models**, and **Fit**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Extract

Objective: Extract the measured plane primitive using the nominal primitive.

Required: The project contains a feature with a nominal primitive and an aligned Data object.

Steps

1. In the Tree View, select the plane feature that will be extracted.



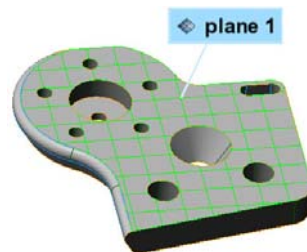
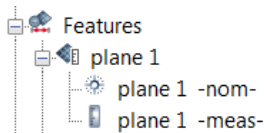
2. Choose *Measure > Extract Measured*.



Results

The measured primitive is added to the plane feature in the Tree View, under the Features branch.

The primitives are displayed in the 3D scene.



Probe

Objective: Create the measured plane primitive by probing it.

Required: The project contains a nominal plane primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the plane feature that will be probed.



2. Choose *Measure > Features > Define Measured*.

Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.



The Define Measured Feature Components dialog box opens.

3. In the **Method** list, select **Probe**.

4. In the **Submethod** list, select **Standard**.

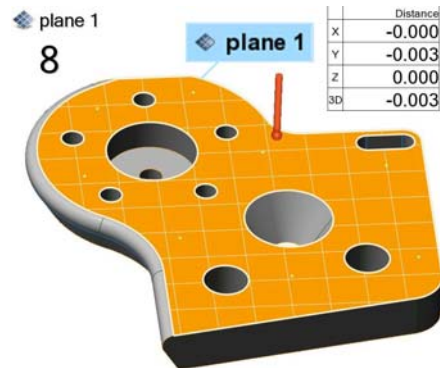
5. Click **Probe**.

6. Probe the measured plane primitive.

Creation method	
Method:	Probe
Submethod:	Standard

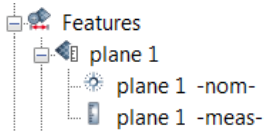
Tips

- If a feature contains a nominal primitive, it is highlighted and centered in the 3D scene for guidance.
- A minimum number of points must be probed to fit a valid primitive.
- The probed primitive is compensated according to the specified compensation method set in the **More** section.
- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the plane.

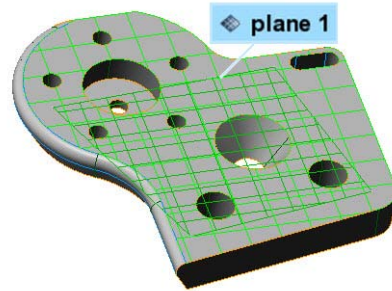


Results

The probed measured primitive is added to the plane feature, under the Features branch of the Tree View.



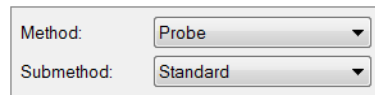
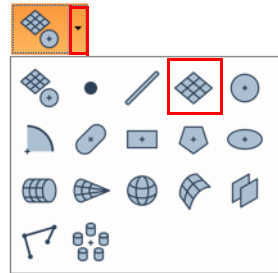
The primitives are displayed in the 3D scene.



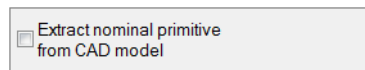
Notes

A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Plane**.
- In the **Method** list, select **Probe**.

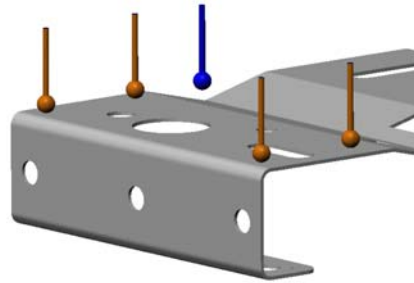


If the probing device is already aligned to a CAD model, the nominal primitive can be extracted at the same time by activating the **Extract nominal primitive from CAD model** option.



When probing, probed points are collected at the center of the probe tip. Compensation is necessary to offset the measured points to the point of contact.

When using **Compensation point** as the compensation method, a plane is compensated by lifting the probe slightly off the surface and ending the probing.



Fit

Objective: Create measured plane primitives interactively using data elements.

Required: The project contains a Data object.

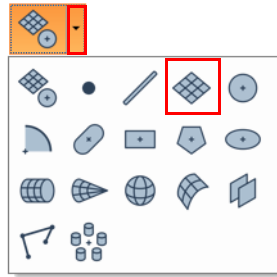
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Plane**.



3. In the **Method** list, select **Fit**.
4. In the **Submethod** list, select **Pick, then Fit**.
5. Specify the **Max distance**.
6. Select a **Fit type**.
7. Click **Pick** to launch the interactive picking mode.

Creation method

Method: Fit

Submethod: Pick, then Fit

Parameters

Max distance: 4.000

Fit type:

Best-fit

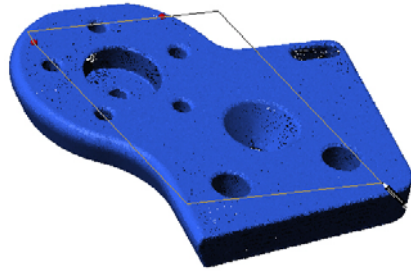
Min

Max

Trim using reference object elements

- In the 3D scene, anchor three points.

A preview plane is shown as points are anchored. If the preview plane does not look oriented as desired, restart the picking session by pressing ESC to cancel the current picking session and click **Pick** to start a new picking session.



- After picking the third point, a plane is fitted to the Data object using the anchored plane as an initial guess to start the fitting process.

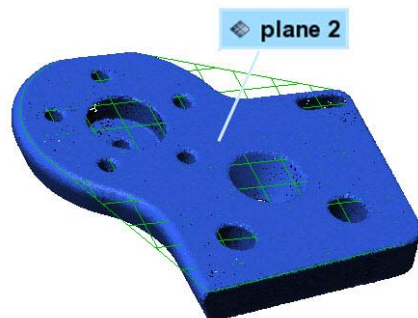
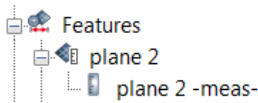
Tips

To use the **Fit** method to define the measured primitive of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The measured primitive is added to the plane feature, under the Features branch of the Tree View.

The primitive is displayed in the 3D scene.



Extract from Polygonal Models

Objective: Create measured plane primitives interactively using a polygonal model.

Required: The project contains a polygonal model.

Steps

- Choose *Measure > Features > Extract from Polygonal Models*.

The Extract Primitives from Polygonal Models dialog box opens.

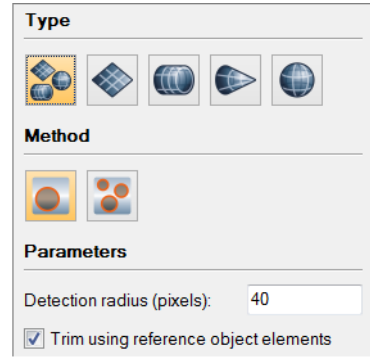


2. Specify the feature **Type**.

The **Automatic** type is specified by default, which automatically determines the feature type that adjusts better to the area selected.

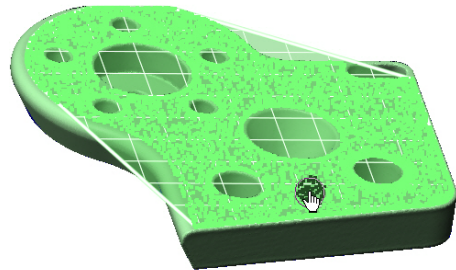
3. Specify the **Method**.

The **Single Detection Zone** method is selected by default. The **Multiple Detection Zones** method is useful when all areas that can be used to extract the feature primitive are not contiguous.



4. In the 3D scene, using the circular detection zone around the mouse pointer, click an area on the polygonal model that is part of the desired feature.

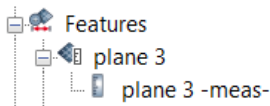
*It is possible to increase or decrease the size of the detection zone at any time in the **Parameters** subsection, or using the mouse wheel.*



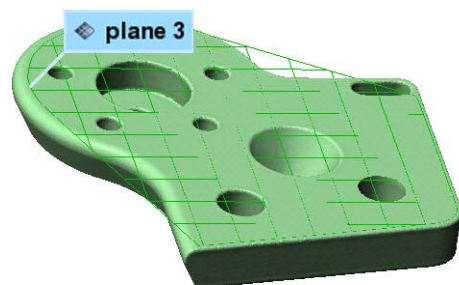
5. Click **Confirm**.

Results

The plane feature, and its measured primitive, are added to the Tree View under the Features branch.



The primitive is displayed in the 3D scene.



Notes

The **Extract from Polygonal Models** method can also be used to create nominal primitives when using a polygonal model as the Reference object.

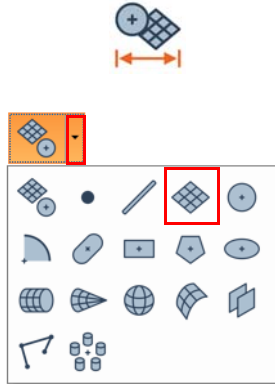
Constructing plane features

Objective: Construct a plane feature using existing objects.

Required: The project contains existing objects.

Steps

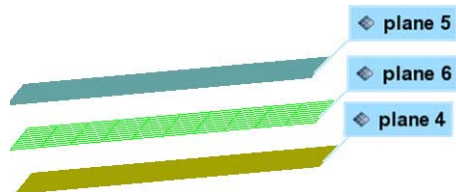
1. In the Tree View, select the objects required for the construction.
2. Choose *Measure > Features > Create*.
The Create Features dialog box opens.
3. Expand the feature type drop-down menu and choose **Plane**.
4. Choose one of the following methods.



From Objects

1. In the **Method** list, select **From Objects**.
2. In the **Submethod** list, select a submethod.
A corresponding construction submethod is automatically specified according to the type of selected objects.
3. Specify the required parameters, if necessary.
4. Click **Create**.

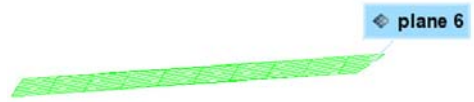
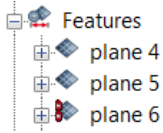
Example: The **Average of Planes** submethod creates a plane by averaging the selected planes.



Results

The constructed plane feature is added to the Tree View, under the Features branch. A chain overlays the icon indicating its dependent status.

The plane feature is displayed in the 3D scene.



Creating circle features

Circle features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal circle primitives

Objective: Create a nominal circle primitive interactively using Reference object elements or a mathematical definition.

Required: The project contains a Reference object, or the information that describes the nominal primitive is available.

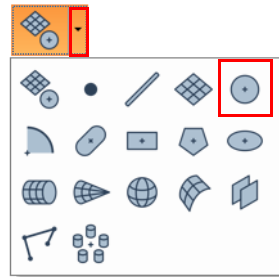
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

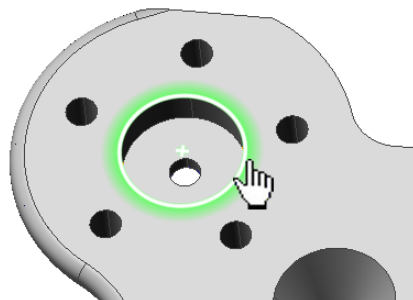


2. Expand the feature type drop-down menu and choose **Circle**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. The interactive picking mode is automatically activated. If not, click **Pick**.
3. In the 3D scene, pick a circle. Two approaches can be used:
 - Move the mouse cursor on the CAD model surface to select the closest circle feature and pick the one desired when it is highlighted.
 - Move the mouse cursor in the void within the hole to select the first found circle based on the current 3D scene view, then pick.



Numerically

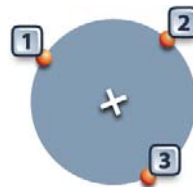
1. In the **Method** list, select **Numerically**.
2. Specify the required values to define the feature **Center, Orientation, and Radius**.
3. Click **Create**.

Center		Orientation	
x:	10.000	i:	0.000
y:	10.000	j:	0.000
z:	10.000	k:	1.000
Radius:		1.000	

Anchor

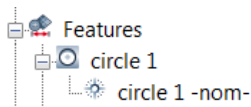
1. In the **Method** list, select **Anchor**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
2. In the 3D scene, anchor three points on a hole boundary to create a circle.

Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.

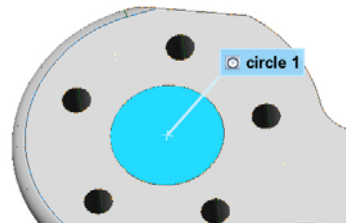


Results

The circle feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured circle primitives

To obtain the measured primitive of a circle feature, the most common methods used are **Extract**, **Probe**, and **Fit**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Extract

Objective: Extract the measured circle primitive using the nominal primitive.

Required: The project contains a circle feature with a nominal primitive and an aligned Data object.

Steps

1. In the Tree View, select the features that will be extracted.



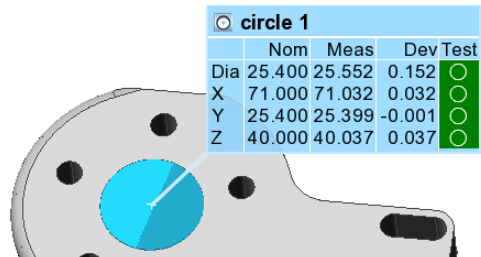
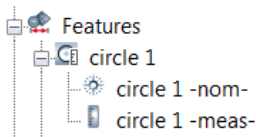
2. Choose *Measure > Extract Measured*.



Results

The measured primitive is added to the circle feature, under the Features branch of the Tree View.

The primitives are displayed in the 3D scene.



Probe

Objective: Create the measured circle primitive by probing it, along with its constraining plane, if required.

Required: The project contains a nominal circle primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the features that will be probed.



2. Choose *Measure > Features > Define Measured*.

Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*. The Define Measured Feature Components dialog box opens.



3. In the **Method** list, select **Probe**.

4. In the **Submethod** list, select **Standard**.

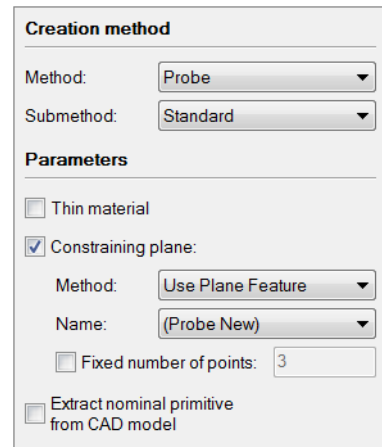
5. Specify a constraining plane.

- Select **Use Plane Feature** to use an existing plane in the Tree View, or to probe a new one that will appear in the Tree View.
- Select **Probe Local Plane** to probe a new plane. Note that this plane is probed near the feature. It is stored within the feature and does not appear in the Tree View.

6. Click **Probe**.

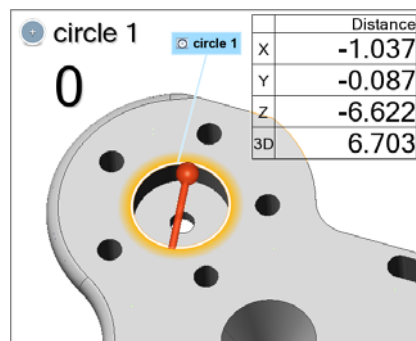
7. Probe the constraining plane, if (**Probe New**) or **Probe Local Plane** is specified.

8. Probe the measured circle primitive.



Tips

- If a feature contains a nominal primitive, it is highlighted and centered in the 3D scene for guidance.
- A minimum number of points must be probed to fit a valid primitive.
- The probed primitive is compensated according to the specified compensation method set in the **More** section.

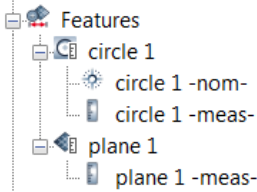


- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the circle.

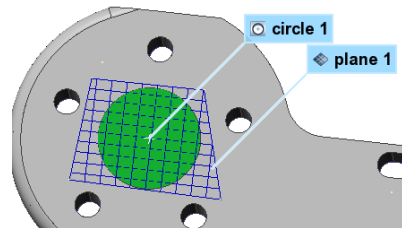


Results

The probed measured primitive is added to the circle feature, under the Features branch of the Tree View.



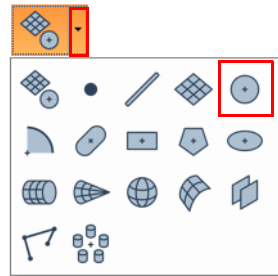
The primitives are displayed in the 3D scene.



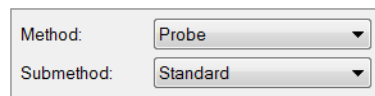
Notes

A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

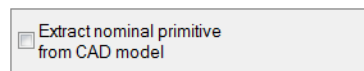
- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Circle**.



- In the **Method** list, select **Probe**.



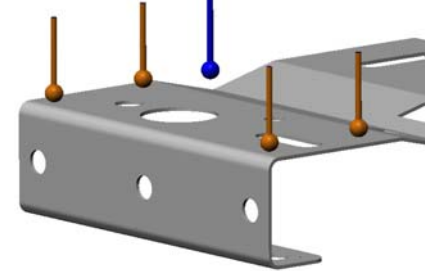
If the probing device is already aligned to a CAD model, the nominal primitive can be extracted at the same time by activating the **Extract nominal primitive from CAD model** item.



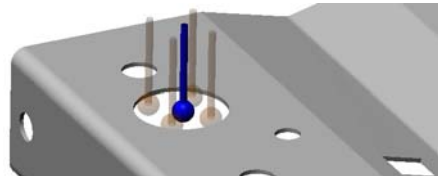
When probing, probed points are collected at the center of the probe tip. Compensation is necessary to offset the measured points to the point of contact.



When using **Compensation point** as the compensation method, a plane is compensated by lifting the probe slightly off the surface and ending the probing.



When using **Compensation point** as the compensation method, a circle is compensated by moving the probe inward (hole) or outward (pin) from the feature boundary, then ending the probing.



Fit

Objective: Create measured circle primitives interactively using data elements.

Required: The project contains a Data object.

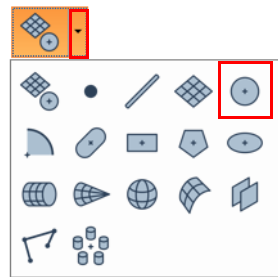
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Circle**.



- In the **Method** list, select **Fit**.
- In the **Submethod** list, select **Fit to Elements**.
- Select the **Fit type**.

*It is recommended to set the **Fit type** to **Min** for a hole and **Max** for a pin.*

- Under **Constraints**, select **Plane**, then select a constraining plane, if required.

It is recommended to use a constraining plane to create a feature that is correctly oriented in space. The plane should be created prior to the fitting.

- Under **Filtering**, adjust the filters based on the element selection.

*It is recommended to remove the **Max angle** filter if no data is selected on the walls of the hole.*

- Choose *Select > Elements > Interactively*.

- In the 3D scene, select the elements that will be used to fit the circle.

See [Selecting elements](#) on page 63 for information on how to select elements interactively.

- Click **Create** to fit a primitive to the selected elements.

Creation method

Method:

Submethod:

Parameters

Fit type:

Best-fit

Min

Max

Constraints:

Plane:

Center:

More ▾

Filtering ▲

Subsampling step:

Max angle:

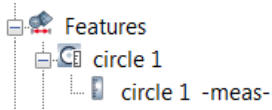


Tips

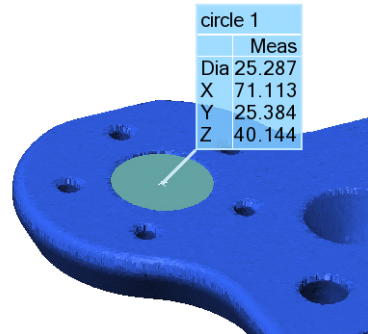
To use the **Fit** method to define the measured primitive of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The measured primitive is added to the circle feature, under the Features branch of the Tree View.



The primitive is displayed in the 3D scene.



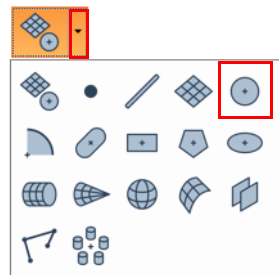
Constructing circle features

Objective: Construct a circle feature using existing objects.

Required: The project contains existing objects.

Steps

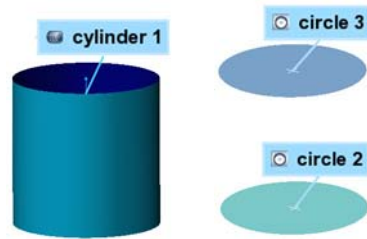
1. In the Tree View, select the objects to use for the construction.
2. Choose *Measure > Features > Create*.
The Create Features dialog box opens.
3. Expand the feature type drop-down menu and choose **Circle**.
4. Choose one of the following methods.



From Objects

1. In the **Method** list, select **From Objects**
2. In the **Submethod** list, select a submethod.
A corresponding construction submethod is automatically specified according to the type of selected objects.
3. Specify the required parameters, if necessary.
4. Click **Create**.

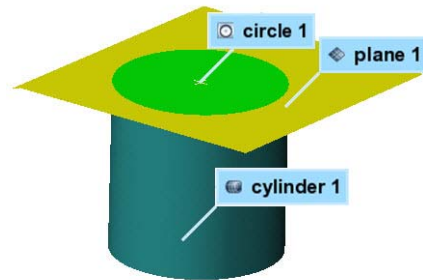
Example: The **Cylinders** submethod creates a circle at each end of the selected cylinder, having the orientation of the cylinder.



From Intersection

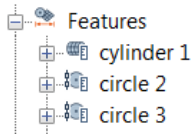
1. In the **Method** list, select **From Intersection**.
2. In the **Submethod** list, select a submethod.
A corresponding construction submethod is automatically specified according to the type of selected objects.
3. Specify the required parameters, if necessary.
4. Click **Create**.

Example: The **Plane and Cylinders** submethod creates a circle centered at the intersection of a selected plane and the axis of each selected cylinder, oriented using the normal of the plane and with a radius equal to the radius of the cylinder.

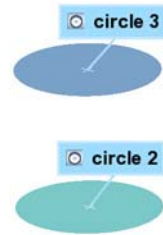


Results

The constructed circle features are added to the Tree View, under the Features branch. A chain overlays the icon, indicating its dependent status.



The circle features are displayed in the 3D scene.



Notes

These methods generate dependent circle features, which means that if something affects the source objects, it affects the dependent circle feature as well. For example, if the source object becomes measured, the dependent circle feature becomes measured as well.

To make the circle features independent, choose *Measure > Features > Remove Dependencies*.

Creating circle features on cross-sections

Circle features can be created on cross-sections for 2D inspection purposes. Nominal circle primitives are created on nominal cross-section components. The measured primitives are subsequently extracted on measured cross-section components. A cross-section object must exist and be specified upon creation of these features.

For more information, see [Creating features on cross-sections for 2D inspection](#) on page 216.

The creation procedures are explained in the following subsections.

Creating nominal cross-section circle primitives

Objective: Create a nominal cross-section circle primitive.

Required: The project contains a nominal cross-section component created on a CAD Reference object.

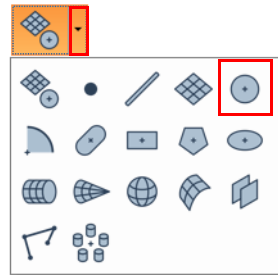
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

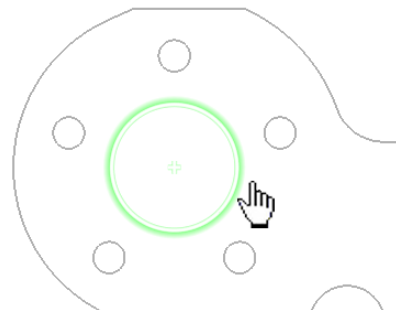


2. Expand the feature type drop-down menu and choose **Circle**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. Enable the **On cross-section** parameter and choose from the list of cross-sections.
3. The interactive picking mode is automatically activated. If not, click **Pick**.
4. In the 3D scene, move the mouse cursor over the cross-section to select the closest circle feature and pick the one desired when it is highlighted.



Numerically

1. In the **Method** list, select **Numerically**.
2. Enable the **On cross-section** parameter and choose from the list of cross-sections.
3. Specify the required values to define the feature **Center** and **Radius**.
4. Click **Create**.

The circle is automatically projected onto the cross-section.

Creation method

Method: Numerically

Parameters

On cross-section: c-s 1 (y=20)

Center

x: 10.000

y: 10.000

z: 10.000

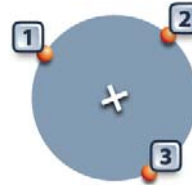
Radius: 1.000

Anchor

1. In the **Method** list, select **Anchor**.
 2. Enable the **On cross-section** parameter and choose from the list of cross-sections.
- The interactive anchoring mode is automatically activated. If not, click **Anchor**.
3. In the 3D scene, anchor three points on the cross-section contour at the desired location to create a circle.

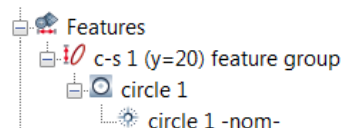
Press and hold SHIFT when anchoring to pick the closest vertex on a cross-section component.

4. Right-click to exit the anchoring mode.



Results

The circle feature created on a cross-section, and its nominal primitive, are added to the Tree View under the Features branch, and also grouped in a cross-section feature group.



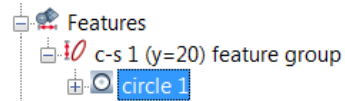
Extracting measured cross-section circle primitives

Objective: Extract the measured primitive of a cross-section circle using its nominal primitive.

Required: The project contains a cross-section circle feature with a nominal primitive, and its linked cross-section is measured. For more information about obtaining cross-section measured components, refer to [Measuring the geometry on the part](#) on page 390.

Steps

1. In the Tree View, select the features that will be extracted.

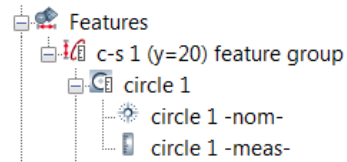


2. Choose *Measure > Extract Measured*.



Results

The measured primitive is added to the cross-section circle feature, under the Features branch.



Constructing cross-section circle features

Objective: Construct a cross-section circle feature using existing objects.

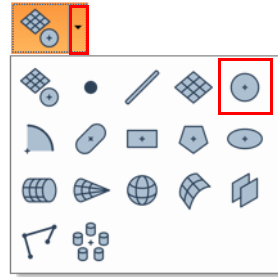
Required: The project contains objects.

Steps

1. In the Tree View, select the objects to use for the construction.
2. Choose *Measure > Features > Create*.
The Create Features dialog box opens.



- Expand the feature type drop-down menu and choose **Circle**.
- Choose one of the following methods.

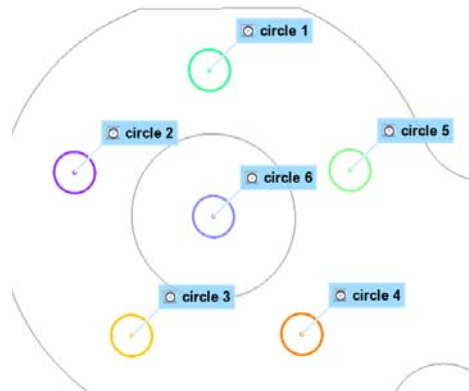


From Objects and From Intersection

- In the **Method** list, select **From Objects** or **From Intersection**.
- In the **Submethod** list, select a submethod.
- Enable the **On cross-section** parameter and choose a cross-section from the list.
- Specify the required parameters, if necessary.
- Click **Create**.

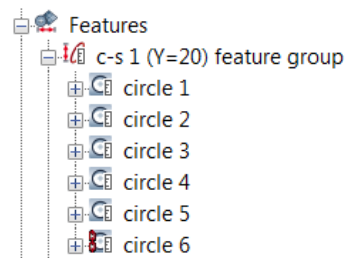
*Cross-section circle features created from other features are automatically projected onto the specified cross-section, while always preserving their form. In this example, **circle 6** was constructed from the average of **circle 1** to **circle 5**.*

For more information, see [Constructing circle features](#) on page 261.



Results

The constructed cross-section circle is added to the Tree View, under the Features branch, in the cross-section feature group. A chain overlays the icon, indicating its dependent status.



Notes

These methods generate dependent circle features, which means that if something affects the source objects, it affects the dependent circle feature as well. For example, if the source object becomes measured, the dependent circle feature becomes measured as well.

Creating arc features

An arc consists of a subsection of a circle. Arc features can only be created on cross-sections, typically with a nominal primitive, using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal arc primitives

Objective: Create a nominal arc primitive interactively using a Reference object or a mathematical definition.

Required: The project contains a cross-section, a Reference object, or the information that describes the nominal primitive is available.

Steps

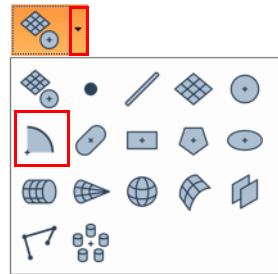
1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



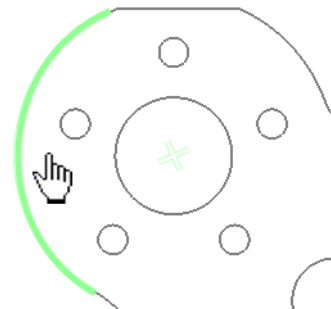
2. Expand the feature type drop-down menu and choose **Arc**.

3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. In the **Cross-section** list, select a cross-section.
3. The interactive picking mode is automatically activated. If not, click **Pick**.
4. In the 3D scene, move the mouse pointer over the cross-section contour, close to the area that defines an arc, and pick when it is highlighted.
5. Right-click to exit the picking mode.



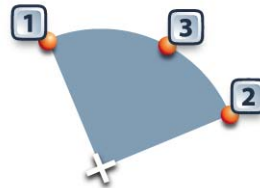
Numerically

1. In the **Method** list, select **Numerically**.
2. In the **Cross-section** list, select a cross-section.
3. Specify the required values to define the feature **Center**, **Radius**, **Angle reference axis**, **Start angle**, and **End angle**.
4. Click **Create**.

Creation method	
Method:	Numerically
Parameters	
Cross-section:	c-s 1 (Y=20)
Center	
X:	0.000
Y:	0.000
Z:	0.000
Radius:	1.000
Angle reference axis:	X
Start angle:	0.000
End angle:	180.000

Anchor

1. In the **Method** list, select **Anchor**.
2. In the **Cross-section** list, select a cross-section.
3. The interactive anchoring mode is automatically activated. If not, click **Anchor**.
4. In the 3D scene, anchor three points on the cross-section contour, at the desired location, to create an arc.

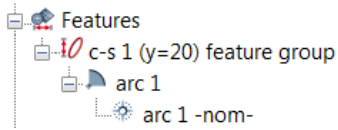


Press and hold SHIFT when anchoring to pick the closest vertex on a cross-section component.

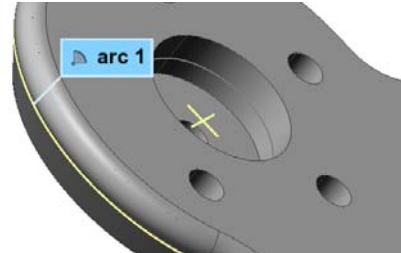
5. Right-click to exit the anchoring mode.

Results

The arc feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Notes

Creating arcs numerically requires the following information:

- **Angle reference axis:** Axis used as a reference for specifying the start and end angles of the arc. This axis is projected onto the cross-section plane.
- **Start angle:** Start angle of the arc, measured counterclockwise from the angle reference axis.
- **End angle:** End angle of the arc, measured counterclockwise from the angle reference axis.

Obtaining measured arc primitives

To obtain the measured primitive of an arc feature, the method used is **Extract from Cross-Section**. The measured primitive of the arc is extracted from the vertices of the measured components of its linked cross-section.

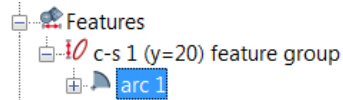
Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Objective: Extract the measured arc primitive using the nominal primitive.

Required: The project contains an arc feature with a nominal primitive and its linked cross-section is measured.

Steps

1. In the Tree View, select the features that will be extracted.

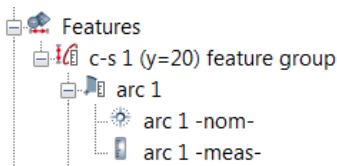


2. Choose *Measure > Extract Measured*.

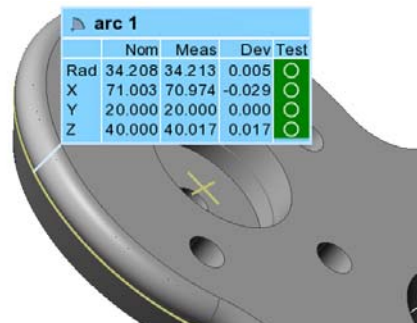


Results

The measured primitive is added to the arc feature, under the Features branch of the Tree View.



The primitives are displayed in the 3D scene.



Creating slot features

Slot features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal slot primitives

Objective: Create a nominal slot primitive interactively using Reference object elements or a mathematical definition.

Required: The project contains a Reference object, or the information that defines the nominal primitive is available.

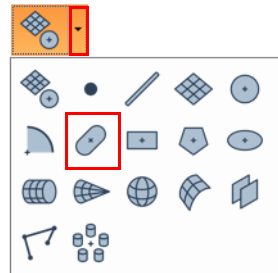
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

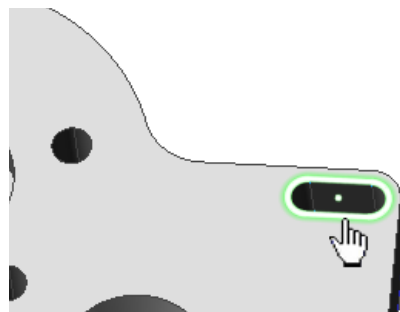


2. Expand the feature type drop-down menu and choose **Slot**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. The interactive picking mode is automatically activated. If not, click **Pick**.
3. In the 3D scene, pick a slot. Two approaches can be used:
 - Move the mouse cursor on the CAD model surface to select the closest slot feature and pick the one desired when it is highlighted.
 - Move the mouse cursor in the void within the hole to select the first found slot based on the current 3D scene view, then pick.



Numerically

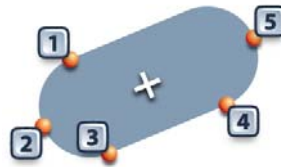
1. In the **Method** list, select **Numerically**.
2. Specify the values to define the feature **Center**, **Axis orientation**, **Length orientation**, **Length**, and **Width**.
3. Click **Create**.

<input type="checkbox"/> On cross-section:	c-s 1 (y=20)
Center	
x:	0.000
y:	0.000
z:	0.000
Axis orientation	
i:	0.000
j:	0.000
k:	1.000
Length orientation	
i:	1.000
j:	0.000
k:	0.000
Length:	1.000
Width:	0.500

Anchor

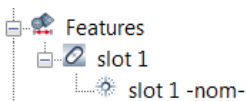
1. In the **Method** list, select **Anchor**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
2. In the 3D scene, anchor three points on one arc boundary and two points on the other arc boundary.

Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.

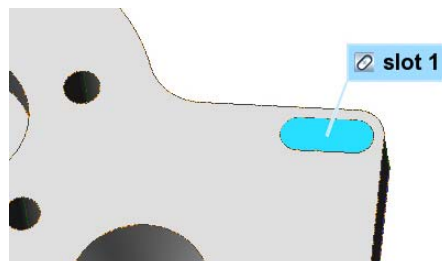


Results

The slot feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured slot primitives

To obtain the measured primitive of a slot feature, the most common methods used are **Extract**, **Probe**, and **Fit**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

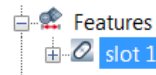
Extract

Objective: Extract the measured slot primitive using the nominal primitive.

Required: The project contains a slot feature with a nominal primitive and an aligned Data object.

Steps

1. In the Tree View, select the features that will be extracted.

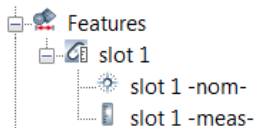


2. Choose *Measure > Extract Measured*.

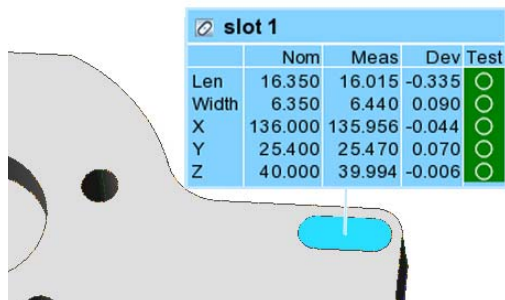


Results

The measured primitive is added to the slot feature in the Tree View, under the Features branch.



The primitives are displayed in the 3D scene.



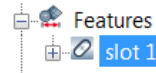
Probe

Objective: Create the measured slot primitive by probing it, along with its constraining plane if required.

Required: The project contains a nominal slot primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the slot feature that will be probed.



2. Choose *Measure > Features > Define Measured*.

Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.



The Define Measured Feature Components dialog box opens.

3. In the **Method** list, select **Probe**.
4. In the **Submethod** list, select **Arcs Only**.
5. Specify a constraining plane.
 - Select **Use Plane Feature** to use an existing plane in the Tree View, or to probe a new one that will appear in the Tree View.
 - Select **Probe Local Plane** to probe a new plane. Note that this plane is probed near the feature. It is stored within the feature and does not appear in the Tree View.
6. Click **Probe**.
7. Probe the constraining plane, if (**Probe New**) or **Probe Local Plane** is specified.
8. Probe the measured slot primitive.

Creation method

Method: Probe

Submethod: Arcs Only

Parameters

Thin material

Constraining plane:

Method: Use Plane Feature

Name: (Probe New)

Fixed number of points: 3

Extract nominal primitive from CAD model

Tips

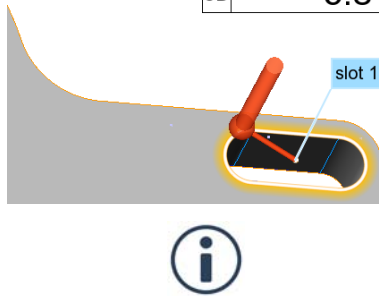
- If a feature contains a nominal primitive, it is highlighted and centered in the 3D scene for guidance.
- A minimum number of points must be probed for each arc to fit a valid primitive.
- The probed primitive is compensated according to the specified compensation method set in the **More** section.
- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the slot.

slot 1

0

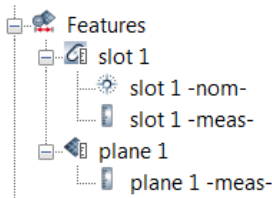
Arc 1/2

	Distance
X	-5.187
Y	1.189
Z	-2.334
3D	5.811

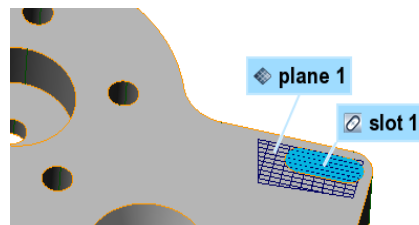


Results

The probed measured slot primitive and its constraining plane are added to the Tree View, under the Features branch.



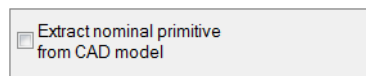
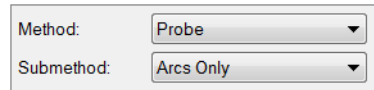
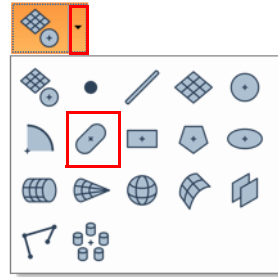
The probed slot feature components are displayed in the 3D scene.



Notes

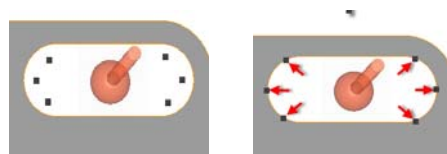
A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Slot**.
- In the **Method** list, select **Probe**.

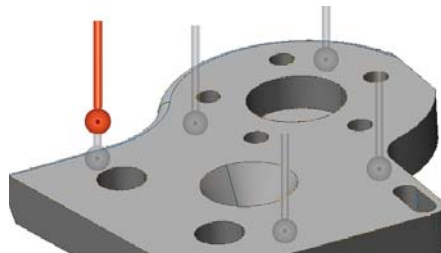


If the probing device is already aligned to a CAD model, the nominal primitive can be extracted at the same time by activating the **Extract nominal primitive from CAD model** item.

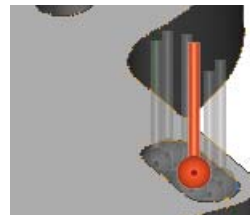
When probing, probed points are collected at the center of the probe tip. Compensation is necessary to offset the measured points to the point of contact.



When using **Compensation point** as the compensation method, a plane is compensated by lifting the probe slightly off the surface and ending the probing.



When using **Compensation point** as the compensation method, a slot is compensated by moving the probe inward (hole) or outward (pin) from the feature boundary, then ending the probing.



Fit

Objective: Create measured slot primitives interactively using data elements.

Required: The project contains a Data object.

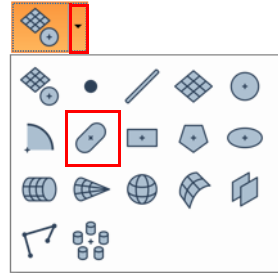
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Slot**.



3. In the **Method** list, select **Fit**.

4. In the **Submethod** list, select **Fit to Elements**.

5. Select the **Fit type**.

*It is recommended to set the **Fit type** to **Min** for a hole and **Max** for a pin.*

6. Under **Constraints**, select **Plane**, then select a constraining plane from the list.

It is recommended to use a constraining plane to create the feature so that it is correctly oriented in space. The plane should be created prior to the fitting.

7. Under **Filtering**, adjust the filters based on the element selection.

*It is recommended to remove the **Max angle** filter if no data is selected on the walls of the hole.*

8. Choose *Select > Elements > Interactively*.



Creation method

Method:

Submethod:

Parameters

Fit type:

Best-fit

Min

Max

Constraints:

Plane:

More ⌵

Filtering ⌵

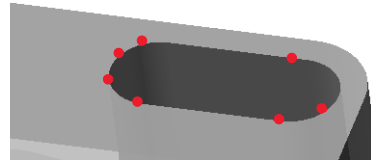
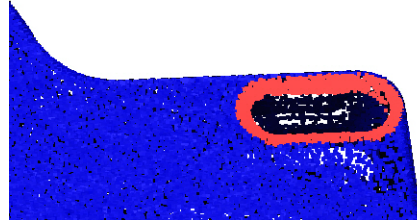
Subsampling step:

Max angle:

- In the 3D scene, select the elements that will be used to fit the slot.

See [Selecting elements on page 63](#) for information on how to select elements interactively.

- Click **Create** to fit a primitive to the selected elements.

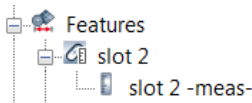


Tips

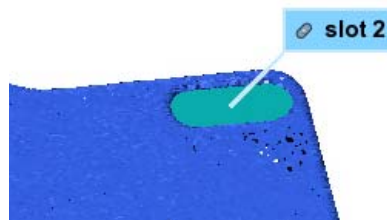
To use the **Fit** method to define the measured primitive of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The measured primitive is added to the slot feature, under the Features branch of the Tree View.



The primitive is displayed in the 3D scene.



Creating slot features on cross-sections

Slot features can be created on cross-sections for 2D inspection purposes. Nominal slot primitives are created on nominal cross-section components. The measured primitives are subsequently extracted on measured cross-section components. A cross-section object must exist and be specified upon creation of these features.

For more information, see [Creating features on cross-sections for 2D inspection](#) on page 216.

The creation procedures are explained in the following subsections.

Creating nominal cross-section slot primitives

Objective: Create a nominal cross-section slot primitive.

Required: The project contains a nominal cross-section component created on a CAD Reference object.

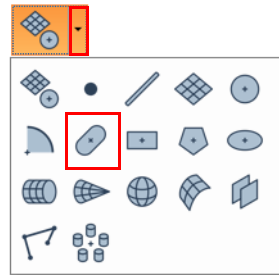
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

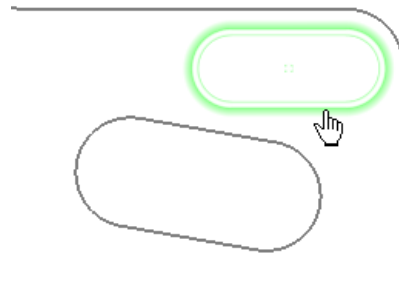


2. Expand the feature type drop-down menu and choose **Slot**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. Enable the **On cross-section** parameter and choose a cross-section from the list.
3. The interactive picking mode is automatically activated. If not, click **Pick**.
4. In the 3D scene, move the mouse cursor over the cross-section to select the closest slot feature and pick the one desired when it is highlighted.
5. Right-click to exit the picking mode.



Numerically

1. In the **Method** list, select **Numerically**.
2. Enable the **On cross-section** parameter and choose a cross-section from the list.
3. Specify the values to define the feature **Center**, **Length orientation**, **Length**, and **Width**.
4. Click **Create**.

The slot is automatically projected onto the cross-section.

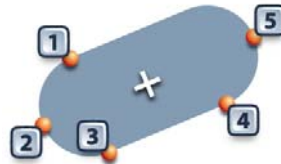
Parameters	
<input checked="" type="checkbox"/> On cross-section:	c-s 2 (y=16.128)
Center	
x:	0.000
y:	0.000
z:	0.000
Length orientation	
i:	1.000
j:	0.000
k:	0.000
Length:	1.000
Width:	0.500

Anchor

1. In the **Method** list, select **Anchor**.
2. Enable the **On cross-section** parameter and choose a cross-section from the list.
3. The interactive anchoring mode is automatically activated. If not, click **Anchor**.
4. In the 3D scene, anchor three points on the cross-section contour at the desired location to define the first arc, then two points to define the second arc to create a slot.

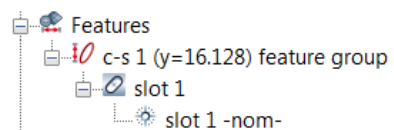
Press and hold SHIFT when anchoring to pick the closest vertex on a cross-section component.

5. Right-click to exit the anchoring mode.



Results

The slot feature created on a cross-section and its nominal primitive are added to the Tree View, under the Features branch, and is also grouped in a cross-section feature group.



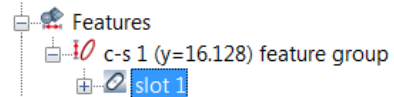
Extracting measured cross-section slot primitives

Objective: Extract the measured primitive of a cross-section slot using its nominal primitive.

Required: The project contains a cross-section slot feature with a nominal primitive, and its linked cross-section is measured. For more information about obtaining cross-section measured components, refer to [Measuring the geometry on the part](#) on page 390.

Steps

1. In the Tree View, select the features that will be extracted.

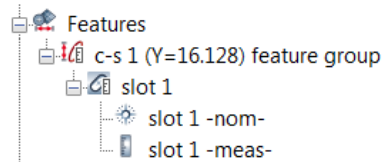


2. Choose *Measure > Extract Measured*.



Results

The measured primitive is added to the cross-section slot feature, under the Features branch.



Creating rectangle features

Rectangle features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal rectangle primitives

Objective: Create a nominal rectangle primitive interactively using Reference object elements or a mathematical definition.

Required: The project contains a Reference object, or the information that describes the nominal primitive is available.

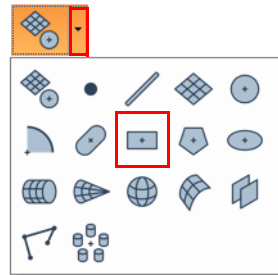
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

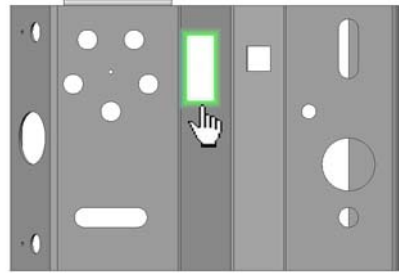


2. Expand the feature type drop-down menu and choose **Rectangle**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. The interactive picking mode is automatically activated. If not, click **Pick**.
3. In the 3D scene, pick a rectangle. Two approaches can be used:
 - Move the mouse cursor on the CAD model surface to select the closest rectangle feature and pick the one desired when it is highlighted.
 - Move the mouse cursor in the void within the hole to select the first found rectangle based on the current 3D scene view, then pick.



Numerically

1. In the **Method** list, select **Numerically**.
2. Specify the required values to define the feature **Center**, **Axis orientation**, **Length orientation**, **Length**, **Width**, and **Corner radius**.
*To create a rectangle with sharp corners, set the **Corner radius** to zero.*
3. Click **Create**.

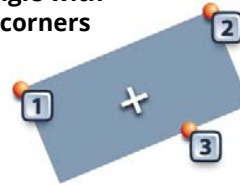
Center		Axis orientation	
x:	<input type="text" value="0.000"/>	i:	<input type="text" value="0.000"/>
y:	<input type="text" value="0.000"/>	j:	<input type="text" value="0.000"/>
z:	<input type="text" value="0.000"/>	k:	<input type="text" value="1.000"/>
Length orientation			
i:	<input type="text" value="1.000"/>		
j:	<input type="text" value="0.000"/>		
k:	<input type="text" value="0.000"/>		
Length:	<input type="text" value="1.000"/>		
Width:	<input type="text" value="0.500"/>		
Corner radius:	<input type="text" value="0.000"/>		

Anchor

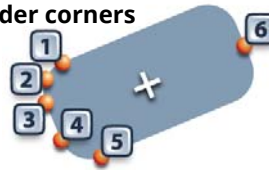
1. In the **Method** list, select **Anchor**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
2. Enable the **Rounded corners** parameters, if required.
3. In the 3D scene, anchor the required number of points on a rectangle boundary to create a rectangle.

Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.

Rectangle with sharp corners



Rectangle with rounder corners

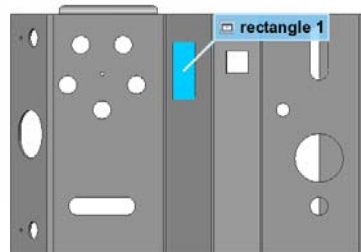


Results

The rectangle feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured rectangle primitives

To obtain the measured primitive of a rectangle feature, the most common methods used are **Extract**, **Probe**, and **Fit**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

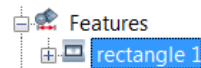
Extract

Objective: Extract the measured rectangle primitive using the nominal primitive.

Required: The project contains a rectangle feature with a nominal primitive and an aligned Data object.

Steps

1. In the Tree View, select the features that will be extracted.

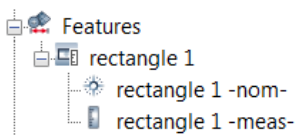


2. Choose *Measure > Extract Measured*.

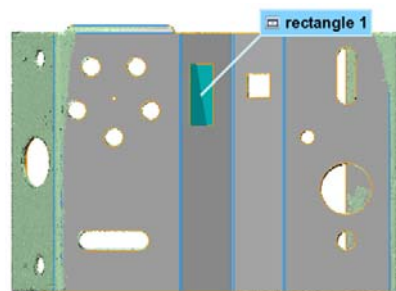


Results

The measured primitive is added to the rectangle feature in the Tree View, under the Features branch.



The primitives are displayed in the 3D scene.



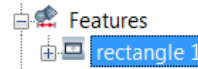
Probe

Objective: Create the measured rectangle primitive by probing it, along with its constraining plane if required.

Required: The project contains a nominal rectangle primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the features that will be probed.



2. Choose *Measure > Features > Define Measured*.
Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.



The Define Measured Feature Components dialog box opens.

3. In the **Method** list, select **Probe**.
4. In the **Submethod** list, select **Standard**.
5. Specify a constraining plane:
 - Select **Use Plane Feature** to use an existing plane in the Tree View, or to probe a new one that will appear in the Tree View.
 - Select **Probe Local Plane** to probe a new plane. Note that this plane is probed near the feature. It is stored within the feature and does not appear in the Tree View.
6. Click **Probe**.
7. Probe the constraining plane, if (**Probe New**) or **Probe Local Plane** is specified.
8. Probe the measured rectangle primitive.

Creation method

Method: Probe

Submethod: Standard

Parameters

Thin material

Constraining plane:

Method: Use Plane Feature

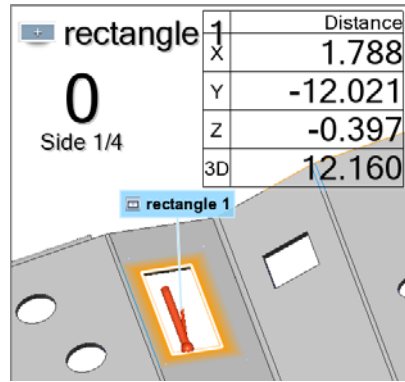
Name: (Probe New)

Fixed number of points: 3

Extract nominal primitive from CAD model

Tips

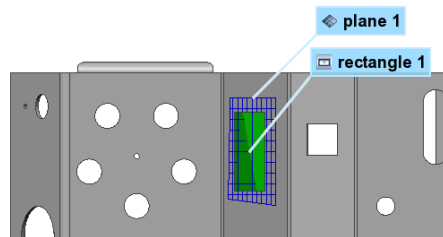
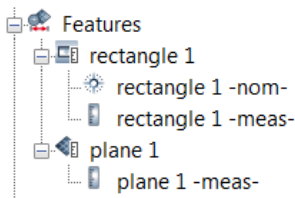
- If a feature contains a nominal primitive, it is highlighted and centered in the 3D scene for guidance.
- A minimum number of points must be probed for each side to fit a valid primitive.
- The probed primitive is compensated according to the specified compensation method set in the **More** section.
- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the rectangle.



Results

The probed measured primitive and the constraining plane are added to the Tree View, under the Features branch.

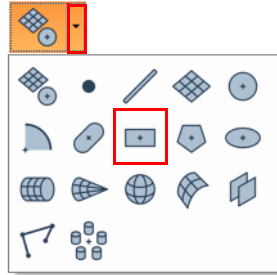
The probed feature components are displayed in the 3D scene.



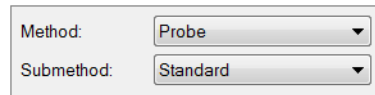
Notes

A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

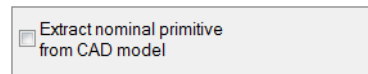
- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Rectangle**.



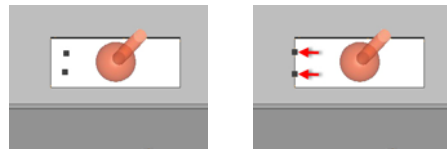
- In the **Method** list, select **Probe**.



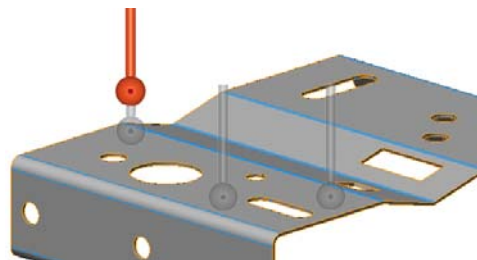
If the probing device is already aligned to a CAD model, the nominal primitive can be extracted at the same time by activating the **Extract nominal primitive from CAD model** item.



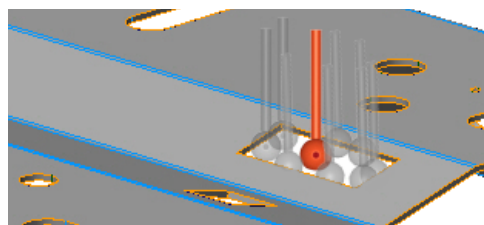
When probing, probed points are collected at the center of the probe tip. Compensation is necessary to offset the measured points to the point of contact.



When using **Compensation point** as the compensation method, a plane is compensated by lifting the probe slightly off the surface and ending the probing.



When using **Compensation point** as the compensation method, a rectangle is compensated by moving the probe inward (hole) or outward (rectangular shaped pin) from the feature boundary, then ending the probing.



Fit

Objective: Create measured rectangle primitives interactively using data elements.

Required: The project contains a Data object.

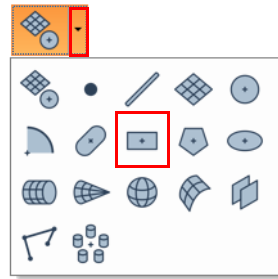
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Rectangle**.



3. In the **Method** list, select **Fit**.
4. In the **Submethod** list, select **Fit to Elements**.
5. Select the **Fit type**.

*It is recommended to set the **Fit type** to **Min** for a hole and **Max** for a rectangular shaped pin.*

6. Under **Constraints**, select **Plane**, then select a constraining plane from the list.

It is recommended to use a constraining plane to create the feature so that it is correctly oriented in space. The plane should be created prior to the fitting.

7. Under **Filtering**, adjust the filters based on the element selection.

*It is recommended to remove the **Max angle** filter if no data is selected on the walls of the hole.*

Creation method

Method: Fit

Submethod: Fit to Elements

Parameters

Fit type:

Best-fit

Min

Max

Constraints:

Plane: plane 1 -meas-

More ⌵

Filtering ⌵

Subsampling step: 1.000

Max angle: 45.000

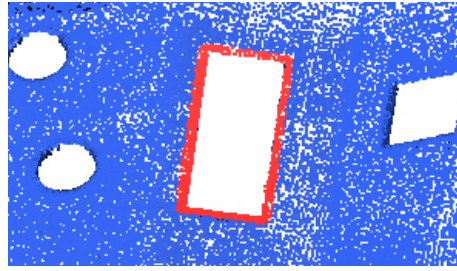
8. Choose *Select > Elements > Interactively*.



9. In the 3D scene, select the elements that will be used to fit the rectangle.

See [Selecting elements](#) on page 63 for information on how to select elements interactively.

10. Click **Create** to fit a primitive to the selected elements.

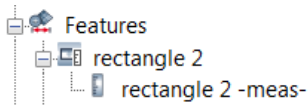


Tips

To use the **Fit** method to define the measured primitive of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The measured primitive is added to the rectangle feature, under the Features branch of the Tree View.



The primitive is displayed in the 3D scene.



Creating polygon features

Polygon features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal polygon primitives

Objective: Create a nominal polygon primitive interactively using Reference object elements or a mathematical definition.

Required: The project contains a Reference object, or the information that describes the nominal primitive is available.

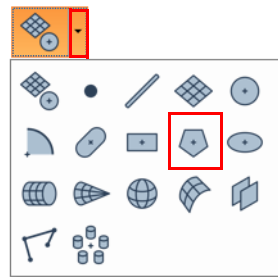
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

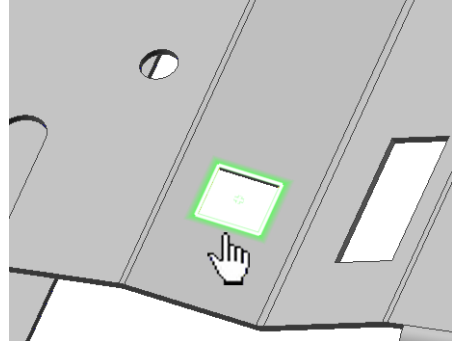


2. Expand the feature type drop-down menu and choose **Polygon**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. The interactive picking mode is automatically activated. If not, click **Pick**.
3. In the 3D scene, pick a polygon. Two approaches can be used:
 - Move the mouse cursor on the CAD model surface to select the closest polygon feature and pick the one desired when it is highlighted.
 - Move the mouse cursor in the void within the hole to select the first found polygon based on the current 3D scene view, then pick.



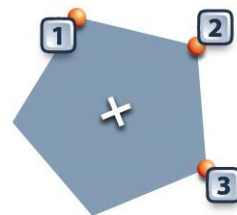
Numerically

1. In the **Method** list, select **Numerically**.
2. Specify the required values to define the feature **Center**, **Axis orientation**, **Vertex direction**, **Diameter**, and **Number of sides**.
3. Click **Create**.

Center		Axis orientation	
x:	<input type="text" value="1.000"/>	i:	<input type="text" value="0.000"/>
y:	<input type="text" value="1.000"/>	j:	<input type="text" value="0.000"/>
z:	<input type="text" value="1.000"/>	k:	<input type="text" value="1.000"/>
Vertex direction			
i:	<input type="text" value="1.000"/>		
j:	<input type="text" value="0.000"/>		
k:	<input type="text" value="0.000"/>		
Diameter:	<input type="text" value="1.000"/>		
Number of sides:	<input type="text" value="4"/>		

Anchor

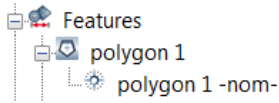
1. In the **Method** list, select **Anchor**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
2. Specify the **Number of sides** for the polygon.
3. In the 3D scene, anchor three points at the corners of the hole to create a polygon.



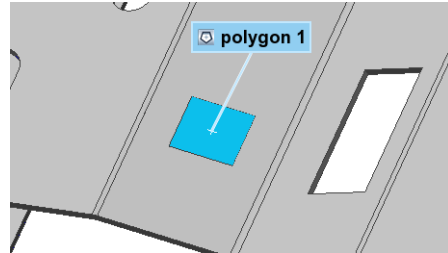
Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.

Results

The polygon feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured polygon primitives

To obtain the measured primitive of a polygon feature, the most common methods used are **Extract**, **Probe**, and **Fit**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Extract

Objective: Extract the measured polygon primitive using the nominal primitive.

Required: The project contains a polygon feature with a nominal primitive and an aligned Data object.

Steps

1. In the Tree View, select the features that will be extracted.

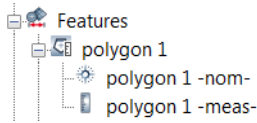


2. Choose *Measure > Extract Measured*.



Results

The measured primitive is added to the polygon feature in the Tree View, under the Features branch.



The primitives are displayed in the 3D scene.

polygon 1				
Edge Len	Nom	Meas	Dev Test	
X	12.000	11.036	-0.964	○
Y	68.604	69.236	0.632	○
Z	101.600	101.749	0.149	○
	3.157	3.242	0.085	○

Probe

Objective: Create the measured polygon primitive by probing it, along with its constraining plane if required.

Required: The project contains a nominal polygon primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the feature that will be probed.



2. Choose *Measure > Features > Define Measured*.

Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.



The Define Measured Feature Components dialog box opens.

3. In the **Method** list, select **Probe**.
4. In the **Submethod** list, select **Standard**.
5. Specify a constraining plane:
 - Select **Use Plane Feature** to use an existing plane in the Tree View, or to probe a new one that will appear in the Tree View.
 - Select **Probe Local Plane** to probe a new plane. Note that this plane is probed near the feature. It is stored within the feature and does not appear in the Tree View.
6. Specify the **Number of sides** for the polygon.
7. Click **Probe**.
8. Probe the constraining plane, if **(Probe New)** or **Probe Local Plane** is specified.
9. Probe the required number of sides for the measured polygon primitive.

Creation method

Method: Probe

Parameters

Thin material

Constraining plane:

Method: Use Plane Feature

Name: (Probe New)

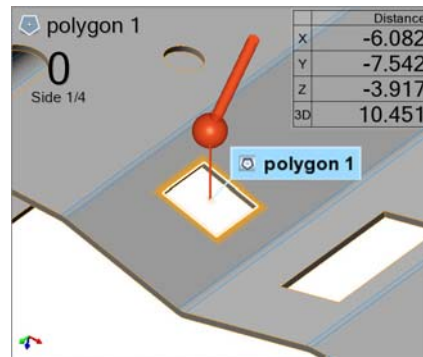
Fixed number of points: 3

Number of sides: 4

Extract nominal primitive from CAD model

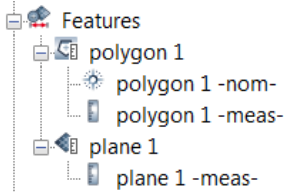
Tips

- If a feature contains a nominal primitive, it is highlighted and centered in the 3D scene for guidance.
 - A minimum number of points must be probed for each side to fit a valid primitive.
 - The probed primitive is compensated according to the specified compensation method set in the **More** section.
-
- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the polygon.

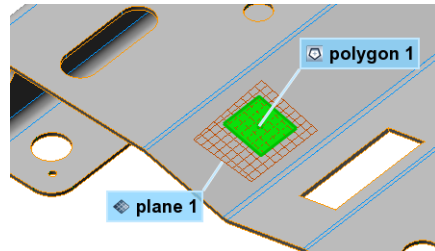


Results

The probed measured primitive and constraining plane are added to the Tree View, under the Features branch.



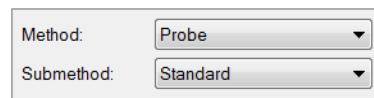
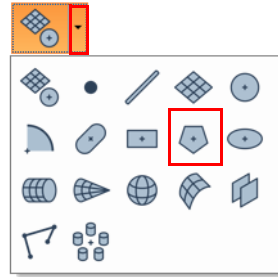
The primitives are displayed in the 3D scene.



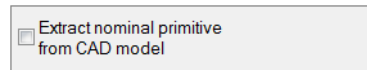
Notes

A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Polygon**.
- In the **Method** list, select **Probe**.



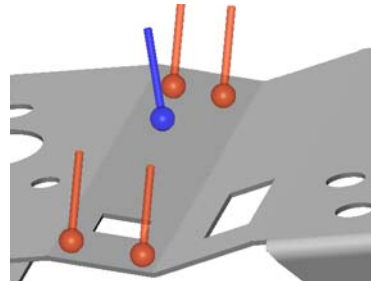
If the probing device is already aligned to a CAD model, the nominal primitive can be extracted at the same time by activating the **Extract nominal primitive from CAD model** option.



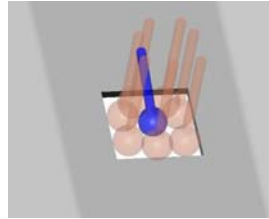
When probing, probed points are collected at the center of the probe tip. Compensation is necessary to offset the measured points to the point of contact.



When using **Compensation point** as the compensation method, a plane is compensated by lifting the probe slightly off the surface and ending the probing.



When using **Compensation point** as the compensation method, a polygon is compensated by moving the probe inward (hole) or outward (polygonal shaped pin) from the feature boundary, then ending the probing.



Fit

Objective: Create measured polygon primitives interactively using data elements.

Required: The project contains a Data object.

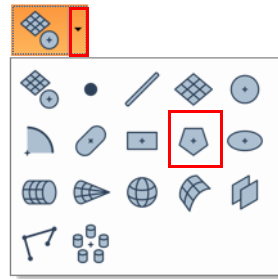
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Polygon**.



3. In the **Method** list, select **Fit**.
4. In the **Submethod** list, select **Fit to Elements**.
5. Select the **Fit type**.

*It is recommended to set the **Fit type** to **Min** for a hole and **Max** for a polygonal shaped pin.*

6. Under **Constraints**, select **Plane**, then select a constraining plane, if required.

It is recommended to use a constraining plane to create the feature so that it is correctly oriented in space. The plane should be created prior to the fitting.

7. Under **Filtering**, adjust the filters based on the element selection.

*It is recommended to remove the **Max angle** filter if no data is selected on the walls of the hole.*

Creation method

Method: Fit

Submethod: Fit to Elements

Parameters

Number of sides: 4

Fit type:

Best-fit

Min

Max

Constraints:

Plane: plane 1 - meas-

More ▾

Filtering ⌵

Subsampling step: 1.000

Max angle: 45.000

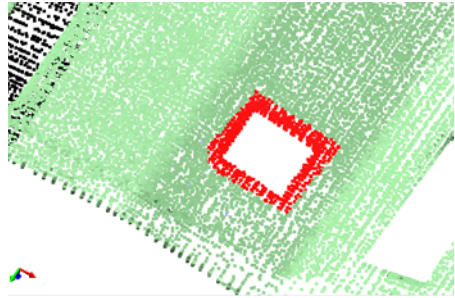
8. Choose *Select > Elements > Interactively*.



- In the 3D scene, select the elements that will be used to fit the polygon.

See [Selecting elements on page 63](#) for information on how to select elements interactively.

- Click **Create** to fit a primitive to the selected elements.

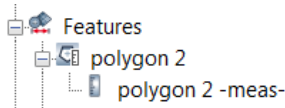


Tips

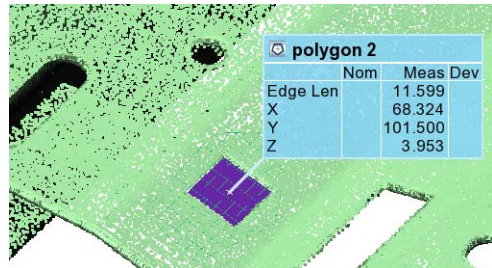
To use the **Fit** method to define the measured primitive of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The measured primitive is added to the polygon feature, under the Features branch of the Tree View.



The primitive is displayed in the 3D scene.



Creating ellipse features

Ellipse features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal ellipse primitives

Objective: Create a nominal ellipse primitive interactively using Reference object elements or a mathematical definition.

Required: The project contains a Reference object, or the information that describes the nominal primitive is available.

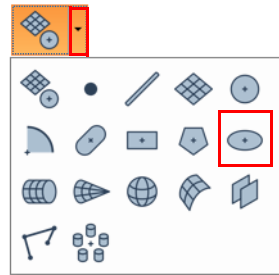
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

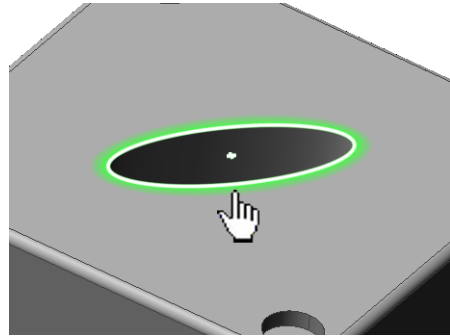


2. Expand the feature type drop-down menu and choose **Ellipse**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. The interactive picking mode is automatically activated. If not, click **Pick**.
3. In the 3D scene, pick an ellipse. Two approaches can be used:
 - Move the mouse cursor on the CAD model surface to select the closest ellipse feature and pick the one desired when it is highlighted.
 - Move the mouse cursor in the void within the hole to select the first found ellipse based on the current 3D scene view, then pick.



Numerically

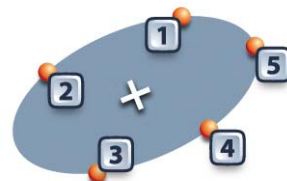
1. In the **Method** list, select **Numerically**.
2. Specify the required values to define the feature **Center**, **Axis orientation**, **Major orientation**, **Major diameter**, and **Minor diameter**.
3. Click **Create**.

Center		Axis orientation	
x:	<input type="text" value="0.000"/>	i:	<input type="text" value="0.000"/>
y:	<input type="text" value="0.000"/>	j:	<input type="text" value="0.000"/>
z:	<input type="text" value="0.000"/>	k:	<input type="text" value="1.000"/>
Major orientation			
i:	<input type="text" value="1.000"/>		
j:	<input type="text" value="0.000"/>		
k:	<input type="text" value="0.000"/>		
Major diameter:	<input type="text" value="1.000"/>		
Minor diameter:	<input type="text" value="0.500"/>		

Anchor

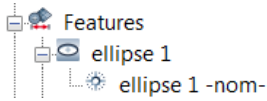
1. In the **Method** list, select **Anchor**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
2. In the 3D scene, anchor five points on a hole boundary to create an ellipse.

Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.

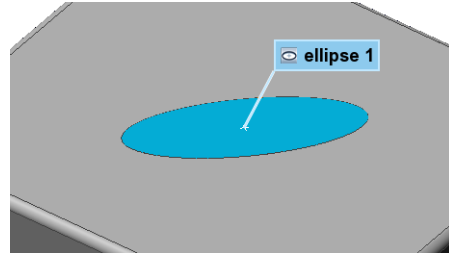


Results

The ellipse feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured ellipse primitives

To obtain the measured primitive of an ellipse feature, the most common methods used are **Extract**, **Probe**, and **Fit**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

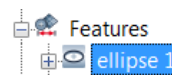
Extract

Objective: Extract the measured ellipse primitive using the nominal primitive.

Required: An ellipse feature with a nominal primitive and an aligned Data object.

Steps

1. In the Tree View, select the feature that will be extracted.

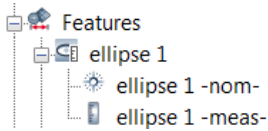


2. Choose *Measure > Extract Measured*.



Results

The measured primitive is added to the ellipse feature in the Tree View, under the Features branch.



The primitives are displayed in the 3D scene.

ellipse 1				
	Nom	Meas	Dev	Test
Major Dia	50.000	49.998	-0.002	○
Minor Dia	18.000	18.000	0.000	○
X	50.000	50.010	0.010	○
Y	30.000	30.000	0.000	○
Z	-50.000	-50.000	0.000	○

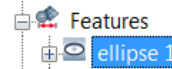
Probe

Objective: Create the measured ellipse primitive by probing it, along with its constraining plane, if required.

Required: The project contains a nominal ellipse primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the feature that will be probed.



2. Choose *Measure > Features > Define Measured*.

Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.



The Define Measured Feature Components dialog box opens.

3. In the **Method** list, select **Probe**.
4. In the **Submethod** list, select **Standard**.
5. Specify a constraining plane:
 - Select **Use Plane Feature** to use an existing plane in the Tree View, or to probe a new one that will appear in the Tree View.
 - Select **Probe Local Plane** to probe a new plane. Note that this plane is probed near the feature. It is stored within the feature and does not appear in the Tree View.
6. Click **Probe**.
7. Probe the constraining plane, if **(Probe New)** or **Probe Local Plane** is specified.
8. Probe the measured ellipse primitive.

Creation method

Method: Probe

Submethod: Standard

Parameters

Thin material

Constraining plane:

Method: Use Plane Feature

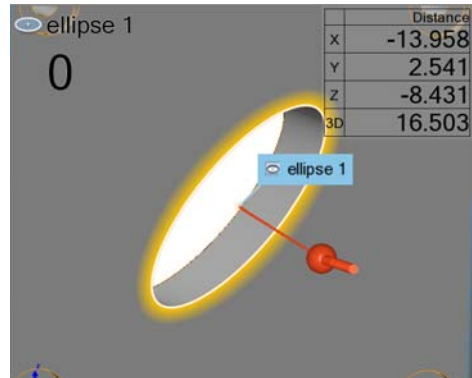
Name: (Probe New)

Fixed number of points: 3

Extract nominal primitive from CAD model

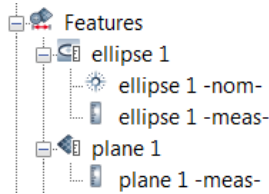
Tips

- If a feature contains a nominal primitive, it is highlighted and centered in the 3D scene for guidance.
 - A minimum number of points must be probed to fit a valid primitive.
 - The probed primitive is compensated according to the specified compensation method set in the **More** section.
- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the ellipse.

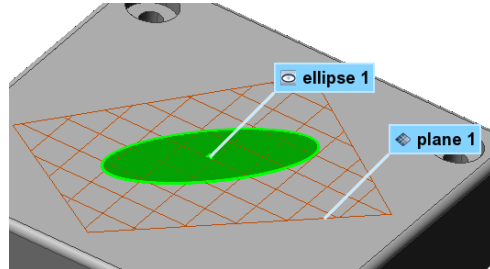


Results

The probed measured primitive is added to the ellipse feature, under the Features branch in the Tree View.



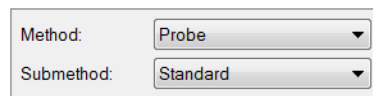
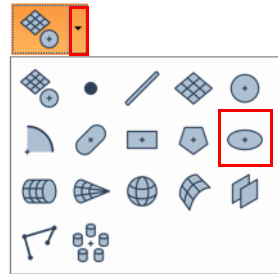
The primitives are displayed in the 3D scene.



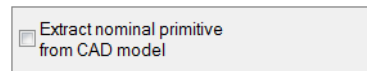
Notes

A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

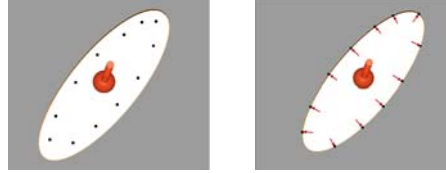
- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Ellipse**.
- In the **Method** list, select **Probe**.



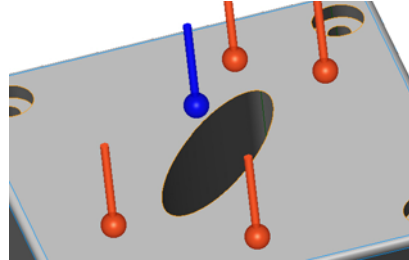
If the probing device is already aligned to a CAD model, the nominal primitive can be extracted at the same time by activating the **Extract nominal primitive from CAD model** item.



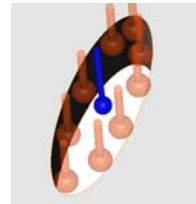
When probing, probed points are collected at the center of the probe tip. Compensation is necessary to offset the measured points to the point of contact.



When using **Compensation point** as the compensation method, a plane is compensated by lifting the probe slightly off the surface and ending the probing.



When using **Compensation point** as the compensation method, an ellipse is compensated by moving the probe inward (hole) or outward (pin) from the feature boundary, then ending the probing.



Fit

Objective: Create measured ellipse primitives interactively using data elements.

Required: The project contains a Data object.

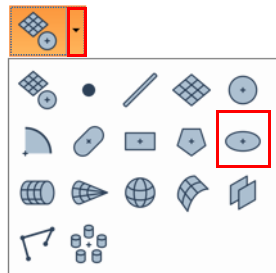
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Ellipse**.



3. In the **Method** list, select **Fit**.
4. In the **Submethod** list, select **Fit to Elements**.
5. Select the **Fit type**.

*It is recommended to set the **Fit type** to **Min** for a hole and **Max** for a pin.*

6. Under **Constraints**, select **Plane**, then select a constraining plane, if required.

It is recommended to use a constraining plane to create the feature so that it is correctly oriented in space. The plane should be created prior to the fitting.

7. Under **Filtering**, adjust the filters based on the element selection.

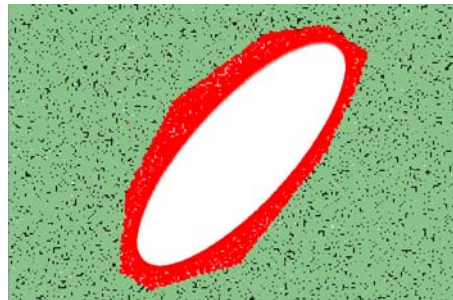
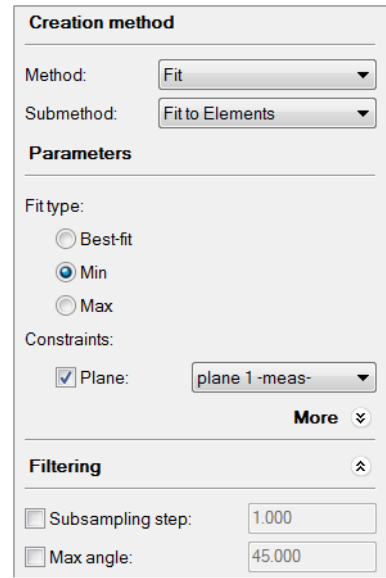
*It is recommended to remove the **Max angle** filter if no data is selected on the walls of the hole.*

8. Choose *Select > Elements > Interactively*.

9. In the 3D scene, select the elements that will be used to fit the ellipse.

See [Selecting elements](#) on page 63 for information on how to select elements interactively.

10. Click **Create** to fit a primitive to the selected elements.

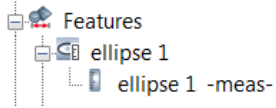


Tips

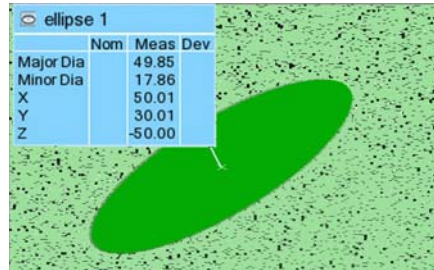
To use the **Fit** method to define the measured primitive of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The measured primitive is added to the ellipse feature, under the Features branch of the Tree View.



The primitive is displayed in the 3D scene.



Creating cylinder features

Cylinder features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal cylinder primitives

Objective: Create a nominal cylinder primitive interactively using Reference object elements or a mathematical definition.

Required: A Reference object or the information that defines the nominal primitive is available.

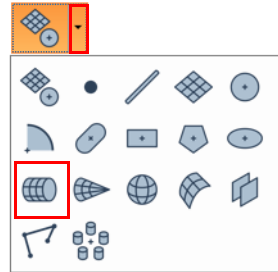
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

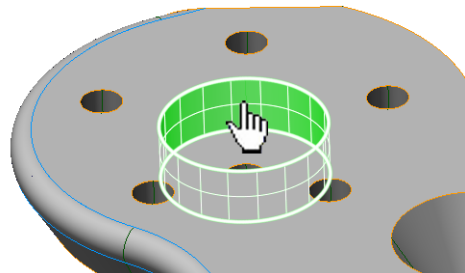


2. Expand the feature type drop-down menu and choose **Cylinder**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. The interactive picking mode is automatically activated. If not, click **Pick**.
3. In the 3D scene, move the mouse cursor on the CAD model surface to select the closest cylinder feature and pick the one desired when it is highlighted.
4. Right-click to exit the picking mode.



Numerically

1. In the **Method** list, select **Numerically**.
2. Specify the required parameter values to define the feature **Origin, Orientation, Radius, and Height**.
3. Click **Create**.

Origin		Orientation	
x:	1.000	i:	0.000
y:	1.000	j:	0.000
z:	1.000	k:	1.000
Radius:	1.000		
Height:	1.000		

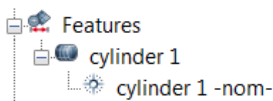
Anchor

1. In the **Method** list, select **Anchor**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
2. In the 3D scene, anchor four points on the surface to create a cylinder.
Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.

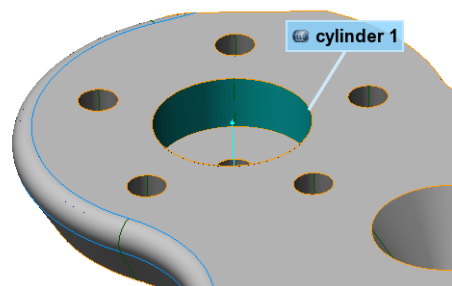


Results

The cylinder feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured cylinder primitives

To obtain the measured primitive of a cylinder feature, the most common methods used are **Extract**, **Probe**, **Extract from Polygonal Models** and **Fit**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

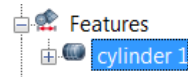
Extract

Objective: Extract the measured cylinder primitive using the nominal primitive.

Required: The project contains a feature with a nominal primitive and an aligned Data object.

Steps

1. In the Tree View, select the feature that will be extracted.

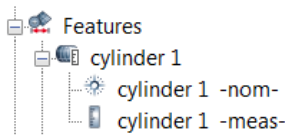


2. Choose *Measure > Extract Measured*.

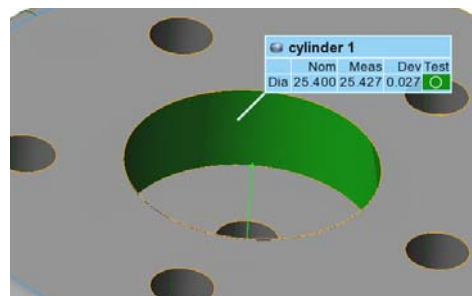


Results

The measured primitive is added to the cylinder feature in the Tree View, under the Features branch.



The primitives are displayed in the 3D scene.



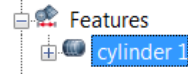
Probe

Objective: Create the measured cylinder primitive by probing it.

Required: The project contains a nominal cylinder primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the feature that will be probed.



2. Choose *Measure > Features > Define Measured*.
Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.



The Define Measured Feature Components dialog box opens.

3. In the **Method** list, select **Probe**.
4. In the **Submethod** list, select **Standard**.
5. Click **Probe**.
6. Probe the measured cylinder primitive.

Creation method

Method: Probe

Submethod: Standard

Parameters

Constraining plane:

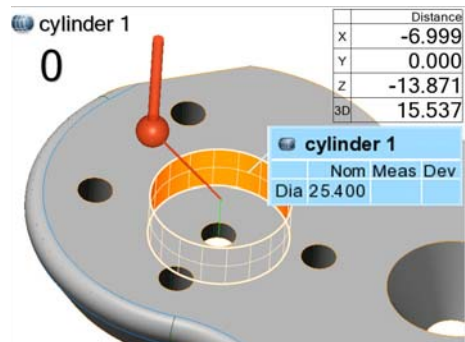
Method: Use Plane Feature

Name: (Probe New)

Fixed number of points: 3

Tips

- If a feature contains a nominal primitive, it will be highlighted and centered in the 3D scene for guidance.
- A minimum number of points must be probed to fit a valid primitive.
- The probed primitive is compensated according to the specified compensation method set in the **More** section.
- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the cylinder.

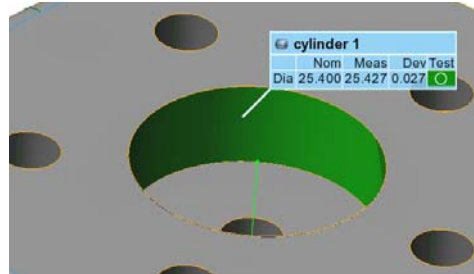


Results

The probed measured primitive is added to the cylinder feature, under the Features branch of the Tree View.



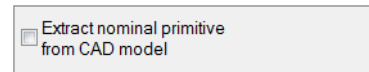
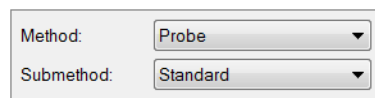
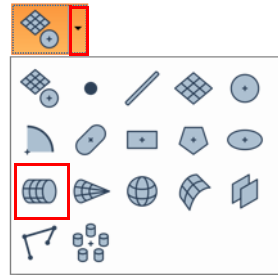
The primitives are displayed in the 3D scene.



Notes

A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Cylinder**.
- In the **Method** list, select **Probe**.

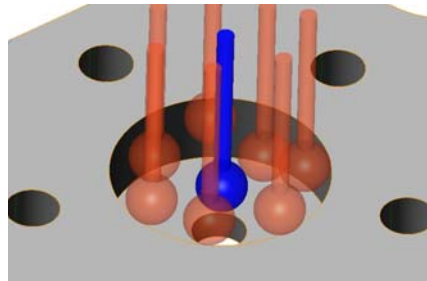


If the probing device is already aligned to a CAD model, the nominal primitive can be extracted at the same time by activating the **Extract nominal primitive from CAD model** option.

When probing, probed points are collected at the center of the probe tip. Compensation is necessary to offset the measured points to the point of contact.



When using **Compensation point** as the compensation method, a cylinder is compensated by moving the probe inward (hole) or outward (pin) from the feature boundary, then ending the probing.



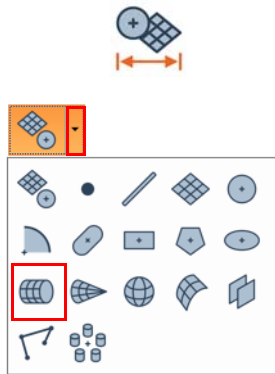
Fit

Objective: Create measured cylinder primitives interactively using data elements.

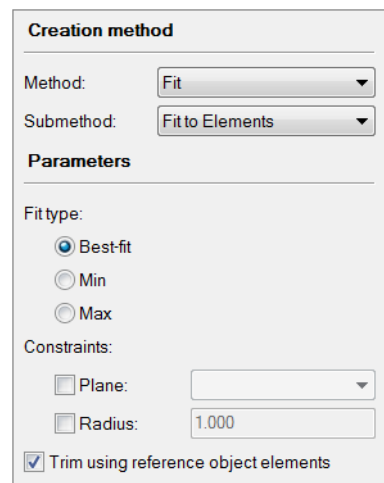
Required: The project contains a Data object.

Steps

1. Choose *Measure > Features > Create*.
The Create Features dialog box opens.
2. Expand the feature type drop-down menu and choose **Cylinder**.



3. In the **Method** list, select **Fit**.
4. In the **Submethod** list, select **Fit to Elements**.
5. Select the **Fit type**.



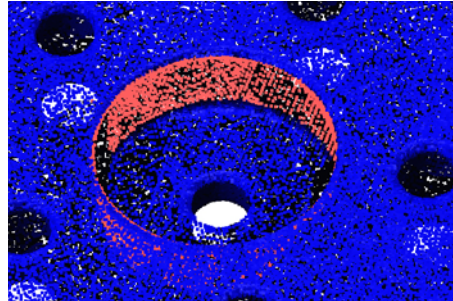
6. Choose *Select > Elements > Interactively*.



7. In the 3D scene, select the elements that will be used to fit the cylinder.

See *Selecting elements* on page 63 for information on how to select elements interactively.

8. Click **Create** to fit a primitive to the selected elements.

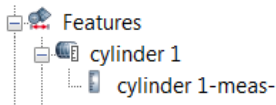


Tips

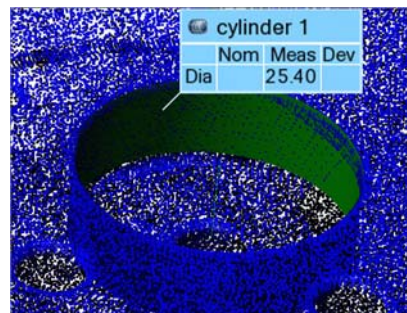
To use the **Fit** method to define the measured primitive of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The measured primitive is added to the cylinder feature, under the Features branch of the Tree View.



The primitive is displayed in the 3D scene.



Extract from polygonal models

Objective: Create measured cylinder feature components interactively using a polygonal model.

Required: A polygonal model is available in the project.

Steps

1. Choose *Measure > Features > Extract from Polygonal Models*.

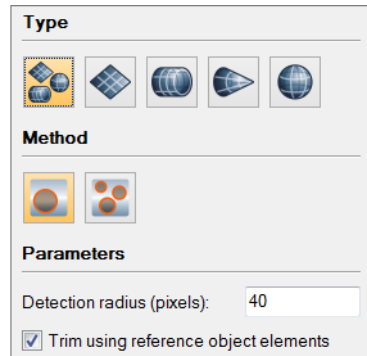


2. Specify the feature **Type**.

The **Automatic** type is specified by default, which automatically determines the feature type that adjusts best to the area selected.

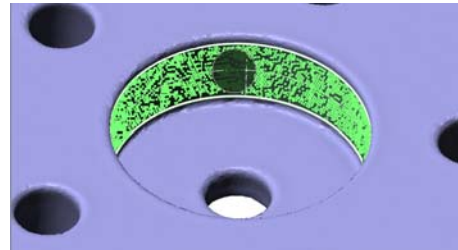
3. Specify the **Method**.

The **Single Detection Zone** method is selected by default. The **Multiple Detection Zones** method is useful when all areas that can be used to extract the feature primitive are not contiguous.



4. In the 3D scene, using the circular detection zone around the mouse pointer, click an area on the polygonal model that is part of the desired feature.

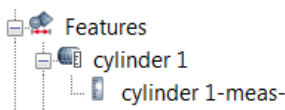
*It is possible to increase or decrease the size of the detection radius at any time in the **Parameters** subsection, or by using the mouse wheel.*



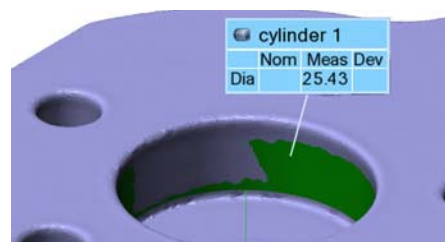
5. Click **Confirm**.

Results

The measured cylinder feature primitive is added to the Tree View, under the Features branch.



The measured feature primitive is displayed in the 3D scene.



Notes

The **Extract from Polygonal Models** method can also be used to create nominal primitives when using a polygonal model as the Reference object.

Creating cone features

Cone features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal cone primitives

Objective: Create a nominal cone primitive interactively using Reference object elements or a mathematical definition.

Required: The project contains a Reference object, or the information that defines the nominal primitive is available.

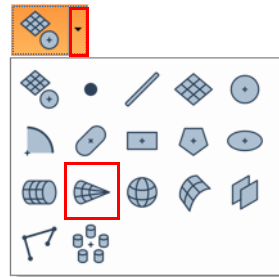
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

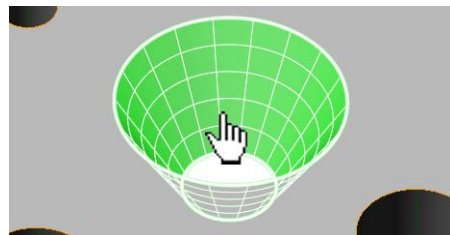


2. Expand the feature type drop-down menu and choose **Cone**.
3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. The interactive picking mode is automatically activated. If not, click **Pick**.
3. In the 3D scene, move the mouse cursor on the CAD model surface to select the closest cone feature and pick the one desired when it is highlighted.
4. Right-click to exit the picking mode.



Numerically

1. In the **Method** list, select **Numerically**.
2. Specify the required values to define the feature **Apex, Orientation, Slope (degrees), Distance from apex,** and **Height**.
3. Click **Create**.

Apex		Orientation	
x:	1.000	i:	0.000
y:	1.000	j:	0.000
z:	1.000	k:	1.000
Slope (degrees):		45.000	
Distance from apex:		0.000	
Height:		1.000	

Anchor

1. In the **Method** list, select **Anchor**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
2. In the 3D scene, anchor four points on the surface to create a cone.
Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.
3. Right-click to exit the anchoring mode.

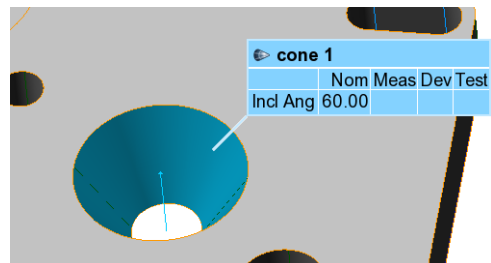


Results

The cone feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured cone primitives

To obtain the measured primitive of a cone feature, the most common methods used are **Extract**, **Probe**, **Extract from Polygonal Models**, and **Fit**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Extract

Objective: Extract the measured cone primitive using the nominal primitive.

Required: The project contains a feature with a nominal primitive and an aligned Data object.

Steps

1. In the Tree View, select the features that will be extracted.

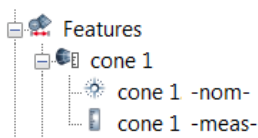


2. Choose *Measure > Extract Measured*.

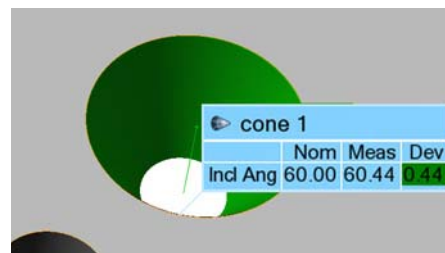


Results

The measured primitive is added to the cone feature in the Tree View, under the Features branch.



The primitives are displayed in the 3D scene.



Probe

Objective: Create the measured cone primitive by probing it.

Required: The project contains a nominal cone feature. A probing device is ready to use.

Steps

1. In the Tree View, select the feature that will be probed.



2. Choose *Measure > Features > Define Measured*.

Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.



The Define Measured Feature Components dialog box opens.

3. In the **Method** list, select **Probe**.

4. In the **Submethod** list, select **Standard**.

5. Click **Probe**.

6. Probe the measured cone primitive.

Creation method

Method: Probe

Submethod: Standard

Parameters

Constraining plane:

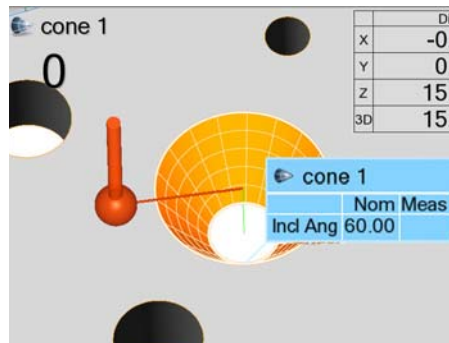
Method: Use Plane Feature

Name: (Probe New)

Fixed number of points: 3

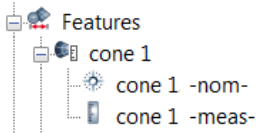
Tips

- If a feature contains a nominal primitive, it is highlighted and centered in the 3D scene for guidance.
- A minimum number of points must be probed to fit a valid primitive.
- The probed primitive is compensated according to the specified compensation method set in the **More** section.
- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the cone.

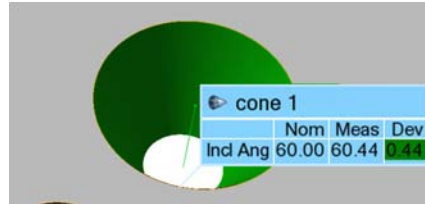


Results

The probed measured primitive is added to the cone feature, under the Features branch of the Tree View.



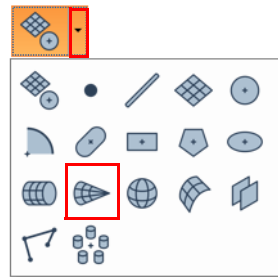
The primitives are displayed in the 3D scene.



Notes

A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Cone**.
- In the **Method** list, select **Probe**.



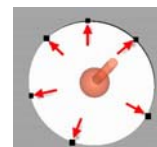
Method: Probe

Submethod: Standard

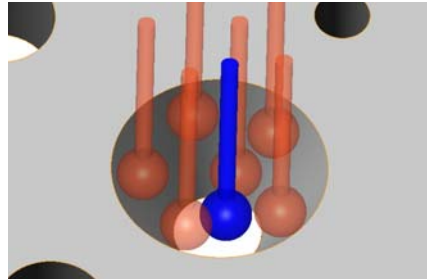
Extract nominal primitive from CAD model

If the probing device is already aligned to a CAD model, the nominal primitive can be extracted at the same time by activating the **Extract nominal primitive from CAD model** item.

When probing, probed points are collected at the center of the probe tip. Compensation is necessary to offset the measured points to the point of contact.



When using **Compensation point** as the compensation method, a cone is compensated by moving the probe inward (hole) or outward (pin) from the feature boundary, then ending the probing.



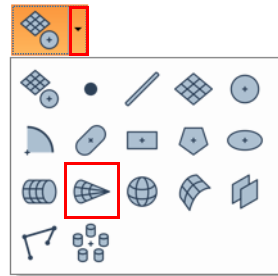
Fit

Objective: Create measured cone primitives interactively using data elements.

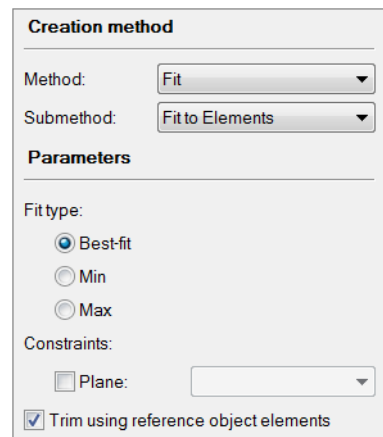
Required: The project contains a Data object.

Steps

1. Choose *Measure > Features > Create*.
The Create Features dialog box opens.
2. Expand the feature type drop-down menu and choose **Cone**.



3. In the **Method** list, select **Fit**.
4. In the **Submethod** list, select **Fit to Elements**.
5. Select the **Fit type**.



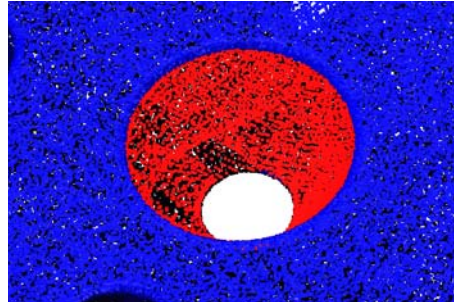
6. Choose *Select > Elements > Interactively*.



- In the 3D scene, select the elements that will be used to fit the cone.

See [Selecting elements](#) on page 63 for information on how to select elements interactively.

- Click **Create** to fit a primitive to the selected elements.



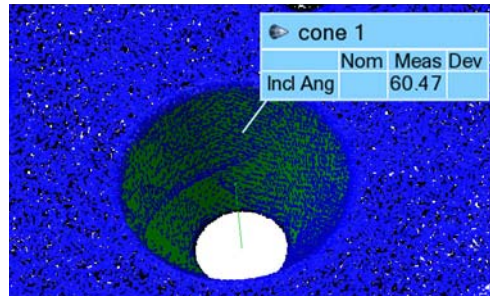
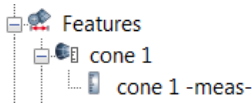
Tips

To use the **Fit** method to define the measured primitive of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The measured primitive is added to the cone feature, under the Features branch of the Tree View.

The primitive is displayed in the 3D scene.



Extract from polygonal model

Objective: Create a measured cone primitive interactively using a polygonal model.

Required: The project contains a polygonal model.

Steps

- Choose *Measure > Features > Extract from Polygonal Models*.



- Specify the feature **Type**.

The **Automatic** method is specified by default, which automatically determines the feature type that adjusts best to the area selected.

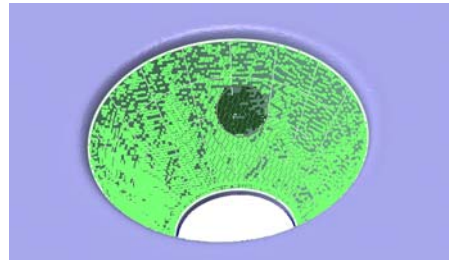
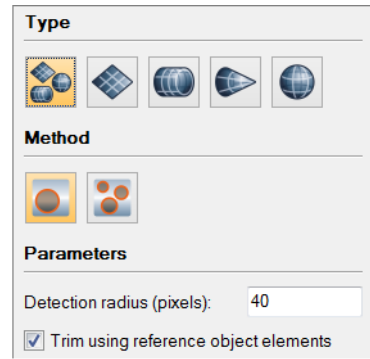
- Specify the **Method**.

The **Single Detection Zone** method is selected by default. The **Multiple Detection Zones** method is useful when all areas that can be used to extract the feature primitive are not contiguous.

- In the 3D scene, using the circular detection zone around the mouse pointer, click an area on the polygonal model that is part of the desired feature.

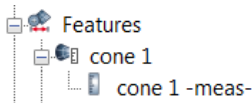
*It is possible to increase or decrease the size of the detection radius at any time in the **Parameters** subsection, or by using the mouse wheel.*

- Click **Confirm**.

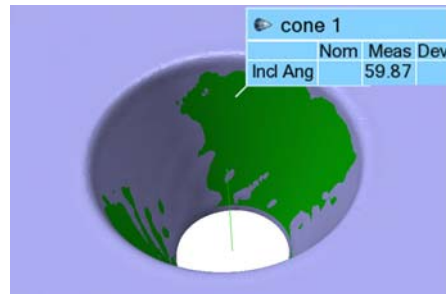


Results

The measured primitive is added to the cone feature, under the Features branch.



The measured primitive is displayed in the 3D scene.



Notes

The **Extract from Polygonal Models** method can also be used to create nominal primitives when using a polygonal model as the Reference object.

Creating sphere features

Sphere features are typically created with a nominal primitive using different creation methods. The measured primitive is subsequently added to the feature using specific methods.

The creation procedures are explained in the following subsections. For information on the creation methods and their key parameters, see [The feature creation methodology](#) on page 207.

Creating nominal sphere primitives

Objective: Create a nominal sphere primitive interactively using Reference object elements or a mathematical definition.

Required: The project contains a Reference object, or the information that defines the nominal primitive is available.

Steps

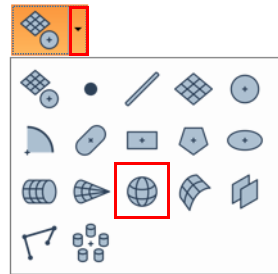
1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



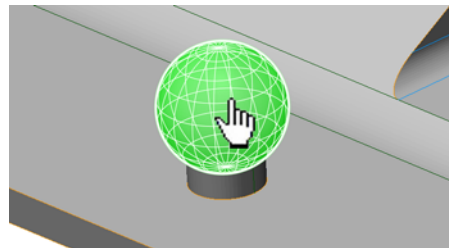
2. Expand the feature type drop-down menu and choose **Sphere**.

3. Choose one of the following methods.



Pick on CAD Model

1. In the **Method** list, select **Pick on CAD Model**.
2. The interactive picking mode is automatically activated. If not, click **Pick**.
3. In the 3D scene, move the mouse cursor on the CAD model surface to select the closest sphere feature and pick the one desired when it is highlighted.
4. Right-click to exit the picking mode.



Numerically

1. In the **Method** list, select **Numerically**.
2. Specify the required values to define the feature **Center** and **Radius**.
3. Click **Create**.

Center	
x:	0.000
y:	0.000
z:	0.000
Radius:	1.000

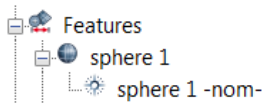
Anchor

1. In the **Method** list, select **Anchor**.
The interactive anchoring mode is automatically activated. If not, click **Anchor**.
2. In the 3D scene, anchor four points on the surface to create a sphere.
Press and hold SHIFT when anchoring to pick the closest vertex on a polygonal model.
3. Right-click to exit the anchoring mode.

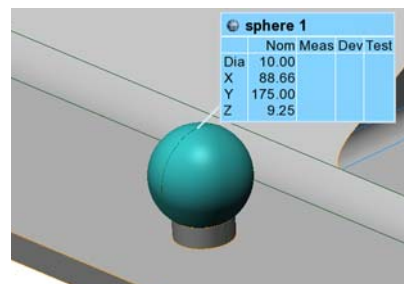


Results

The sphere feature and its nominal primitive are added to the Tree View, under the Features branch.



The nominal primitive is displayed in the 3D scene.



Obtaining measured sphere primitives

To obtain the measured primitive of a sphere feature, the most common methods used are **Extract**, **Probe**, **Extract from Polygonal Models**, and **Fit**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

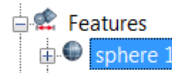
Extract

Objective: Extract the measured sphere primitive using the nominal primitive.

Required: The project contains a feature with a nominal primitive and an aligned Data object.

Steps

1. In the Tree View, select the features that will be extracted.

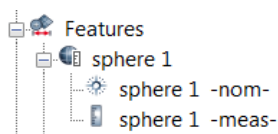


2. Choose *Measure > Extract Measured*.

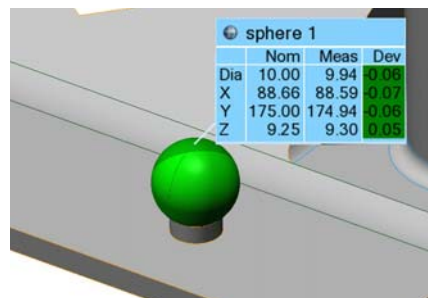


Results

The measured primitive is added to the sphere feature in the Tree View, under the Features branch.



The primitives are displayed in the 3D scene.



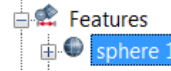
Probe

Objective: Create the measured sphere primitive by probing it.

Required: The project contains a nominal sphere primitive. A probing device is ready to use.

Steps

1. In the Tree View, select the features that will be probed.

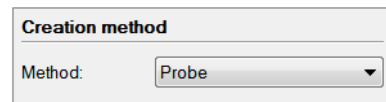


2. Choose *Measure > Features > Define Measured*.
Alternatively, right-click the selected features in the Tree View and choose *Probe Measured*.



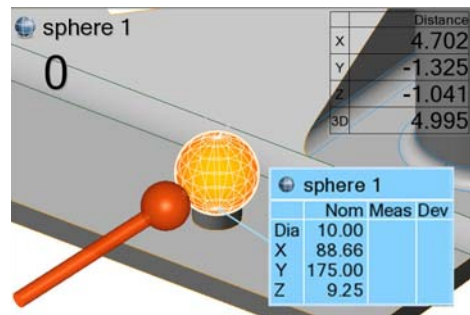
The Define Measured Feature Components dialog box opens.

3. In the **Method** list, select **Probe**.
4. Click **Probe**.
5. Probe the measured sphere primitive.



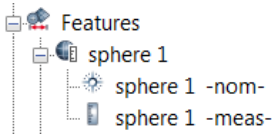
Tips

- If a feature contains a nominal primitive, it is highlighted and centered in the 3D scene for guidance.
- A minimum number of points must be probed to fit a valid primitive.
- The probed primitive is compensated according to the specified compensation method set in the **More** section.
- Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the sphere.

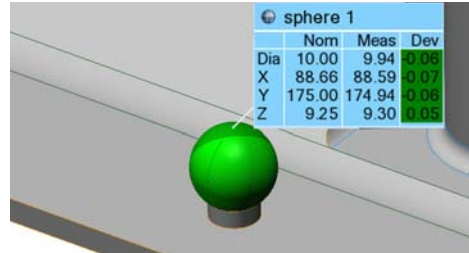


Results

The probed measured primitive is added to the sphere feature, under the Features branch of the Tree View.



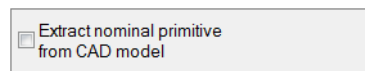
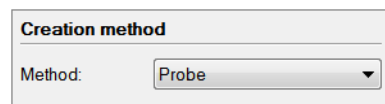
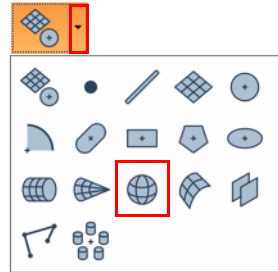
The primitives are displayed in the 3D scene.



Notes

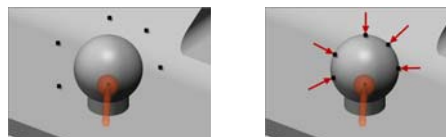
A nominal primitive offers guidance when probing the measured primitive, but is not necessary. To probe a measured primitive without a nominal primitive:

- Choose *Measure > Features > Create*.
- Expand the feature type drop-down menu and choose **Sphere**.
- In the **Method** list, select the **Probe** creation method.

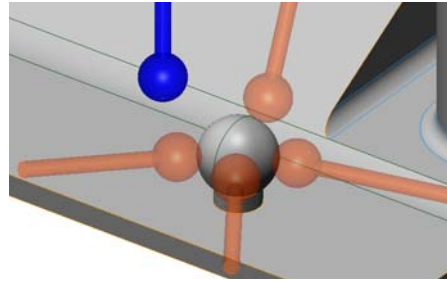


If the probing device is already aligned to a CAD model, the nominal primitive can be extracted at the same time by activating the **Extract nominal primitive from CAD model** item.

When probing, probed points are collected at the center of the probe tip. Compensation is necessary to offset the measured points to the point of contact.



When using **Compensation point** as the compensation method, a sphere is compensated by moving the probe inward (concave surface) or outward (convex surface) from the feature boundary, then ending the probing.



Fit

Objective: Create measured sphere primitives interactively using data elements.

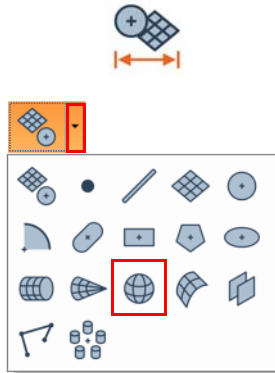
Required: The project contains a Data object.

Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

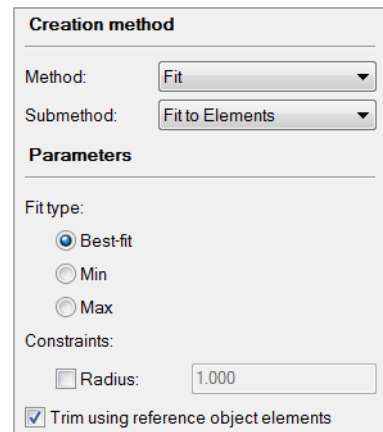
2. Expand the feature type drop-down menu and choose **Sphere**.



3. In the **Method** list, select **Fit**.

4. In the **Submethod** list, select **Fit to Elements**.

5. Select the **Fit type**.



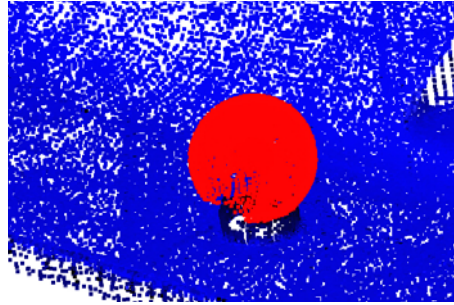
6. Choose *Select > Elements > Interactively*.



- In the 3D scene, select the elements that will be used to fit the sphere.

See [Selecting elements](#) on page 63 for information on how to select elements interactively.

- Click **Create** to fit a primitive to the selected elements.

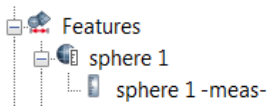


Tips

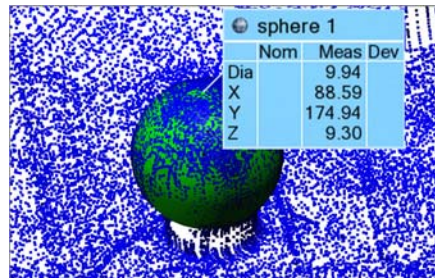
To use the **Fit** method to define the measured primitive of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The measured primitive is added to the sphere feature, under the Features branch of the Tree View.



The primitive is displayed in the 3D scene.



Extract from Polygonal Models

Objective: Create measured sphere primitives interactively using a polygonal model.

Required: The project contains a polygonal model.

Steps

1. Choose *Measure > Features > Extract from Polygonal Models*.

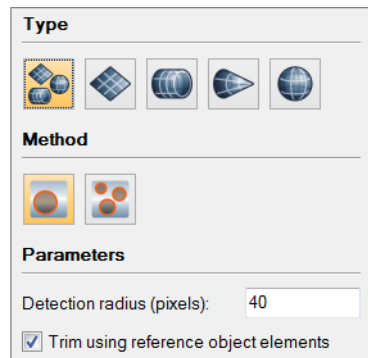


2. Specify the feature **Type**.

The **Automatic** method is specified by default, which automatically determines the feature type that adjusts best to the area selected.

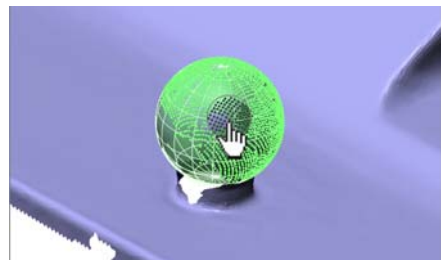
3. Specify the **Method**.

The **Single Detection Zone** method is selected by default. The **Multiple Detection Zones** method is useful when all areas that can be used to extract the feature primitive are not contiguous.



4. In the 3D scene, using the circular detection zone around the mouse pointer, click an area on the polygonal model that is part of the desired feature.

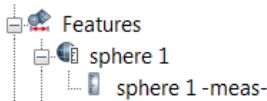
*It is possible to increase or decrease the size of the detection zone at any time in the **Parameters** subsection, or by using the mouse wheel.*



5. Click **Confirm**.

Results

The measured sphere feature primitive is added to the Tree View, under the Features branch.



The measured feature primitive is displayed in the 3D scene.



Notes

The **Extract from Polygonal Models** method can also be used to create nominal primitives when using a polygonal model as the Reference object.

Creating surface features

Surface features are created with a nominal component using different creation methods. The measured component is subsequently added to the feature using specific methods described in the following subsections.

Creating the nominal component of surface features

Objective: Create the nominal component of a surface feature interactively using Reference object elements.

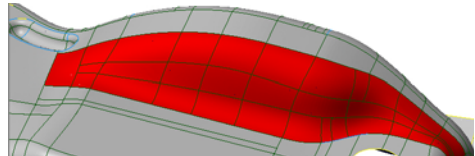
Required: The project contains a Reference object in the form of a CAD model.

Steps

1. Choose *Select > Elements > Interactively*.



2. In the 3D scene, select the CAD surfaces used to create the nominal component of the surface feature.

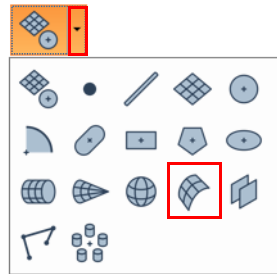


3. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



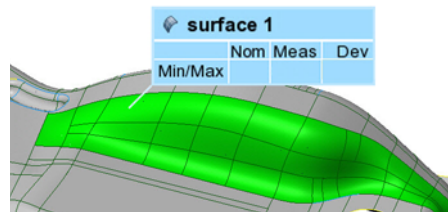
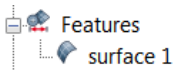
4. Expand the feature type drop-down menu and choose **Surface**.
5. In the **Method** list, choose **From Reference Elements**.



6. Click **Create**.

Results

The surface feature is added to the Tree View, under the Features branch. The feature is displayed in the 3D scene.



Obtaining the measured component of a surface feature

To obtain the measured component of a surface feature, the most common methods used are **Extract**, **Probe**, and **From Data Points**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

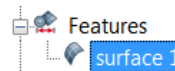
Extract

Objective: Extract the measured surface component by detecting it near the nominal component.

Required: The project contains a feature with a nominal component and an aligned Data object.

Steps

1. In the Tree View, select the features to extract.

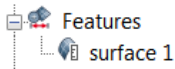


2. Choose *Measure > Extract Measured*.

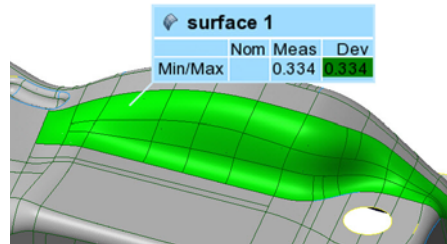


Results

The measured component is extracted and a measured status icon is added to the surface feature icon in the Tree View, under the Features branch.



The feature is displayed in the 3D scene.



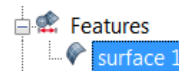
Probe – when aligned to a CAD model

Objective: Create the measured surface component by probing data points and compensating them.

Required: The project contains nominal surface components. A probing device is ready to use and is aligned to a CAD model.

Steps

1. In the Tree View, select the surface feature that will be probed.
2. Choose *Measure > Features > Define Measured*.



Alternatively, right-click the selected feature in the Tree View and choose *Probe Measured*.



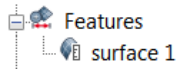
The Define Measured Feature Components dialog box opens.

3. In the **Method** list, select **Probe**.
4. Select **Compensate when probing ends** to compensate the probed data points using the normal orientation of the surface.
5. Click **Probe**.
6. Probe the measured surface feature.

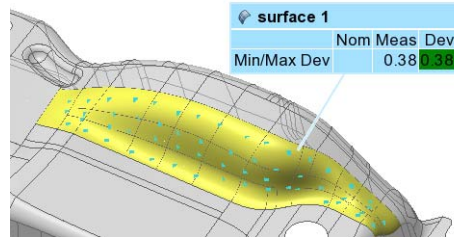
Creation method	
Method:	Probe
Parameters	
<input checked="" type="checkbox"/>	Compensate when probing ends

Results

The measured component is probed and a measured status icon is added to the surface feature in the Tree View, under the Features branch.



The feature is displayed in the 3D scene.



Probe – without being aligned to a CAD model

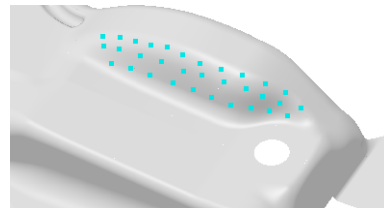
Sometimes, large parts need to be measured in a short period of time for reasons as diverse as part availability, safety reasons, or temperature variations. The measured component of the surface feature can be probed without having a nominal component and without the need to be aligned to a CAD model. The data points of the measured surface feature can later be compensated using the nominal components of surface features.

The procedure involves the following steps, which are detailed hereafter:

1. Probe the surface feature



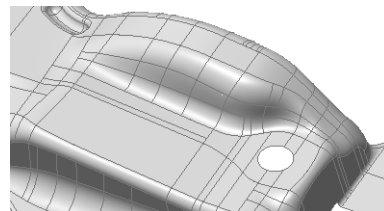
Probe the surface feature on the part. The probed data points are stored in a “probed surface” Data object.



2. Import the CAD model



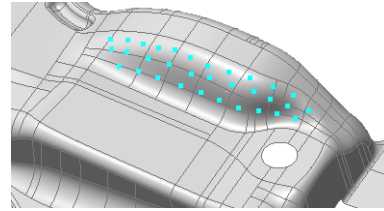
Import the CAD model into the inspection project.



- Align the probed data to the CAD model



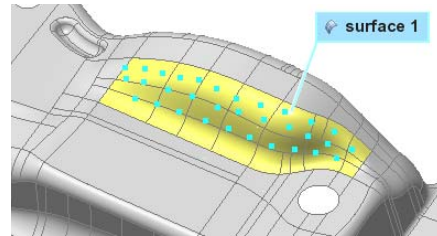
Align the probed data to the CAD model using one of the available methods, including the Best-Fit Data to Reference Objects alignment or the Best-Fit Measurement Objects alignment.



- Define the nominal component of the surface feature

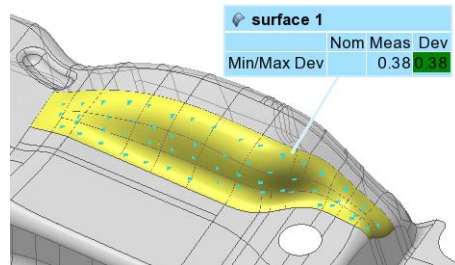


Define the nominal component of the surface feature. It will be used to compensate the probed data points and subsequently used to calculate the deviations.



- Compensate the data points

Compensate the probed data points to obtain the measured component of the surface feature. Once the data points are compensated, the deviations of the measured component to the nominal component are obtained.



Probe the surface feature

Objective: Measure a surface feature by probing it.

Required: A probing device is ready to be used.

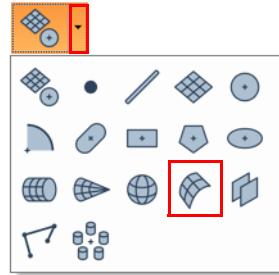
Steps

- Choose *Measure > Features > Create*.

The Create Features dialog box opens.

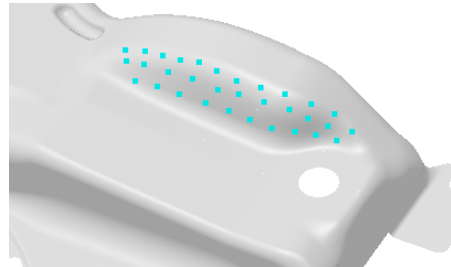
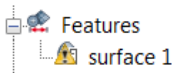


2. Expand the feature type drop-down menu and choose **Surface**.
3. Set the **Method** to **Probe**.
4. Click **Probe**.
5. Probe the surface feature.



Results

The surface feature is created and measured, but not yet compensated. It is added to the Tree View, under the Features branch. A warning icon indicates that the probed points are not yet compensated.



Define the nominal component of surface features

Once the CAD model is imported and the probed data points are aligned to the CAD model, define the nominal component of the surface feature.

When uncompensated probed data points are available and are aligned to the CAD model, Reference elements can be found automatically using the measured component of the surface feature. An option is available to quickly select Reference elements.

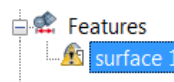
For information on how to import CAD models and proceed with an alignment, see [Importing a CAD model](#) on page 76 and [Introducing alignments](#) on page 130.

Objective: Define the nominal component of a surface feature.

Required: The measured component is probed (uncompensated data points), and the probed points are aligned to the CAD model.

Steps

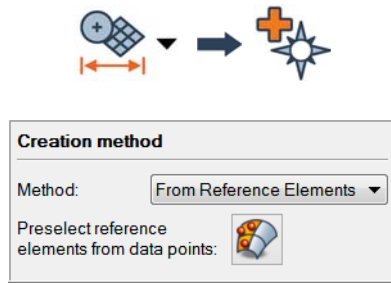
1. In the Tree View, select the surface feature.



2. Choose *Measure > Features > Define Nominal*.

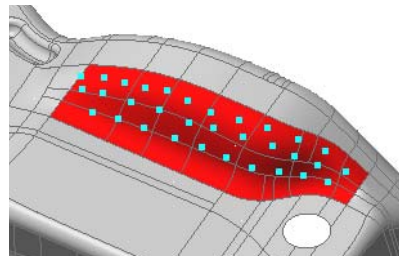
The Define Nominal Feature Components dialog box opens.

3. In the **Method** list, select **From Reference Elements**.



4. Click the **Preselect reference elements from data points** button to select Reference elements matching the data points of the measured components.

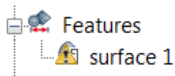
If desired, modify the selection of elements using the interactive selection mode.



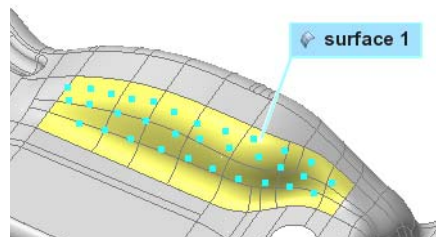
5. Click **Create**.

Results

The nominal component of the surface feature is created.



The feature is displayed in the 3D scene.



Compensate data points

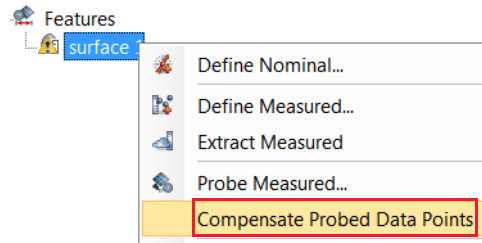
Once the probed data points are aligned to the CAD model, it is possible to compensate them to obtain the correct measured values of the surface feature.

Objective: Compensate a measured surface feature.

Required: The nominal component of the surface feature has been defined.

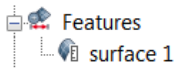
Steps

1. Select the surface feature in the Tree View.
2. Right-click and choose **Compensate Probed Data Points**.

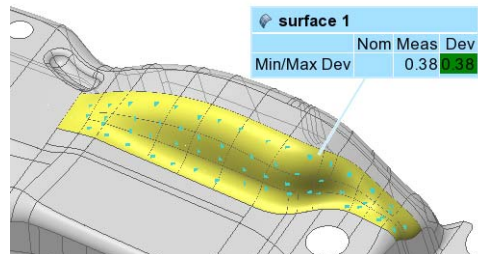


Results

The surface feature is now correctly measured. A measured status icon confirms the measured status of the surface feature.



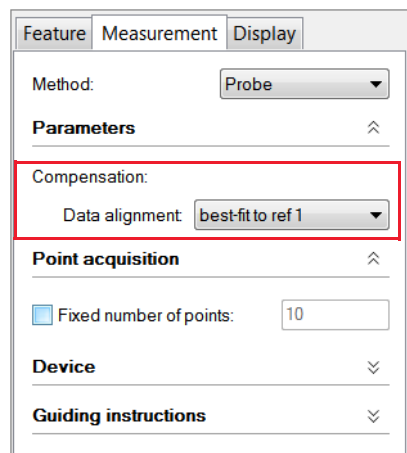
The probed data points are compensated using the normal orientation of the nominal component of the surface feature. The surface feature annotation shows the measured values for controls.



Notes

- The data alignment context in which a surface is compensated is saved to the **Measurement** tab of its property sheet.

To view the data alignment context, choose *Edit > Object Properties*, then click the **Measurement** tab.



- It is possible to change the data alignment used for compensation. When it is changed, select the surface feature in the Tree View, right-click, then choose **Compensate Probed Data Points**.

Probe existing surface comparison points

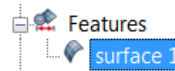
Surface features also support **Probe Surface Comparison Points** as a measurement method for probing. This method consists in probing surface comparison points linked to the surface feature in order to measure the surface feature. In this context, the same guidance mechanism offered when probing surface comparison points provides guidance when measuring the surface feature.

Objective: Measure a surface feature by probing existing surface comparison points linked to the surface feature.

Required: The project contains a Reference object and a surface feature, and the probing device must be aligned to the Reference object.

Steps

1. In the Tree View, select a surface feature.

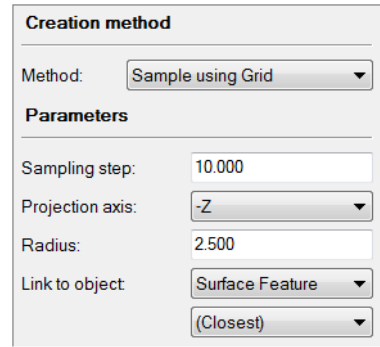


2. Choose *Measure > Comparison Points > Create > Surface Points*.

The Create Surface Comparison Points dialog box opens.

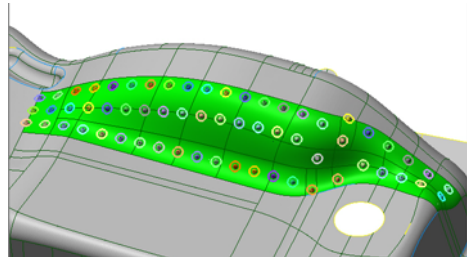


3. In the **Method** list, select **Sample using Grid**.
4. Specify the **Sampling step**.
Here, the value is specific to the part.
5. Specify the **Projection axis**.
6. Specify the **Radius**.
7. In the **Link to object** list, select **Surface Feature**.
8. Click **Create**.



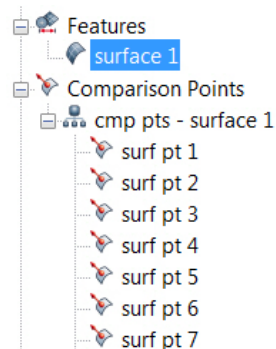
The surface comparison points are created on the surface feature and added to the Tree View in a group named according to the surface feature.

Each surface comparison point is automatically linked to the surface feature upon creation.



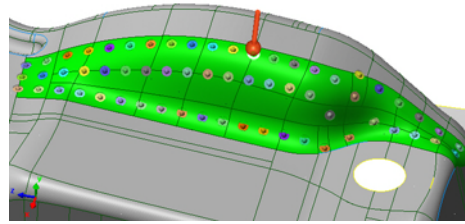
9. In the Tree View, select the surface feature to measure.
10. Right-click and choose *Probe Measured Comparison Points*.

The Probe Measured Comparison Point Components dialog box opens.



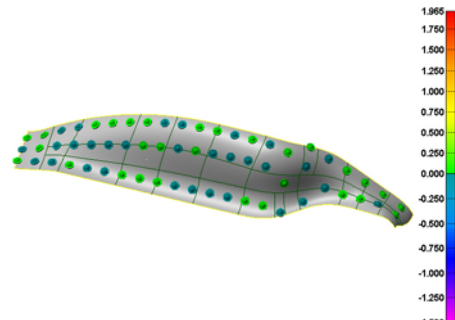
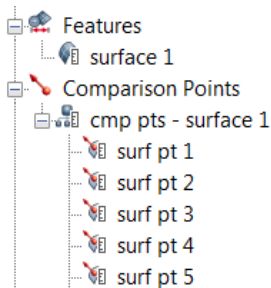
11. Click **Probe** to probe the surface comparison points following the guidance.

surf pt 17 of surface 1
0/1



Results

The surface feature and the surface comparison points are displayed as measured in the Tree View and in the 3D scene.



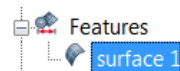
Probe new surface comparison points

Objective: Measure a surface feature by probing new surface comparison points in real time.

Required: The project must contain a Reference object and a surface feature, and the probing device must be aligned to the Reference object.

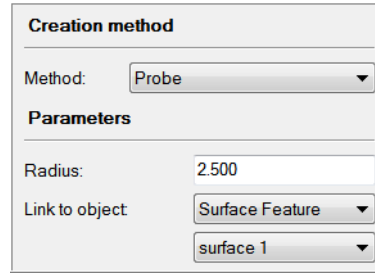
Steps

1. In the Tree View, select a surface feature.
2. Right-click and choose *Probe New Comparison Points*.

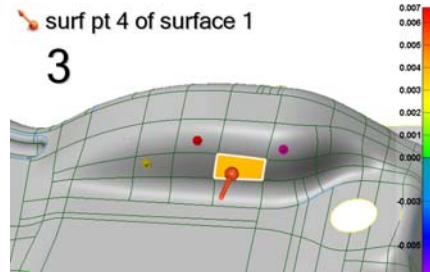


The Create Surface Comparison Points dialog box opens. The **Method** is set to **Probe** and the **Link to object** information is automatically set.

3. Specify the **Radius**.
4. Click **Probe**.
5. Probe new surface comparison points on the corresponding surface.

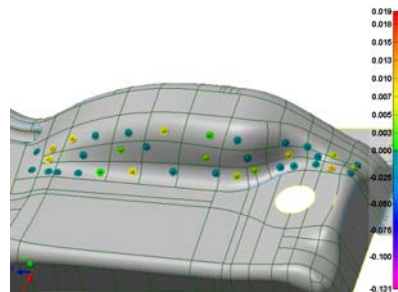
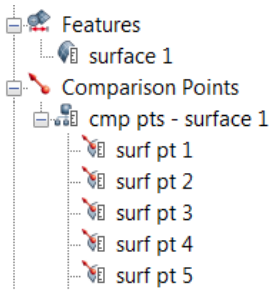


The surface feature is automatically highlighted in the 3D scene to guide the probing of the surface comparison points.



Results

The surface feature is measured. The surface comparison points are created, measured, linked to the surface feature and added to the Tree View in a group named according the surface feature.



From data points

Objective: Create measured surface components interactively using data elements.

Required: The project contains a Data object.

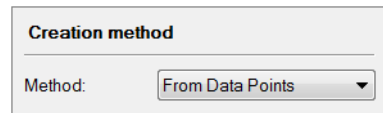
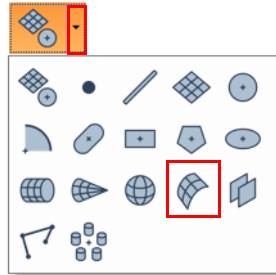
Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Surface**.
3. In the **Method** list, select **From Data Points**.



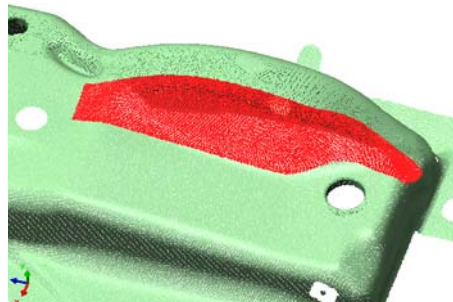
4. Choose *Select > Elements > Interactively*.



5. In the 3D scene, select the elements that will be used to fit the surface.

See [Selecting elements](#) on page 63 for information on how to select elements interactively.

6. Click **Create** to create the feature using the selected elements.

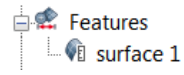


Tips

To define the measured component of an existing feature, select the feature in the Tree View and choose *Measure > Features > Define Measured*.

Results

The surface feature is added to the Tree View, under the Features branch. Surface features created from data points have no display in the 3D scene.



Creating slab features

Use slab features to obtain measurements on grooves or keyways with parallel sides.

Objective: Create a slab between two parallel planes.

Required: The project contains two parallel plane features having opposite orientations.

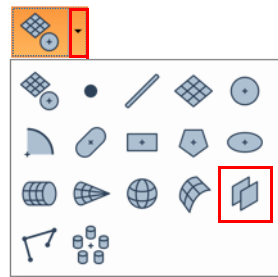
Steps

1. Choose *Measure > Features > Create*.

The Create Slab Features dialog box opens.



2. Expand the feature type drop-down menu and choose **Slab**.

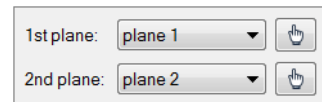


3. Specify a **Name** and assign a **Datum feature label**, if needed.
4. Choose a **1st plane** and a **2nd plane**, between which the slab will be created.

If two features are selected prior to accessing the creation dialog box, the fields are automatically populated.

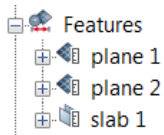
The nominal planes must be parallel and have opposite orientations.

5. Click **Create**.

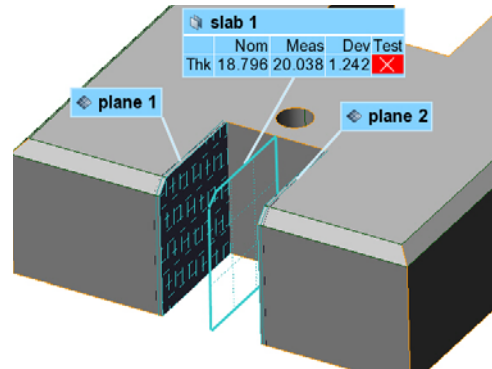


Results

The slab feature is added to the Tree View, under the Features branch.



The slab feature is displayed in the 3D scene.



Notes

The slab feature is dependent on the two source features. If, for any reason, they are modified, the slab feature is automatically updated.

If the source features contain only nominal components, the slab feature therefore only contains a nominal component. If both source features contain measured components, the slab feature therefore contains a measured component.

Creating polyline features

Polyline features are typically created using existing objects or by data acquisition and they generally contain a measured primitive. The nominal primitive is rarely used for the measurement operations.

Creating nominal polyline primitives

Objective: Create a nominal polyline primitive interactively using a Reference object or a Data object.

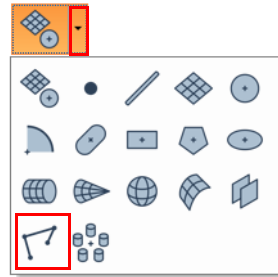
Required: The project contains a Reference object, a Data object, or a plane feature.

Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

2. Expand the feature type drop-down menu and choose **Polyline**.



3. In the **Method** list, select **Anchor**.
4. Specify whether the **Polyline type** is **Open** or **Closed**.
5. If required, select **Constraining plane** and specify a standard plane or an existing plane primitive to constrain the anchored points to a plane.

The interactive anchoring mode is automatically activated. If not, click **Anchor**.

Creation method

Method: Anchor

Parameters

Polyline type:

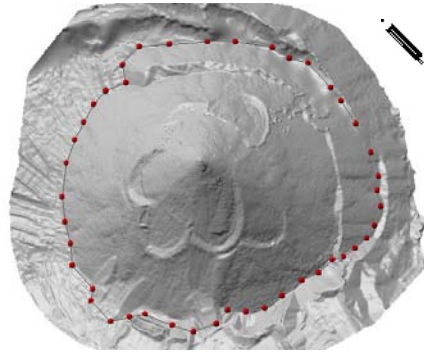
Open

Closed

Constraining plane: XY

Z: 0.000

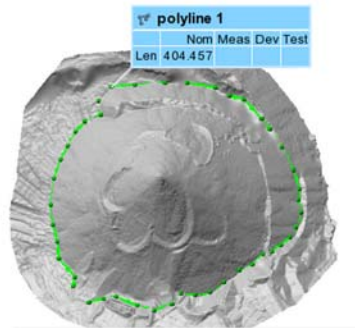
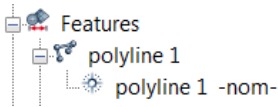
Left-click to anchor a point, middle-click to delete the last anchored point, and right-click to end the interactive picking mode.



Results

The polyline feature and its nominal primitive are added to the Tree View, under the Features branch.

The primitive is displayed in the 3D scene.



Obtaining measured polyline primitives

To obtain the measured primitive of a polyline feature, the most common methods used are **Probe**, **From Data Error Contours**, and **From Tracking**. Each method is explained in the following subsections.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

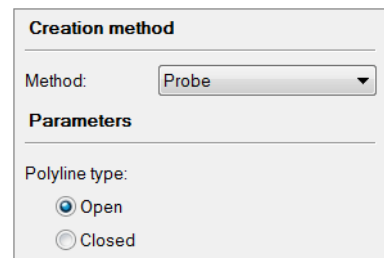
Probe

Objective: Create the measured polyline primitive by probing it.

Required: A probing device is ready to use.

Steps

1. Choose *Measure > Features > Create*.
The Create Features dialog box opens.
2. Expand the feature type drop-down menu and choose **Polyline**.
3. In the **Method** list, select **Probe**.
4. Specify whether the **Polyline type** is **Open** or **Closed**.
5. Click **Probe** to probe the measured polyline primitive.

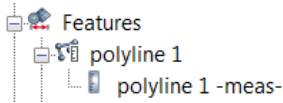


Click the **How To Window** button of the interactive mode guidance toolbar to view the different steps required to probe the polyline.

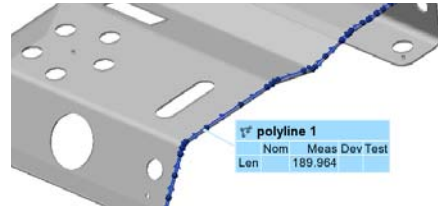


Results

The probed measured primitive is added to the Tree View, under the Features branch.



The polyline is displayed in the 3D scene.



Notes

- When probing, probed points are collected at the center of the probe tip. It is not possible to compensate as the polyline is being probed.
- It is possible to project the polyline onto a plane or offset it to apply a compensation on 2D polylines.



From Data Error Contours

Objective: Create the measured polyline primitive from data error contours.

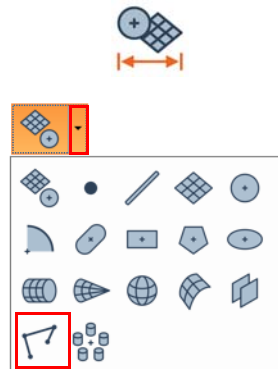
Required: The project contains an active data color map on a polygonal model colored using the Enhanced coloring option with a color scale that has 25 colors or less.

Steps

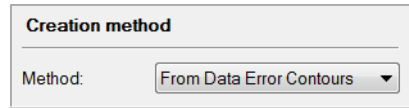
1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

2. Expand the feature type drop-down menu and choose **Polyline**.



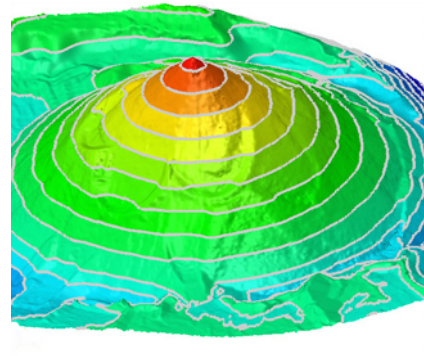
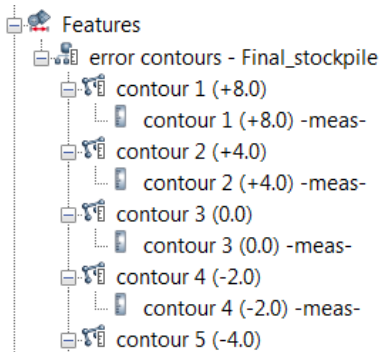
- In the **Method** list, select **From Data Error Contours**.
- Click **Create**.



Results

The measured polyline primitives are added to the Tree View and are automatically grouped, under the Features branch.

The polylines are displayed in the 3D scene.



From Tracking

Objective: Create the measured polyline primitive from tracking.

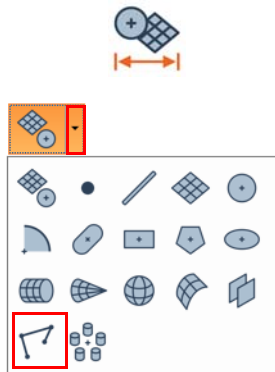
Required: The project contains a Data object.

Steps

- Choose *Measure > Features > Create*.

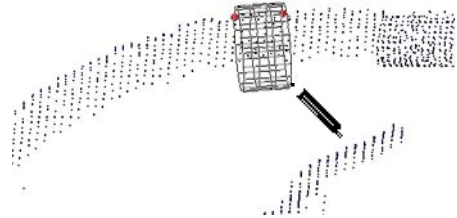
The Create Features dialog box opens.

- Expand the feature type drop-down menu and choose **Polyline**.



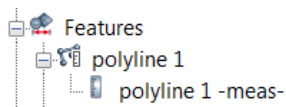
3. In the **Method** list, select **From Tracking**.
4. In the **Submethod** list, select a submethod.
5. In the 3D scene, pick three points on the Data object to define the tracking start point, search direction, and search radius.

Creation method	
Method:	From Tracking
Submethod:	Pipe

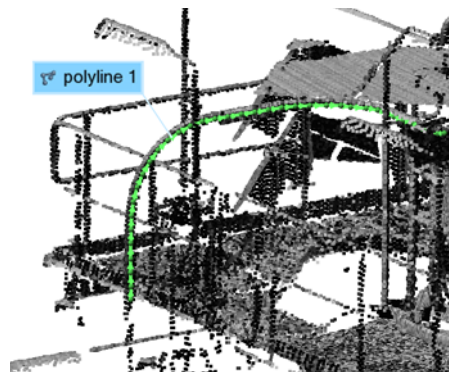


Results

The measured polyline primitive is added to the Tree View, under the Features branch.



The polyline feature is displayed in the 3D scene.



Constructing polyline features

Objective: Construct a polyline feature using existing objects.

Required: The project contains existing objects required by the specified submethod.

Steps

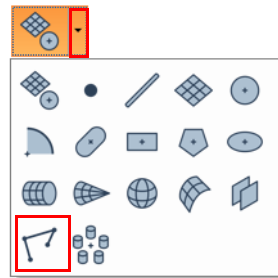
1. In the Tree View, select the objects required for the construction.



2. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

3. Expand the feature type drop-down menu and choose **Polyline**.

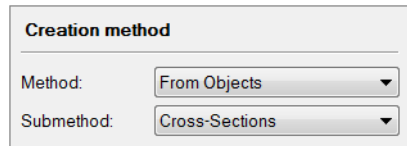


4. In the **Method** list, select **From Objects**.

5. In the **Submethod** list, select a submethod.

6. Click **Create**.

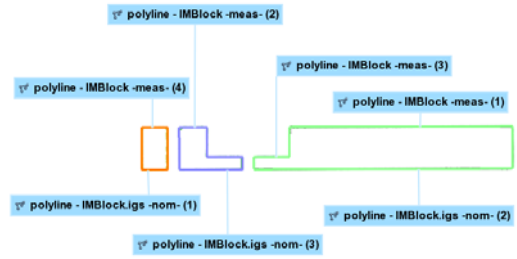
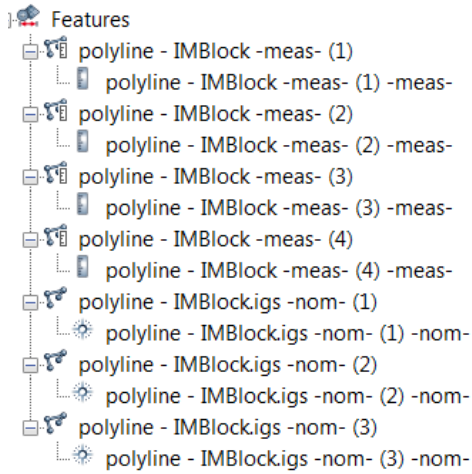
Example: The **Cross-Sections** submethod creates a polyline for each segment of the selected nominal and measured cross-section.



Results

The constructed polyline features are added to the Tree View, under the Features branch. If the objects used have nominal and measured components, the corresponding polylines are created separately.

The polyline features are displayed in the 3D scene.



Creating pattern features

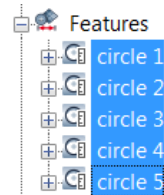
Patterns are created by grouping together features of the same type. The pattern is necessary when a GD&T control is assigned to a group of features, such as it is often the case on a Position control for holes.

Objective: Create a pattern of holes.

Required: The project contains at least two features of the same type.

Steps

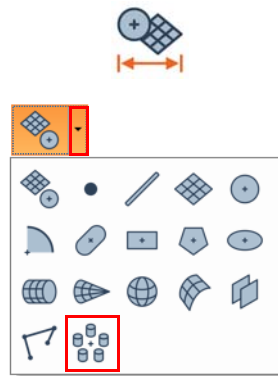
1. In the Tree View, select the features that will be used.



2. Choose *Measure > Features > Create*.

The Create Features dialog box opens.

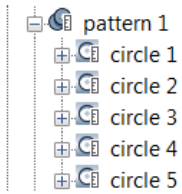
3. Expand the feature type drop-down menu and choose **Pattern**.



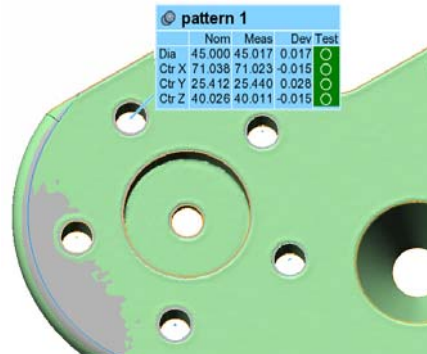
4. Specify a **Name** and assign a **Datum feature label**, if required.
5. Click **Create**.

Results

The pattern feature is added to the Tree View, under the Features branch. The features are grouped under the pattern feature.



The pattern feature is displayed in the 3D scene and the annotation is attached to one of its features.



Creating distance features

Use distance features to measure the distance between two existing features.

Objective: Create a distance dimension between two features.

Required: The project contains two valid features.

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Select **Distance**.



3. Choose a **Reference feature** and a **Feature** between which the distance will be measured.

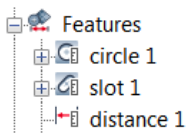
*Click the hand symbol adjacent to the list boxes to pick features in the 3D scene. If two features are selected prior to accessing the creation dialog box, the fields are automatically populated. When necessary, the **Reference feature** is considered to be extended to infinity.*

Reference feature:	circle 1	
Feature:	slot 1	

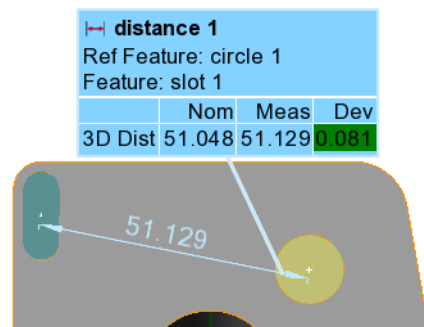
4. Click **Create**.

Results

The distance feature is added to the Tree View, under the Features branch.



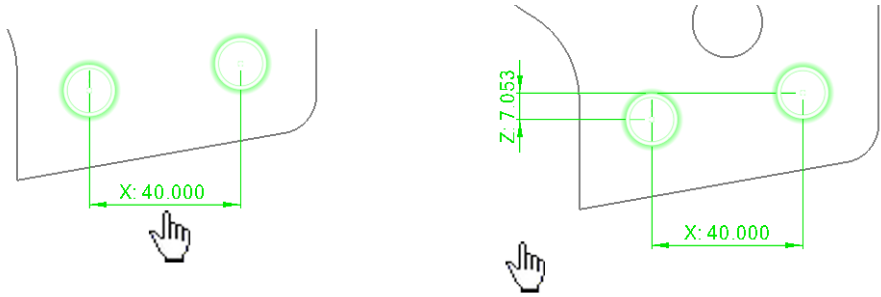
The distance feature is displayed in the 3D scene.



Notes

- When the pick tool (hand symbol) or the list boxes are used to specify the features to use, a third left-click is possible to position the distance annotation in the 3D scene.

Depending on the position of the pointer with respect to the features used, different controls are activated.



Once the third left-click is performed, the distance is automatically created.

- The controls shown in the 3D scene are the ones activated in the Geometry Controls pane for the distance feature that was created.
- The distance feature is dependent on the source features and is automatically updated if the source features are modified.
- If the source features have a nominal component, the resulting distance feature also has a nominal component. If all the source features have a measured component, the resulting distance feature also has a measured component.

Creating distance features on cross-sections

Distance features can be created on cross-sections for 2D inspection purposes. A cross-section object must exist and be specified upon creation of these features.

For more information, see [Creating features on cross-sections for 2D inspection](#) on page 216.

Objective: Create a cross-section distance feature.

Required: The project contains a valid cross-section and two valid features.

Steps

1. Choose *Measure > Features > Create*.
The Create Features dialog box opens.

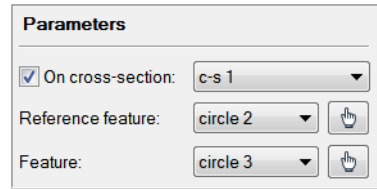


2. Select **Distance**.



3. Enable the **On cross-section** parameter and choose from the list of cross-sections.
4. Choose a **Reference feature** and a **Feature** between which the distance will be measured.

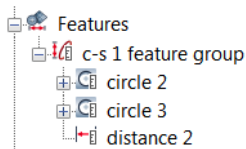
*Click the hand symbol adjacent to the list boxes to pick features in the 3D scene. If two features are selected prior to accessing the creation dialog box, the fields are automatically populated. When necessary, the **Reference feature** is considered to be extended to infinity.*



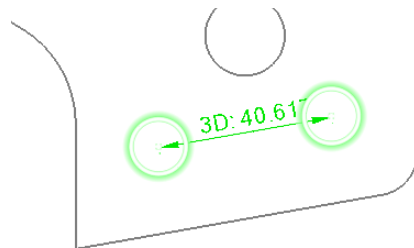
5. Click **Create**.

Results

The distance feature is added to the Tree View, under the Features branch, and is also grouped in a cross-section feature group.



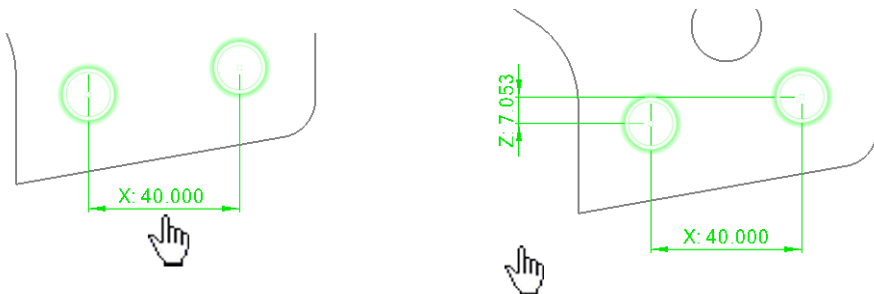
The distance feature is displayed in the 3D scene.



Notes

- When the pick tool is used (hand symbol) to specify the features to use, a third left-click is required to position the distance annotation in the 3D scene.

Depending on the position of the pointer with respect to the features used, different controls are activated.



Once the third left-click is performed, the distance is automatically created.

- The controls shown in the 3D scene are the ones activated in the Geometry Controls pane for the distance feature that was created.

Creating angle features

Use angle features to measure the angle between two existing features.

Objective: Create an angle dimension between two features.

Required: The project contains two features.

Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Select **Angle**.



3. Choose a **Reference feature** and a **Feature**, between which the angle will be measured.

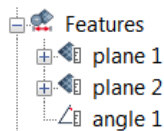
*Click the hand symbol adjacent to the list boxes to pick features in the 3D scene. If two features are selected prior to accessing the creation dialog box, the fields are automatically populated. When necessary, the **Reference feature** is considered to be extended to infinity. The measurement is calculated from the **Reference feature** to the **Feature**.*

Reference feature:	plane 1	
Feature:	plane 2	

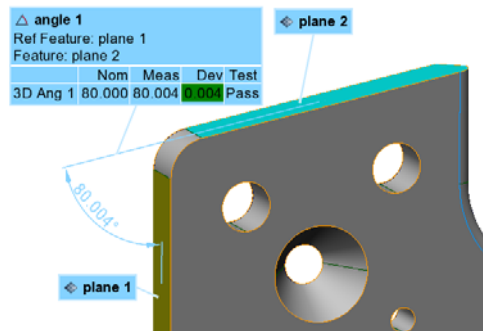
4. Click **Create**.

Results

The angle feature is added to the Tree View, under the Features branch.



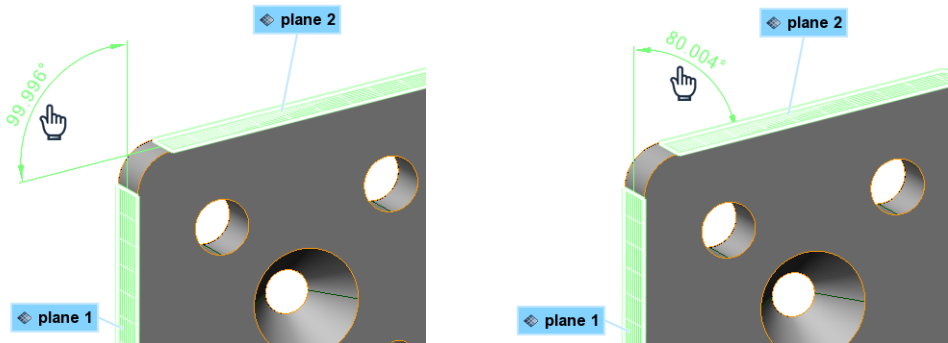
The angle feature is displayed in the 3D scene.



Notes

- When the pick tool (hand symbol) or the list boxes are used to specify which features to use, a third left-click is possible to position the angle annotation in the 3D scene.

Depending on the position of the pointer with respect to the features used, different controls are activated.



Once the third left-click is performed, the angle is automatically created.

- The controls shown in the 3D scene are the ones activated in the Geometry Controls pane for the angle feature that was created.
- The angle feature is dependent on the source features and is automatically updated if the source features are modified.
- If the source features have a nominal component, the resulting angle feature also has a nominal component. If all the source features have a measured component, the resulting angle feature also has a measured component.

Creating angle features on cross-sections

Angle features can be created on cross-sections for 2D inspection purposes. A cross-section object must exist and be specified upon creation of these features.

For more information, see [Creating features on cross-sections for 2D inspection](#) on page 216.

Objective: Create a cross-section angle feature.

Required: The project contains a valid cross-section and two valid features.

Steps

1. Choose *Measure > Features > Create*.

The Create Features dialog box opens.



2. Select **Angle**.



3. Enable the **On cross-section** parameter and choose a cross-section from the list.
4. Choose a **Reference feature** and a **Feature**, between which the angle will be measured.

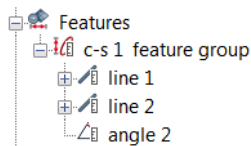
*Click the hand symbol adjacent to the list boxes to pick features in the 3D scene. If two features are selected prior to accessing the creation dialog box, the fields are automatically populated. When necessary, the **Reference feature** is considered to be extended to infinity. The measurement is calculated from the **Reference feature** to the **Feature**.*

Parameters	
<input checked="" type="checkbox"/> On cross-section:	c-s 1
Reference feature:	line 1
Feature:	line 2

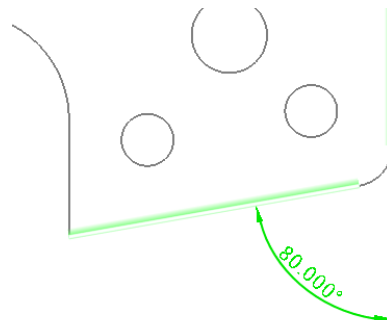
5. Click **Create**.

Results

The angle feature is added to the Tree View, under the Features branch, and is also grouped in a cross-section feature group.



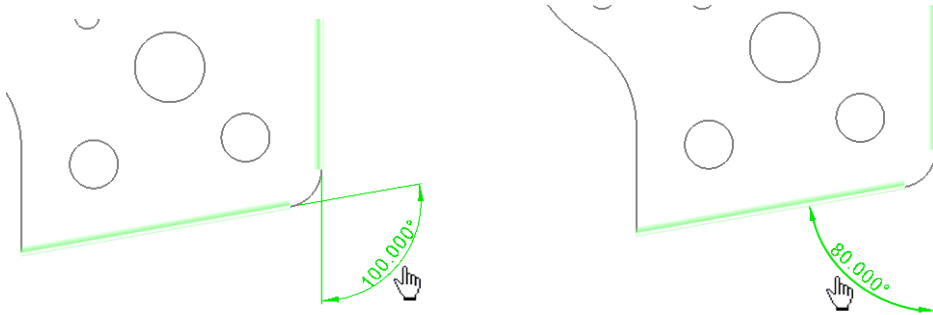
The angle feature is displayed in the 3D scene.



Notes

- When the pick tool is used (hand symbol) to specify which features to use, a third left-click is required to position the angle annotation in the 3D scene.

Depending on the position of the pointer with respect to the features used, different controls are activated.

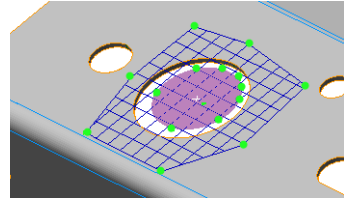


Once the third left-click is performed, the angle is automatically created.

- The controls shown in the 3D scene are the ones activated in the Geometry Controls pane for the angle feature that was created.

Editing feature primitives

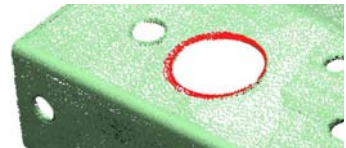
Once a measured primitive has been extracted or probed, it can be reviewed and edited to refine the measured value. For example, it is possible to select the measured data points used to create the primitive, edit the measured data points to see the coordinates and fitting errors of each point, add probed data points to an existing primitive or set measured primitive boundaries using the nominal primitive.



Note that other feature editing tools are available on the *Measure > Feature Primitives* menu. A feature can also be edited by adjusting parameters on the Measurement tab of its properties.

Selecting data points of measured primitives

It is useful to highlight the data points used in calculating a measured primitive or its local constraining plane. The selection shows whether unwanted data points have been included. This allows making adjustments and recalculating the measured primitive.

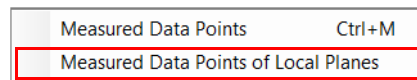
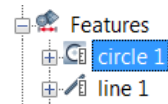


Objective: View the points used to fit the measured feature component.

Required: The project contains a Data object and a measured feature.

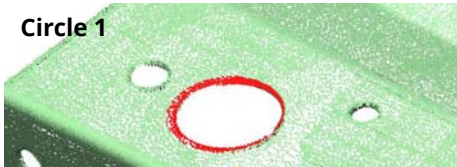
Steps

1. Select a feature in the Tree View and choose *Select > Data Points > Measured Data Points*.
2. To view the points used for the local constraining plane of a curve-based feature, choose *Select > Data Points > Measured Data Points of Local Planes*.

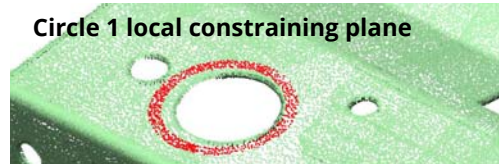


Results

The data points used to calculate the measured primitive are selected in the 3D scene.



If specified, the data points used to calculate the local constraining plane of the measured primitive are selected in the 3D scene.



Editing the measured points of extracted feature primitives

Using the Edit Measured Points interface, the exact fitting statistics of each data point used in the extracted measured primitive can be reviewed. If necessary, individual points can be removed from the solution to improve the overall result.

Objective: Review measured feature point fitting statistics and remove unwanted data points.

Required: The project contains a Data object, and an extracted measured feature component.

Steps

1. Select a feature in the Tree View and choose *Measure > Features > Edit Measured Points*.



Each point used in the measured feature component fit is listed, with its coordinates and deviation. They can be sorted by column value by clicking on the column header.

2. Under **Fit statistics**, a summary of current and initial fitting values is displayed.
3. In the **Fit parameters** section, the fitting parameters specified during the feature extraction are shown.
4. To remove points from the fitting solution, clear them from the list or right-click and choose **Ignore**. The current fit statistics will update.
5. Click **Apply**.

Points Points of local plane

Index	Deviation	X	Y	Z
<input checked="" type="checkbox"/> 1	0.019	83.459	24.302	42.278
<input checked="" type="checkbox"/> 2	0.014	83.507	24.230	42.029
<input checked="" type="checkbox"/> 3	-0.000	83.500	23.908	42.159
<input checked="" type="checkbox"/> 4	-0.013	83.534	23.522	42.040
<input checked="" type="checkbox"/> 5	-0.005	83.473	23.684	42.339
<input checked="" type="checkbox"/> 6	-0.003	83.544	23.842	41.909
<input checked="" type="checkbox"/> 7	0.015	83.526	24.561	41.902
<input type="checkbox"/> 8	0.008	83.551	24.879	41.773
<input type="checkbox"/> 9	0.021	83.555	24.500	41.650

Fit statistics

	Current	Initial
No. of points:	2733	2735
Min deviation:	-0.050	-0.050
Max deviation:	0.047	0.047
Mean deviation:	0.000	-0.000
Std deviation:	0.019	0.019
RMS deviation:	0.019	0.019

Fit parameters

Fit type: Best-fit

Reject outliers: On (5.0%)

Results

The **Fit statistics** are updated to display the current fitting summary, and the measured feature component is updated.

	Current	Initial
No. of points:	2733	2733
Min deviation:	-0.050	-0.050
Max deviation:	0.047	0.047
Mean deviation:	0.000	0.000
Std deviation:	0.019	0.019
RMS deviation:	0.019	0.019

Notes

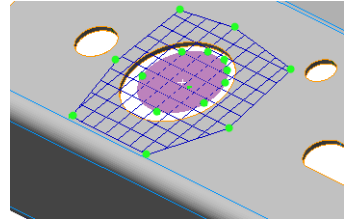
For surface features, statistics report the deviation between the measured points and the nominal component, as no fitting is performed.

Statistics

	Current	Initial
No. of points:	10912	10912
Min deviation:	-0.036	-0.036
Max deviation:	0.147	0.147
Mean deviation:	0.044	0.044
Std deviation:	0.026	0.026
RMS deviation:	0.051	0.051

Editing the measured points of probed feature primitives

The Edit Measured Points interface can be used to review the exact fitting statistics of each point probed for the measured feature component. If necessary, individual points can be removed from the solution to improve the overall result. Additionally, the interface allows for the review and correction of probing compensation and constraints.



Objective: Review probed feature point fitting statistics, remove unwanted points, and correct compensation errors.

Required: The project contains a Data object and a probed measured feature component.

Steps

1. Select a feature in the Tree View and choose *Measure > Features > Edit Measured Points*.



Points tab:

2. Each point used in the probed feature component fit is listed, with its coordinates and deviation. Points can be sorted by column value, by clicking on the column header.
3. Under **Fit statistics**, a summary of current and initial fitting values is displayed.
4. In the **Fit parameters** section, the fitting parameters used for probing are shown.
5. To remove points from the fitting solution, clear them from the list or right-click and choose **Ignore**. The current fit statistics will be updated.
6. Click **Apply**.

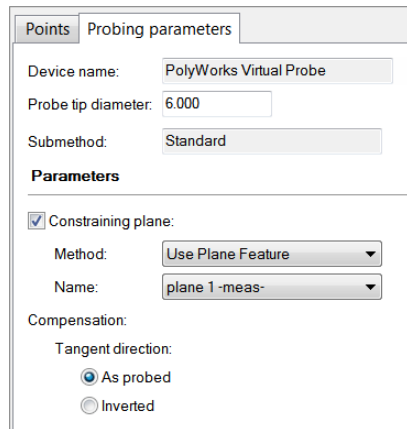
Index	Deviation	X	Y	Z
<input checked="" type="checkbox"/> 1	0.145	79.667	18.983	48.574
<input checked="" type="checkbox"/> 2	0.067	71.944	21.122	52.846
<input checked="" type="checkbox"/> 3	-0.044	61.562	23.206	49.310
<input checked="" type="checkbox"/> 4	-0.005	57.800	23.165	38.709
<input checked="" type="checkbox"/> 5	0.089	61.535	21.650	31.015
<input checked="" type="checkbox"/> 6	-0.131	72.490	18.839	27.570
<input checked="" type="checkbox"/> 7	0.108	81.694	17.411	35.612
<input type="checkbox"/> 8	-0.064	82.784	17.631	41.123
<input checked="" type="checkbox"/> 9	-0.229	79.660	19.033	49.146

Fit statistics		
	Current	Initial
No. of points:	8	9
Min deviation:	-0.229	-0.005
Max deviation:	0.145	0.010
Mean deviation:	-0.000	-0.000
Std deviation:	0.129	0.005
RMS deviation:	0.121	0.005

Fit parameters	
Fit type:	Best-fit
Reject outliers:	Off

Probing parameters tab:

7. The **Probe tip diameter** can be adjusted if incorrect.
8. The **Constraining plane** parameter, and the plane used for curve-based features, can be redefined.
9. The **Compensation**, if incorrectly performed, can be reversed.
10. Click **Apply**.

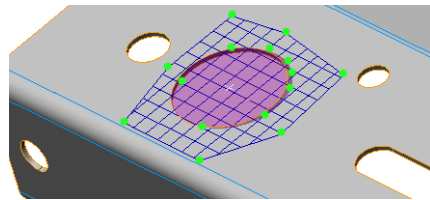


Results

The **Fit statistics** are updated to display the current fitting summary.

The probed feature component is updated.

	Current	Initial
No. of points:	8	8
Min deviation:	-0.229	-0.229
Max deviation:	0.145	0.145
Mean deviation:	-0.000	-0.000
Std deviation:	0.129	0.129
RMS deviation:	0.121	0.121



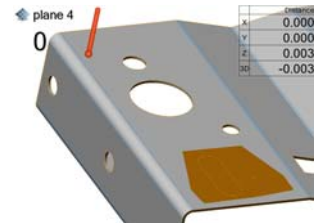
Notes

For surface features, statistics report the deviation between the measured points and the nominal component, as no fitting is performed.

Statistics		
	Current	Initial
No. of points:	69	69
Min deviation:	-0.127	-0.127
Max deviation:	0.127	0.127
Mean deviation:	-0.005	-0.005
Std deviation:	0.057	0.057
RMS deviation:	0.057	0.057

Adding points to a probed feature

If a probed feature contains too few points for the desired inspection, more points can be added and incorporated into the probed feature fitting solution. Adding probed points is useful in creating a more robust feature fit, or to continue defining a very large feature from a later device position.



Objective: Add additional probed points to a measured feature component.

Required: The project contains a Data object, and a probed feature.

Steps

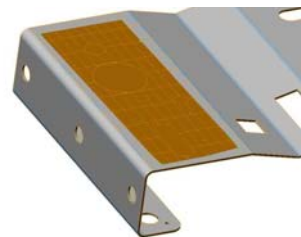
1. Select the feature in the Tree View and choose *Measure > Feature Primitives > Add Probed Data Points*.
2. Specify the required **Point acquisition** parameters.
3. Click **Probe**.



Primitive selection	
Primitive:	plane 3-meas-
Point acquisition	
Mode:	Standard
<input type="checkbox"/> Fixed number of points:	1

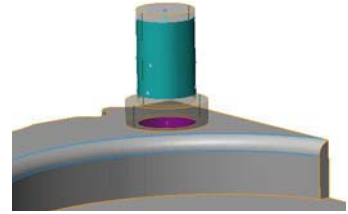
Results

The probed feature component is updated to include the newly-probed points.



Setting measured feature boundaries using the nominal component

In some cases, a measured feature component cannot have the same extent as the nominal component. For instance, a pin is used to define a nominal hole, so the measured component is above the nominal hole boundary. It is then useful to adjust the measured feature component to fit within the nominal component boundaries. This functionality is available for surface-based features.

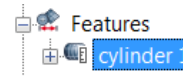


Objective: Set the measured feature extent to match the nominal feature component extent.

Required: A Data object; a surface feature with a nominal and measured component.

Steps

1. Select a feature in the Tree View and choose *Measure > Features > Set Measured Boundaries using Nominal*.

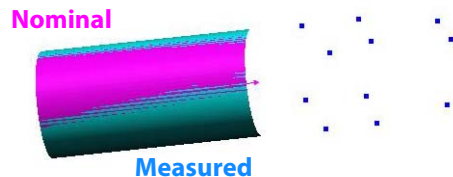
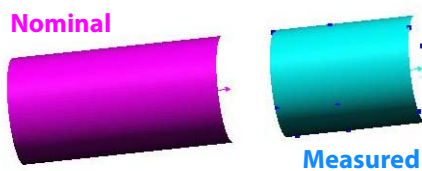


Alternatively, right-click the feature in the Tree View and choose *Edit > Set Measured Boundaries using Nominal*.

Results

The measured feature originally had the extent of the probed data points.

After processing, it is extended to the boundaries of the nominal feature component.



Exporting feature primitives

Most nominal and measured primitives can be exported for use outside of the PolyWorks Metrology Suite. All features, except for angles, distances, surfaces, and slabs, can be exported as an IGES file. Points, lines, planes, circles, and polylines can also be exported in text format.

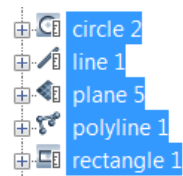


Objective: Export primitives as an IGES file.

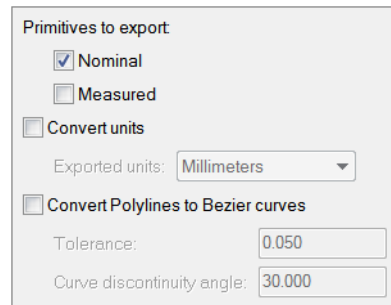
Required: The project contains features.

Steps

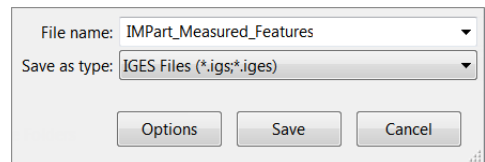
1. In the Tree View, select the features that will be exported and choose *File > Export > Features*.



2. Click **Options**.
3. Specify which feature primitives to export, either **Nominal**, **Measured**, or both.
4. Click **OK**.

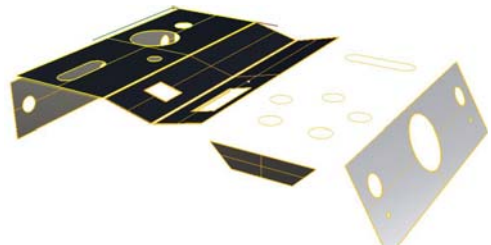


5. Navigate to the desired file-saving location.
6. Name the file.
7. Click **Save**.



Results

The selected primitives are saved to an IGES file and can be opened in another software package.



Cross-sections



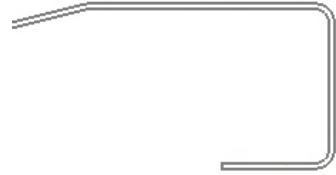
Standard cross-sections



Offset cross-sections

Inspecting 2D part profiles with cross-sections

Cross-sections are measurement tools used for 2D inspection of surfaces. They allow calculating deviations along profiles obtained from slicing planes.



Introducing cross-sections

Cross-sections are used to inspect a profile of a part along a cross-sectional plane using the nominal geometry and the geometry of the part in order to produce results determined by geometry controls. The nominal geometry and the part geometry are captured by intersecting aligned Reference and Data objects with a slicing plane.

Cross-sections can be created along an axis, around an axis (radially), along a curve, or using several other methods.

Cross-sections are available on the *Measure > Cross-Sections* menu or on the **Cross-Sections** toolbar.



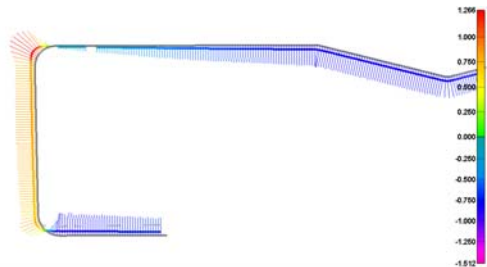
Types of cross-sections

There are two types of cross-sections:



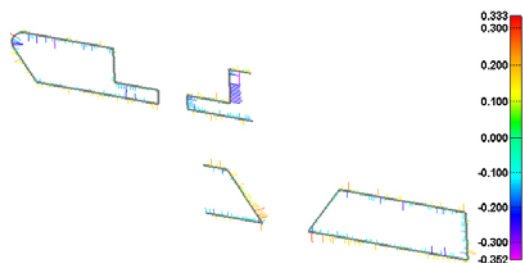
Standard cross-sections

A standard cross-section is created by intersecting aligned Reference and Data objects with a slicing plane delimited by a measurement zone.



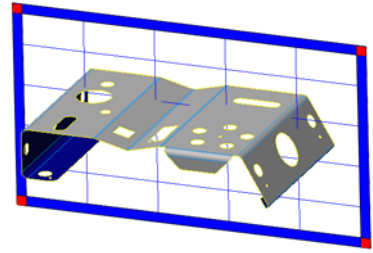
Offset cross-sections

An offset cross-section is created by intersecting aligned Reference and Data objects with parallel slicing planes to produce a single multistep cross-section. Offset cross-sections are useful in controlling 2D dimensions defined between features situated on different slicing planes.



Measurement zone

The measurement zone defines the 3D limits of the slicing process, delimiting a slicing plane. As shown to the right, it is represented by a blue rectangle with red corners. The dimensions of the measurement zone are defined during cross-section creation, and can be edited by dragging rectangle sides.



Measurement methods

There are two main measurement methods used to obtain the part geometry. **Extract** obtains the part geometry by slicing an aligned Data object, while **Probe** involves probing the part geometry on the part that is within the measurement zone.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

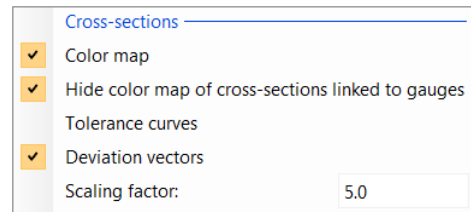
Display options

A number of display options are available to visualize cross-sections with captured nominal and part geometry.

They are offered in the **Object Display Options**.



Under **Cross-sections**, several display options are offered. They are only available when **Color map** is selected.



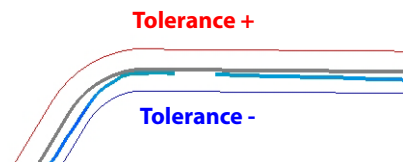
- **Color map**

Displays the deviation of the measured geometry to the nominal geometry. Note that the color map display can be customized. For information, see [Changing the color map display](#) on page 201.



- **Tolerance curves**

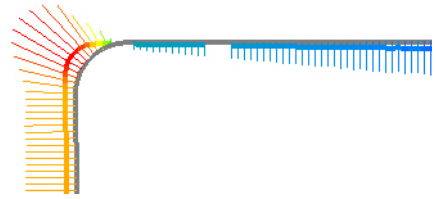
Displays the cross-section with respect to the tolerances of the Reference object, allowing a pass/fail representation.



- **Deviation vectors**

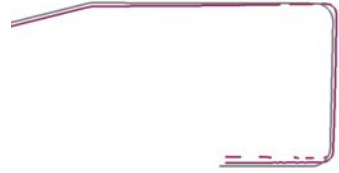
Displays the direction and amplitude of the deviations.

Increasing the **Scaling factor** increases the length of the deviation vectors for better visibility.



Creating standard cross-sections

A cross-section is a measurement object. It is created to obtain the deviation along a cross-sectional plane. The geometry of the part can be obtained by extracting it from an aligned Data object, or by probing the geometry on the part.



Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Start inspecting a part along a 2D profile by creating a cross-section to capture the nominal component.

Required: The project contains a Reference object. If the project also contains a Data object, the Data object must be aligned to the Reference object.

Steps

1. Choose *Measure > Cross-Sections > Create > Standard Cross-Sections*.

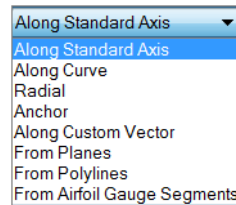
The Create Cross-Sections dialog box opens.



2. Choose a creation **Method**.

The creation of cross-sections using the most frequently used methods follows.

All of the possible creation methods are shown to the right.



Along Standard Axis

This method allows creating a cross-section perpendicular to a standard system axis, such as the X axis. It is recommended to have a properly oriented coordinate system to obtain the anticipated results.

1. In the **Method** list, select **Along Standard Axis**.
2. Under **Axis**, select **X**, **Y**, or **Z**.
The cross-section will be positioned along and perpendicular to the axis.

3. Under **Mode**, select **Individual** to create a single cross-section.
*The **Multiple** mode allows creating evenly spaced cross-sections at a specified step either within a range or by slicing whole objects.*
4. Under **Position**, specify a position along the axis using one of these methods:

- Numerically, by typing a value.
- Manually, by clicking **Anchor Point**, then picking a location on an object in the 3D scene.

5. Click **Preview**.

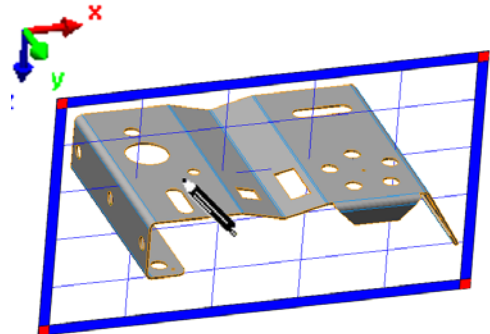
A preview of the measurement zone of the cross-section is displayed in the 3D scene.

The preview is displayed automatically when anchoring a point.

Right-click to exit the anchoring mode.

6. To resize the measurement zone to include only the area of interest, drag the blue borders.
7. Click **Create**.

Name:	<input type="text" value="c-s 1"/>
Method:	<input type="text" value="Along Standard Axis"/>
Axis	
<input checked="" type="radio"/> X	<input type="radio"/> Y
<input type="radio"/> Z	
Mode	
<input checked="" type="radio"/> Individual	<input type="radio"/> Multiple
Position	
x:	<input type="text" value="0.000"/>
<input type="button" value="Anchor Point"/>	
Options <input type="button" value="v"/>	
<input type="button" value="Preview"/>	



Along Curve

This method allows creating a cross-section perpendicular to a curve. It involves specifying the type of curve to use, and then specifying the location of the cross-section along the curve.

1. In the **Method** list, select **Along Curve**.
2. Under **Curve**, select **From reference object curves**.

The cross-section will be positioned along and perpendicular to a curve on the Reference object.

From polyline allows cross-sections to be located either along an existing polyline feature, or by interactively anchoring a polyline.

3. Under **Mode**, select **Individual** to create a single cross-section.

*The **Multiple** mode allows creating evenly spaced cross-sections at a specified step either within a range or by slicing whole objects.*

4. Under **Position**, click **Anchor Point**.

An interactive anchoring mode is launched.

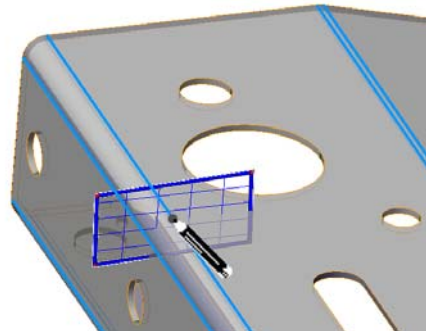
5. Anchor a point along the curve by picking a point on the Reference object in the 3D scene. Right-click to exit the anchoring mode.

*If **At intersection with plane** is selected, only the location of the intersection of the curve and the chosen standard plane can be picked.*

A measurement zone is displayed.

6. To resize the measurement zone to include only the area of interest, drag the blue borders.

7. Click **Create**.



Tips

Cross-sections can be created along a whole curve or just using a portion of the curve. When the **Mode** is set to **Multiple**, two anchoring modes become available:

- **Whole curve:** Cross-sections are created along the entire length of the curve.
- **Partial curve:** Cross-sections are created between the specified start and end point on the curve. Click a first point on the curve to indicate the start point and click a second point to indicate the end point.

Radial

This method allows creating a cross-section that radiates from an axial primitive, such as a cylinder or cone.

1. In the **Method** list, select **Radial**.
2. Under **Rotation axis**, select in the **Axial primitive** list a feature with an axis, such as a cone or a cylinder.

The cross-section will be positioned radially, originating from the selected axis.

3. Set the mode to **Individual** to create a single cross-section.

*The **Multiple** mode allows creating evenly spaced cross-sections at a specified angular step either within a range or by slicing whole objects.*

4. Under **Position**, specify a position using one of these methods:
 - Numerically, by typing a value.
 - Manually, by clicking **Anchor Point**, then picking a location on an object in the 3D scene.
5. Click **Preview**.

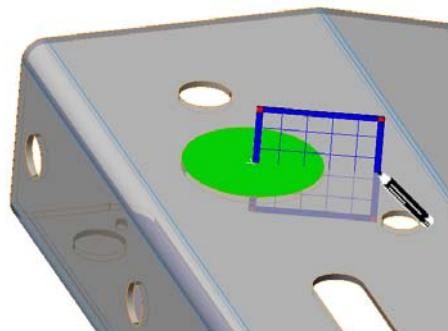
A preview of the measurement zone of the cross-section is displayed in the 3D scene.

The preview is displayed automatically when anchoring a point.

Right-click to exit the anchoring mode.

6. To resize the measurement zone to include only the area of interest, drag the blue borders.
7. Click **Create**.

The screenshot shows the settings for the Radial measurement tool. The 'Name' field contains 'c-s 1'. The 'Method' is set to 'Radial'. Under 'Rotation axis', the 'Axial primitive' is set to 'circle 1 -nom-'. The 'Mode' section has 'Individual' selected with a radio button, and 'Multiple' is unselected. The 'Position' section has three input fields: 'x' with '0.000', 'y' with '39.476', and 'z' with '24.477'. Below these is an 'Anchor Point' button. The 'Options' section is collapsed, and a 'Preview' button is at the bottom.



Anchor

This method allows creating a cross-section by anchoring a line in the 3D scene.

1. In the **Method** list, select **Anchor**.
2. Under **Mode**, select **Individual** to create a single cross-section.

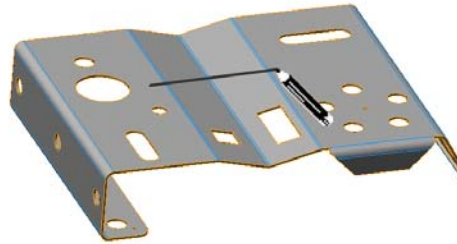
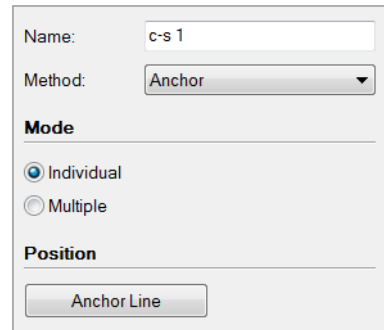
*The **Multiple** mode allows creating evenly spaced cross-sections at a specified step either within a range or by slicing whole objects.*

3. Under **Position**, click **Anchor Line** and click two locations in the 3D scene. Right-click to exit the anchoring mode.

A measurement zone is displayed in the 3D scene.

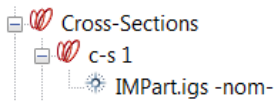
4. To resize the measurement zone to include only the area of interest, drag the blue borders.

5. Click **Confirm**.



Results

The new cross-section, with a nominal component, is added to the Tree View under the Cross-Sections branch.

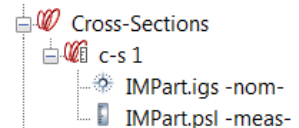


The cross-section is displayed in the 3D scene along with a color scale showing the deviations (requires both the nominal and the measured component to obtain results).

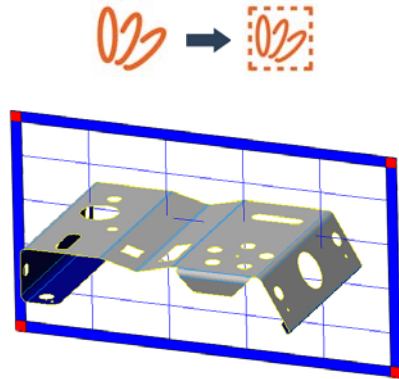


Notes

- If a Data object is available and aligned when the cross-section is created, then the measured component is extracted.



- The measurement zone of a cross-section can be edited post creation by following these steps:
 1. Select the cross-section in the Tree View.
 2. Choose *Measure > Cross-Sections > Edit Measurement Zones*.
 3. The measurement zone is displayed in the 3D scene and can be edited by dragging the borders.
 4. Press ESC to exit the editing mode.



Measuring the geometry on the part

For an existing cross-section, the corresponding geometry on the part can be obtained by extraction on an aligned Data object or by probing a physical part. If proceeding by probing, the digitizing device must be aligned to the Reference object. In PolyWorks|Inspector, the geometry obtained on the part is called the *measured component* of the measurement object.

Extract the geometry

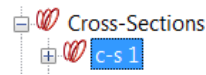
When there is a Data object available in the project at cross-section creation, the measured component is extracted automatically. Otherwise, when the Data object becomes available and is aligned, a manual extraction operation must be performed using the cross-section.

Objective: Finish inspecting the profile for an existing cross-section by extracting the corresponding profile geometry on the part to obtain the measured component of the cross-section.

Required: The project contains a Reference object, an aligned Data object, and a cross-section with a nominal component.

Steps

1. Select a cross-section in the Tree View.

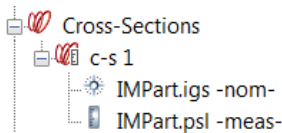


2. Choose *Measure > Extract Measured*.

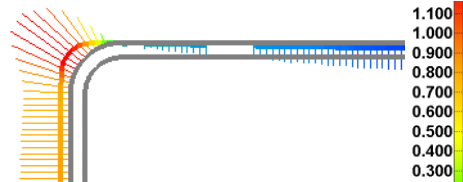


Results

The measured component is added to the cross-section. The measured status icon indicates its measured status.



Deviations are calculated using the nominal and the measured components and are displayed in the 3D scene using a color scale.



Notes

The extraction parameters of a cross-section are saved to the **Measurement** tab of its property sheet. They can be edited post creation by selecting the cross-section in the Tree View, choosing *Edit > Object Properties*, opening the tab, and making other specifications.

Probe the geometry

The geometry for a profile on the part can be obtained by probing the measured component of a cross-section. The probed points are projected onto the plane defined by the measurement zone of the cross-section. Subsequently, the projected points are joined by a polyline to define the measured component of the cross-section.

Objective: Finish inspecting the profile for an existing cross-section by probing the corresponding part geometry to obtain the measured component of the cross-section.

Required: The project contains a Reference object and a cross-section with a nominal component. The probing device is aligned to the Reference object.

Steps

1. In the Tree View, select the cross-section.
2. Choose *Measure > Cross-Sections > Probe Measured*.



The Probe Measured Cross-Section Component dialog box opens.

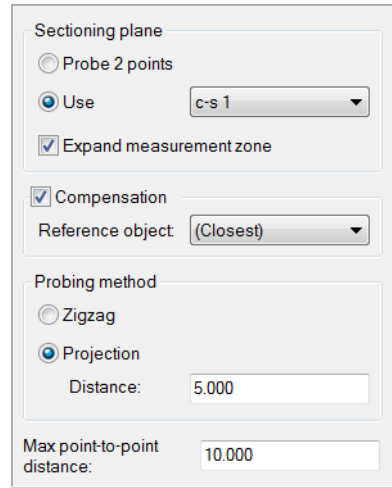
- Under **Sectioning plane**, click **Use** and choose a cross-section from the list.

*By default, the probe **Compensation** is calculated using the closest Reference object.*

- Under **Probing method**, select **Projection**.

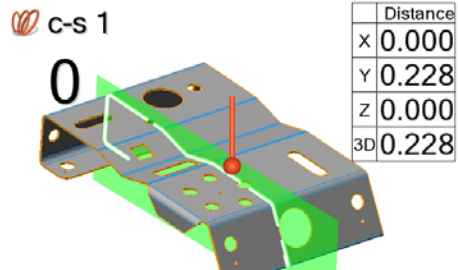
*All points acquired within the specified projection **Distance** of the plane are used to create the measured cross-section component.*

*The **Max point-to-point distance** determines how close the probed points must be spaced to be used to generate the measured cross-section.*



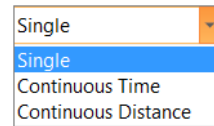
- Click **Probe**.

Use the visual and audio guidance to move the probe to the cross-section location. Once within the cross-section projection distance, probing is allowed.



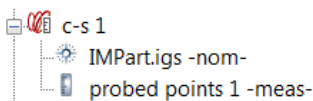
Tip

Use the **Continuous Time** or **Continuous Distance** probing mode to facilitate the probing.

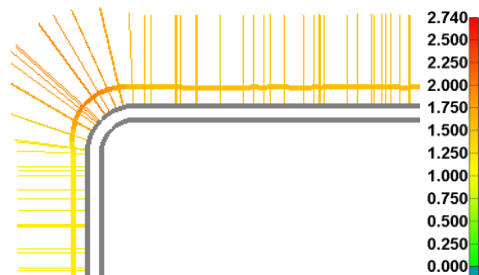


Results

The measured component is added to the cross-section. The measured status icon indicates its measured status.

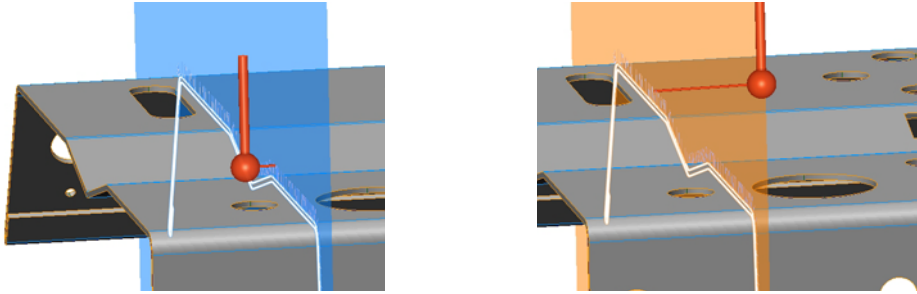


The cross-section is displayed in the 3D scene along with a color scale showing the deviations.



Notes

- Probing can also be done without an existing cross-section by using the **Probe 2 points** method to define the location of the sectioning plane.
- The **Zig-zag** probing method is another way to probe the geometry of the part. Each probed point must be captured on alternating sides of the sectioning plane. The plane changes color from blue to orange to indicate which side of the plane the probe is currently on. The intersection of the sectioning plane with the line connecting the currently probed point and the last probed point determines the acquired point.



- Additional probed points can be added to the probed measured component of a cross-section to better capture the part geometry. Select the cross-section in the Tree View and choose *Measure > Cross-Sections > Add Probed Data Points*.



Creating offset cross-sections

Occasionally, a profile must be inspected along offset parallel segments along the part. An offset cross-section is a measurement object that is useful for measuring deviations from the nominal contour located on different parallel slicing planes. The contour of the part can be obtained by extracting it from an aligned Data object.



Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Start inspecting the form of a part along discontinuous cutting planes by creating an offset cross-section, thereby obtaining its nominal component.

Required: The project contains a Reference object. In the example presented below, two circle primitives are in the project to position each segment of the offset cross-section.

If the project contains a Data object, it should be aligned to the Reference object.

Steps

1. Choose *Measure > Cross-Sections > Create > Offset Cross-Sections*.

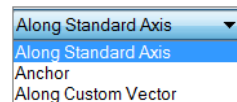
The Create Offset Cross-Sections dialog box opens.



2. Choose a creation **Method**.

The creation of cross-sections using the most frequently used methods follows.

All of the possible creation methods are shown to the right.



Along Standard Axis

This method allows creating an offset cross-section as discontinuous cross-section segments perpendicular to a standard system axis, such as the X axis. It is recommended to have a properly oriented coordinate system to obtain the anticipated results.

1. In the **Method** list, select **Along Standard Axis**.

2. Under **Axis**, select **X**, **Y**, or **Z**.

The cross-section segments will be positioned along and perpendicular to the axis.

3. Under **Position**, click **Pick Primitives**.

An interactive picking mode is launched.

Two circle primitives are already in the project.

4. In the 3D scene, click the primitive by which the first segment of the cross-section will pass.

The cross-section offset passes by the center of the picked primitive.

5. In the 3D scene, click the second primitive by which the second segment passes.

The new cross-section segment is parallel to the other segments and does not overlap another segment.

Repeat as many times as necessary.

6. Right-click to exit the mode.

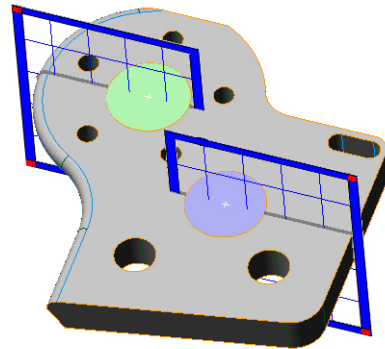
7. To resize the measurement zone to include only the area of interest, drag the blue borders of the desired measurement zone.

The other measurement zones resize themselves to avoid any overlap of segments.

8. If other segments are needed, click **Add Step**.

The measurement zone of the preceding segment is displayed.

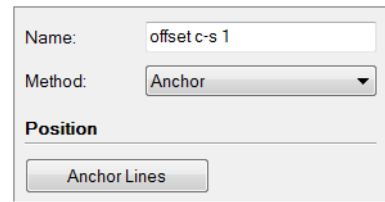
9. Click **Confirm** to create the offset cross-section.



Anchor

This method allows creating an offset cross-section by anchoring discontinuous cross-section segments in the 3D scene.

1. In the **Method** list, select **Anchor**.
2. Under **Position**, click **Anchor Lines**.
An interactive anchoring mode is launched.
3. In the 3D scene, click a start and end point for the first segment of the cross-section.

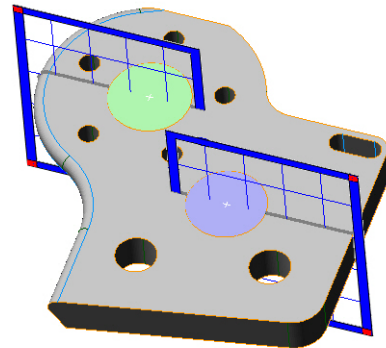


4. Click again to define the end point for the second segment of the cross-section.

The new cross-section segment is parallel to the other segments and does not overlap another segment.

The measurement zone of the preceding segment is displayed.

Repeat as many times as necessary.



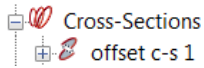
5. Right-click to finish.
6. To resize the measurement zone to include only the area of interest, drag the blue borders of the desired measurement zone.

The measurement zones of other segments resize themselves as consecutive segments never overlap.

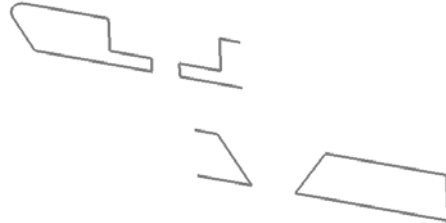
7. Click **Confirm**.

Results

The new offset cross-section, with a nominal component, is added to the Tree View under the Cross-Sections branch.



The offset cross-section is displayed in the 3D scene.



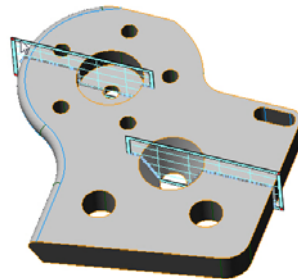
Notes

To adjust the measurement zones of an offset cross-section post creation:

1. Select the cross-section in the Tree View.
2. Choose *Measure > Cross-Sections > Edit Measurement Zones*.
3. The cross-section extent of each segment is displayed in the 3D scene and can be edited by dragging the borders.

The size of the measurement zone of neighboring segments is automatically adjusted to avoid segment overlap.

4. Press ESC to exit the editing mode.



Extracting the geometry on the part

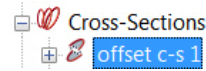
When there is a Data object available in the project at cross-section creation, the measured component is extracted automatically. Otherwise, when the Data object becomes available and is aligned, a manual extraction operation must be performed using the offset cross-section.

Objective: Finish inspecting the profile for an existing offset cross-section by extracting the corresponding profile geometry on the part to obtain the measured component of the cross-section.

Required: The project contains a Reference object, an aligned Data object, and an offset cross-section with a nominal component.

Steps

1. Select an offset cross-section in the Tree View.

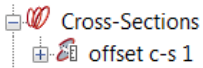


2. Choose *Measure > Extract Measured*.

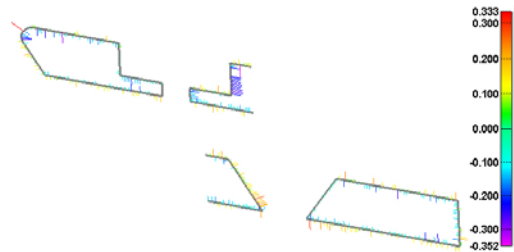


Results

The measured component is added to the offset cross-section. The measured status icon indicates its measured status.



The offset cross-section is displayed in the 3D scene along with a color scale showing nominal-to-measured deviations.



Notes

The extraction parameters of a cross-section are saved to the **Measurement** tab of its property sheet. They can be edited post creation by selecting the cross-section in the Tree View, choosing *Edit > Object Properties*, opening the tab, and making other specifications.

Comparison points



Surface comparison points



Trimmed edge comparison points



Hemmed edge comparison points

Measuring specific locations on a part using comparison points

A comparison point contains an exact coordinate, usually located on a CAD object, at which the data deviation is to be measured. These coordinates can be entered numerically, or generated by anchoring, sampling the CAD object, imported from a text file, or created by way of several other methods. Comparison points can be positioned on a part surface or curve, or along cross-sections. Surface comparison points are the most common.

Introducing comparison points

There are five types of comparison points, each one tailored to calculate deviations in a specific context. The following sections explain the types of comparison points that are most frequently used.

Comparison points are available on the *Measure > Comparison Points* menu or on the **Comparison Points** toolbar.

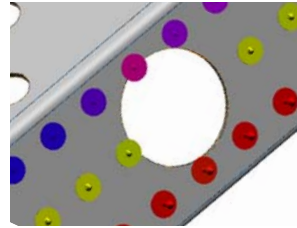


Types of comparison points

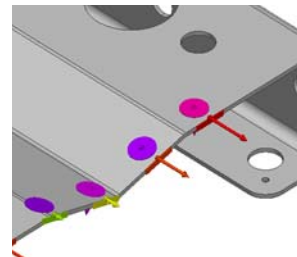
Each type of comparison point is identified by a specific icon.



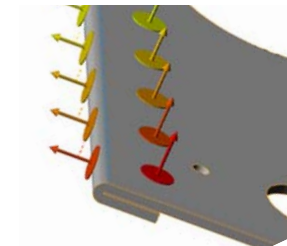
Surface comparison points



Trimmed edge comparison points

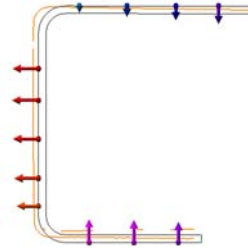


Hemmed edge comparison points





Cross-section comparison points



Comparison point creation and measurement methods

A comparison point is a measurement object that contains nominal and measured information in order to obtain results.

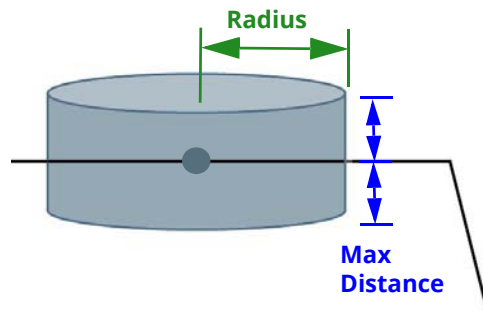
- The nominal component contains the exact coordinate at which the deviation is measured. It is created using different creation methods. The sections that follow explain the most frequently used methods.
- The measured component is created by measuring the part, either by extracting the measured value using predefined parameters, or by probing. By default, the measured component of a comparison point is extracted from available Data objects. If not automatically extracted, comparison points can be measured later. The Extract and Probe measurement methods are presented in the sections that follow.

Understanding key parameters

Radius

Each comparison point has a measurement zone, cylindrical by default, within which measurements will be collected. The measurement zone has a radius, which is the radius of the cylinder, and a maximum distance, which is the height of the cylinder.

The radius is specified during comparison point creation, and can be edited in the properties for that object. The **Max distance**, 4 mm by default, can also be edited in the object properties, along with other filtering criteria and tolerances.



Note that the measurement zone is different for trimmed edge comparison points. In this case, the radius of the measurement zone is the height of the cylinder, while the maximum distance is the radius of the cylinder.

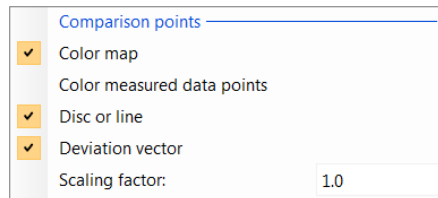
Display options

A number of display options are available to modify the way comparison points are represented.

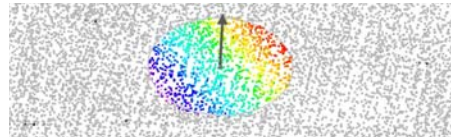
They are offered in the Object Display Options.



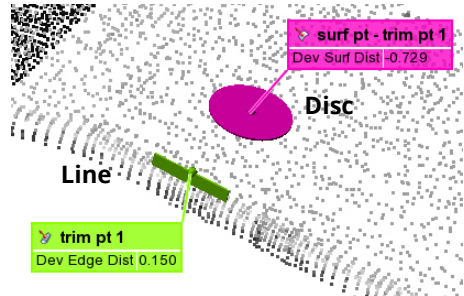
Under **Comparison points**, several display options are offered.



- Select **Color map**, then **Color measured data points** to display the deviation values of every data point used to calculate the measured comparison point. Note that the color map display can be customized. For information, see [Changing the color map display](#) on page 201.



- By default, **Disc or line** is selected to allow specifying the size of the displayed reference disc or line. If cleared, only the center coordinate is displayed.



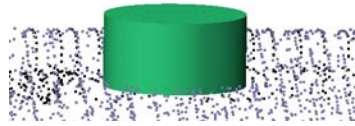
- Select **Deviation vector** to view the direction of deviation. Increasing the **Scaling factor** increases the length of the deviation vector for better visibility.



In the PolyWorks | Inspector Options dialog box (choose *Tools > Options*, click **Display**, then **Comparison Points**):



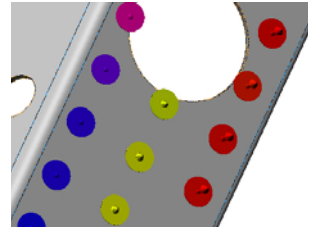
- Select **Measurement zone** to display in 3D the measurement zone.



Creating surface comparison points to inspect the surface of a part

A surface comparison point is a measurement object. It is created to obtain the deviation on a surface, at specific coordinates. The geometry of the surface can be obtained by extracting it from an aligned Data object, or by probing the measured component of the surface comparison point.

If creating by probing, the nominal component is automatically extracted on the closest Reference object. The digitizing device must be aligned to the Reference object.



Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Create surface comparison points to capture the nominal component.

Required: The project contains a Reference object. If the project also contains a Data object, the Data object must be aligned to the Reference object.

Steps

1. Choose *Measure > Comparison Points > Create > Surface Points*.

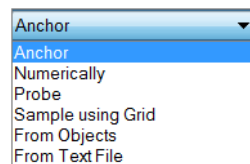
The Create Surface Comparison Points dialog box opens.



2. Choose a **Method** among the following.

The creation of surface comparison points using the most common methods follows.

All creation methods are shown to the right.



Anchor

This method allows creating surface comparison points by anchoring on surfaces.

1. In the **Method** list, select **Anchor**.
2. Enter the **Radius** of the measurement zone for each comparison point. See [Understanding key parameters](#) on page 401.
3. If required, select **Link to object** to bind surface comparison points to a Reference object or a surface feature. Then, specify whether the comparison points are linked to the closest object/feature or to a specific one.
4. Click **Anchor**.
5. In the 3D scene, click a location on the model to place the comparison point.
6. Right-click to exit the anchor mode.

Creation method

Method: **Anchor**

Parameters

Radius: 2.500

Link to object Reference Object

(Closest)

Numerically

This method allows creating surface comparison points by entering values for certain geometrical properties.

1. In the **Method** list, select **Numerically**.
2. Under **Point**, enter the coordinates of the point to create.

If one coordinate is missing, select the **Project** option to get the missing coordinate. The coordinate is calculated by projecting the point onto the Reference surface along the axis corresponding to the missing coordinate.

3. Specify the **Surface normal** information of the point.

By default, **Automatic** is selected. It extracts the normal information from the closest point on the surface of the reference object. To specify values manually, the option must be disabled.

4. Enter the **Radius** of the measurement zone for each comparison point. See [Understanding key parameters](#) on page 401.
5. If required, select **Link to object** to bind surface comparison points to a Reference object or a surface feature. Then, specify whether the comparison points are linked to the closest object/feature or to a specific one.
6. Click **Create**.

Creation method

Method: **Numerically**

Parameters

Point

x: 0.000

y: 0.000

z: 0.000

Project Z

Surface normal

i: 0.000

j: 0.000

k: 1.000

Automatic

Radius: 2.500

Link to object Reference Object

(Closest)

Sample using Grid

This method allows creating uniformly spaced surface comparison points by sampling a virtual grid that covers the selected or visible surfaces of Reference objects.

1. In the **Method** list, select **Sample using Grid**.
2. Specify the **Sampling step** that determines the comparison point spacing.
3. Specify the **Projection axis**, which means the direction in which the comparison points will be projected on the model.
4. Enter the **Radius** of the measurement zone for each comparison point. See [Understanding key parameters](#) on page 401.
5. Under **Link to object**, specify whether surface comparison points will be bound to a Reference object or a surface feature. Then, specify whether the comparison points are linked to the closest object/feature or to a specific one.
6. Under **Considered reference surfaces**, select whether the grid is to be sampled on all of the visible surfaces of the Reference object or on the selected surfaces only.
7. Click **Preview**.

A preview of the comparison point layout is presented in the 3D scene.

8. Click **Confirm**.

The screenshot shows the 'Creation method' dialog box with the following settings:

- Method:** Sample using Grid
- Parameters:**
 - Sampling step:** 10.000
 - Projection axis:** -Z
 - Radius:** 2.500
 - Link to object:** Reference Object
 - (Closest)**
 - Considered reference surfaces:**
 - Visible
 - Selected

From Text File

This method allows creating surface comparison points from values contained in text files. If the new comparison points are not located on a Reference object, they can be projected onto the nearest linked Reference object.

1. In the **Method** list, select **From Text File**.
2. Enter the **Radius** of the measurement zone for each comparison point. See [Understanding key parameters](#) on page 401.
3. If required, select **Link to object** to bind surface comparison points to a Reference object or a surface feature. Then, specify whether the comparison points are linked to the closest object/feature or to a specific one.
4. Click **Browse**.

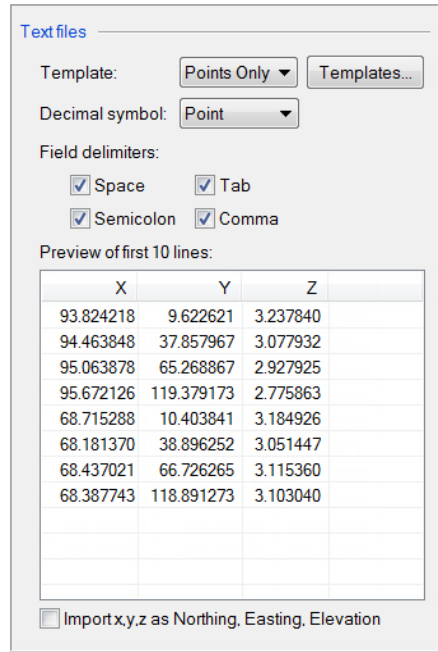
The screenshot shows the 'Creation method' dialog box with the following settings:

- Method:** From Text File
- Parameters:**
 - Radius:** 2.500
 - Link to object:** Reference Object
 - (Closest)**

- Navigate to a text file containing the x, y, z coordinates of the desired measurement locations.

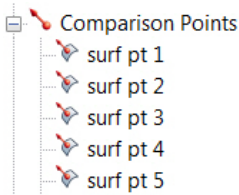
The Import Parameters dialog box opens.

- In the file browser, select a **Template** that corresponds to the file structure. A preview of the file is presented giving feedback if the chosen template is incorrect.
- Click **OK**.

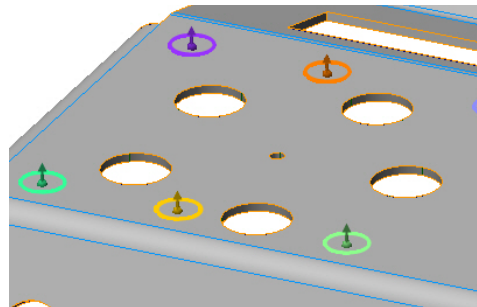


Results

The new comparison points, with a nominal component, are added to the Tree View under the Comparison Points branch.



The comparison points are displayed in the 3D scene.



Measuring the geometry on the part

For an existing surface comparison point, the corresponding geometry on the part can be obtained by extraction on an aligned Data object or by probing a physical part. If proceeding by probing, the digitizing device should be aligned to the Reference object. In PolyWorks | Inspector, the geometry obtained on the part is called the *measured component* of the measurement object.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Extract the geometry

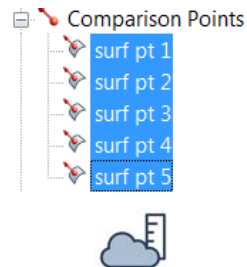
When creating a surface comparison point and a Data object is available in the project, the measured component is extracted automatically. Otherwise, when the Data object becomes available and is aligned, a manual extraction operation must be performed using the surface comparison point.

Objective: Extract the corresponding geometry on the part to obtain the measured component of the comparison point.

Required: The project contains a Reference object, an aligned Data object, and a surface comparison point with a nominal component.

Steps

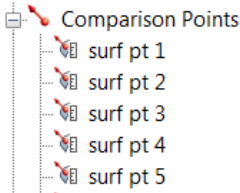
1. Select the surface comparison point in the Tree View.



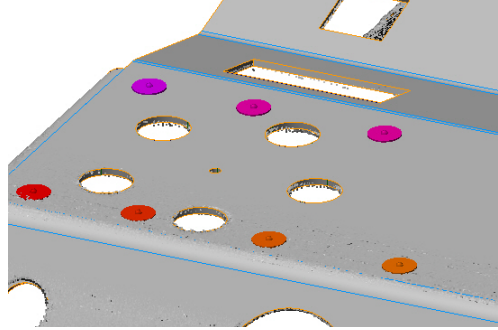
2. Choose *Measure > Extract Measured*.

Results

A measured status icon indicates the measured status of the surface comparison points.



The surface comparison points are displayed in the 3D scene with a solid measurement zone. The color of the comparison points corresponds to the defined color scale.



Probe the geometry

The geometry of the surface on the part can be obtained by probing the measured component of a surface comparison point.

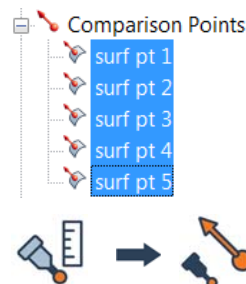
Objective: Probe the corresponding geometry on the part to obtain the measured component of the comparison point.

Required: The project contains a Reference object and a surface comparison point with a nominal component. A probing device is available and ready to use. The probing device is aligned to the Reference object.

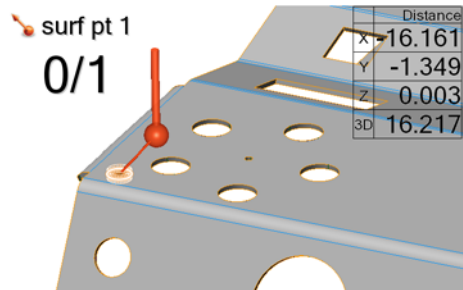
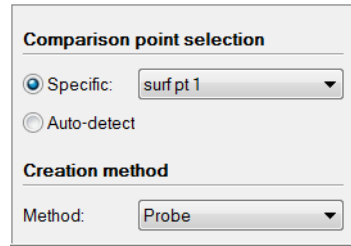
Steps

1. Select the surface comparison points to probe in the Tree View.
2. Choose *Measure > Comparison Points > Define Measured*.

The Define Measured Comparison Point Components dialog box opens.



3. Specify the probing mode:
 - **Specific:** Allows probing the selected comparison points following the Tree View order.
 - **Auto-detect:** Allows probing the comparison point closest to the probing device. Only the selected comparison points will be detected.
4. Click **Probe**.



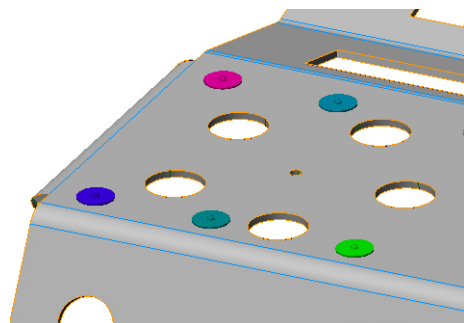
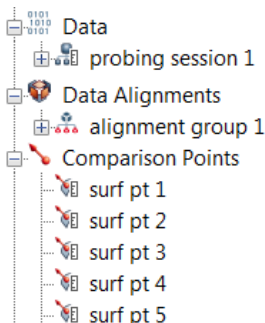
Tips

When probing comparison points, visual and audio feedback are provided. If the probe is out of the measurement zone, the display color of the comparison point disc is red and the farther the probe is from the comparison point the slower the sound is. Moving towards the comparison point will accelerate the sound. Once the measurement zone is reached, the display color of the comparison point disc will change to green and the sound will change to a higher pitch which eventually stops indicating the location of the probing device is good enough to acquire data.

Results

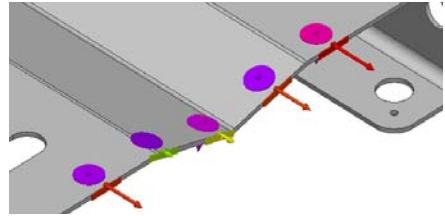
A probing session Data object is added to the Tree View and a measured status icon indicates the measured status of the surface comparison points.

The surface comparison points are displayed in the 3D scene with a solid measurement zone. The color of the comparison points corresponds to the defined color scale.



Creating trimmed edge comparison points to inspect sharp edges of a part

A trimmed edge comparison point is a measurement object. It is created to obtain the deviation on an edge, at specific coordinates. The geometry of the part can be obtained by extracting it from an aligned Data object, or by probing the measured component of the trimmed edge comparison point.



If creating by probing, the nominal component is automatically extracted on the closest Reference object. The digitizing device must be aligned to the Reference object.

Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Create trimmed edge comparison points to capture the nominal component.

Required: The project contains a Reference object. If the project also contains a Data object, the Data object must be aligned to the Reference object.

Steps

1. Choose *Measure > Comparison Points > Create > Trimmed Edge Points*.

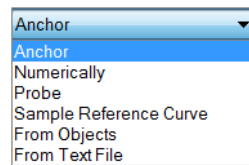
The Create Trimmed Edge Comparison Points dialog box opens.



2. Choose a **Method** among the following.

The creation of trimmed edge comparison points using the most common methods follows.

All creation methods are shown to the right.



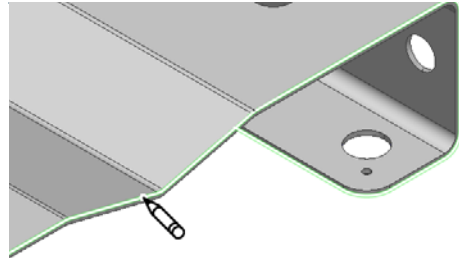
Anchor

This method allows creating trimmed edge comparison points by anchoring on a Reference object curve.

1. In the **Method** list, select **Anchor**.
2. Specify the **Radius** of the measurement zone for each comparison point. See [Understanding key parameters](#) on page 401.
3. Click **Anchor**.

Creation method	
Method:	Anchor
Parameters	
Radius:	2.500

4. In the 3D scene, click an edge curve to anchor the trimmed edge comparison points.
5. Right-click to exit the anchor mode.



Sample Reference Curve

This method allows creating uniformly spaced trimmed edge comparison points by sampling curves of Reference objects using a step.

1. In the **Method** list, select **Sample Reference Curve**.
2. Specify the **Sampling step**. This parameter sets the distance between comparison points along the reference curve.
3. Specify the **Radius** of the measurement zone for each comparison point. See [Understanding key parameters](#) on page 401.
4. Select the **Range**:
 - **Whole Curve:** This option, selected by default, enables creating comparison points along the whole curve.
 - **Partial Curve:** This option enables creating comparison points along a portion of the curve.
5. In the 3D scene, click the reference curve to select it. A preview is displayed to show where the comparison points will be located.

If the **Partial Curve** range option is selected, two clicks are required: the start and the end point of the range.
6. Click **Confirm**.

Creation method

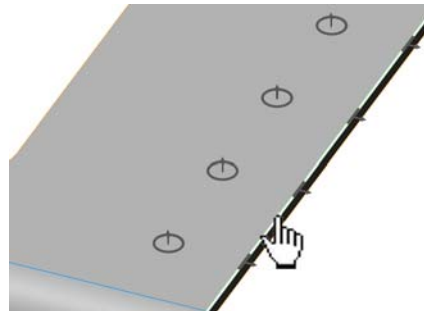
Method: Sample Reference Curve

Parameters

Sampling step:

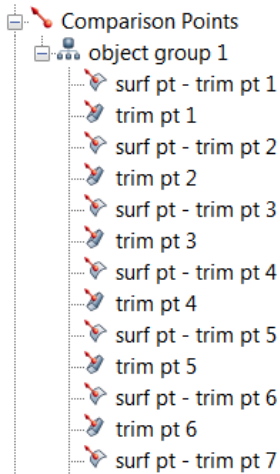
Radius:

Range:

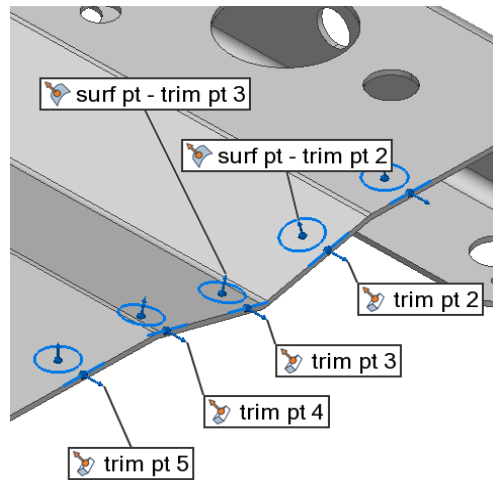


Results

The new comparison points, with a nominal component, are added to the Tree View under the Comparison Points branch.

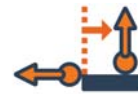


The comparison points are displayed in the 3D scene.



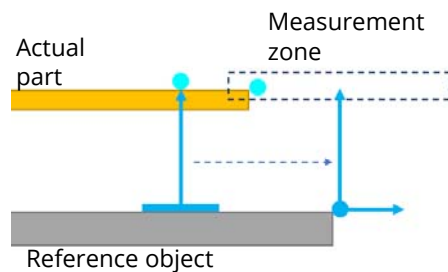
Notes

By default, when creating trimmed edge comparison points, a back point is created. The back point is a surface comparison point offset from the part edge.



The back point locates the actual part surface before measuring the edge, ensuring that deformation of the actual part shape does not prevent edge inspection.

If back points are not desired, the option can be disabled under the **More** section of the creation dialog box.



Measuring the geometry on the part

For an existing trimmed edge comparison point, the corresponding geometry on the part can be obtained by extraction on an aligned Data object or by probing a physical part. If proceeding by probing, the digitizing device should be aligned to the Reference object. In PolyWorks | Inspector, the geometry obtained on the part is called the *measured component* of the measurement object.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Extract the geometry

When there is a Data object available in the project at trimmed edge comparison point creation, the measured component is extracted automatically. Otherwise, when the Data object becomes available and is aligned, a manual extraction operation must be performed using the trimmed edge comparison point.

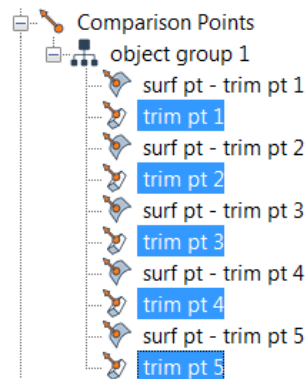
Note that when a trimmed edge comparison point is extracted, the corresponding back point is also automatically extracted.

Objective: Extract the corresponding geometry on the part to obtain the measured component of the comparison point.

Required: The project contains a Reference object, an aligned Data object, and a trimmed edge comparison point with a nominal component.

Steps

1. Select the trimmed edge comparison point in the Tree View.



2. Choose *Measure > Extract Measured*.



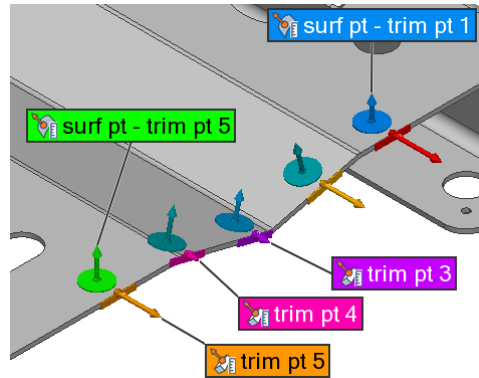
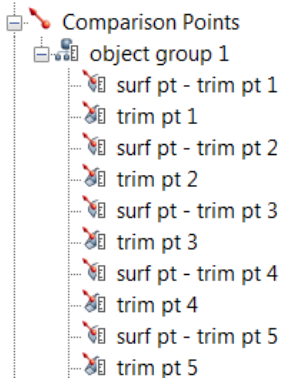
Tips

If a trimmed edge comparison point cannot be extracted, make sure that there is enough data to perform the extraction. Also, modifying the extraction parameters could make a difference. In the Tree View, select the trimmed edge comparison point, choose *Edit > Object Properties*, then click the **Measurement** tab to access the extraction parameters.

Results

A measured status icon indicates the measured status of the trimmed edge comparison points and their corresponding back points.

The trimmed edge comparison points and their back points are displayed in the 3D scene with a solid measurement zone. The color of the comparison points corresponds to the defined color scale.



Probe the geometry

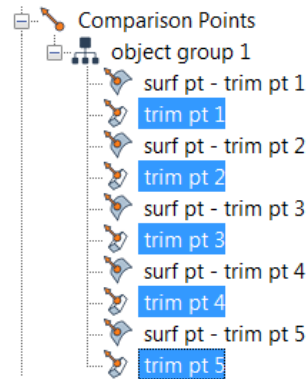
The geometry of the sharp edge on the part can be obtained by probing the measured component of a trimmed edge comparison point.

Objective: Probe the corresponding geometry on the part to obtain the measured component of the comparison point.

Required: The project contains a Reference object as well as a trimmed edge comparison point with a nominal component and its back point. The probing device is aligned to the Reference object.

Steps

1. Select the trimmed edge comparison points to probe in the Tree View.



2. Choose *Measure > Comparison Points > Define Measured*.

Alternatively, right-click the selected comparison points in the Tree View and choose *Probe Measured*.

The Defined Measured Comparison Point Components dialog box opens.

3. Specify the probing mode:

- **Specific:** Allows probing the selected comparison points following the Tree View order.
- **Auto-detect:** Allows probing the comparison point closest to the probing device. Only the selected comparison points will be detected.

Regardless of the probing mode, you will always be prompted to probe the back point first and then the trimmed edge comparison point.

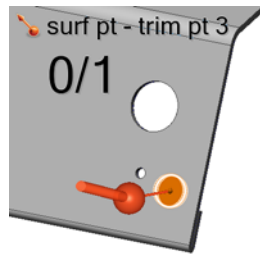
4. In the **Method** list, select **Probe**.



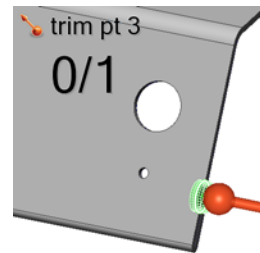
Comparison point selection	
<input checked="" type="radio"/> Specific:	surf pt - trim pt 1
<input type="radio"/> Auto-detect	
Creation method	
Method:	Probe

5. Click **Probe**.

Two points must be probed: first probe the back point, then the corresponding trimmed edge comparison point.



	Distance
X	1.448
Y	6.607
Z	2.523
3D	7.219



	Distance
X	-0.853
Y	0.005
Z	0.492
3D	0.985

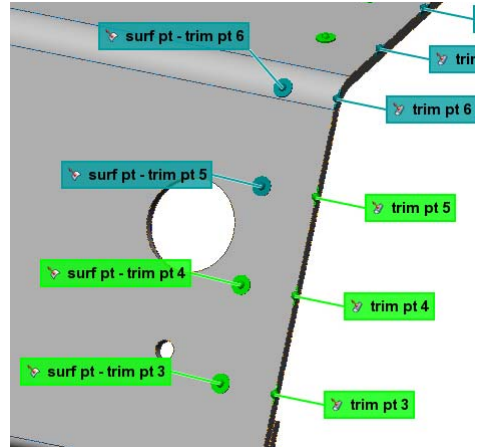
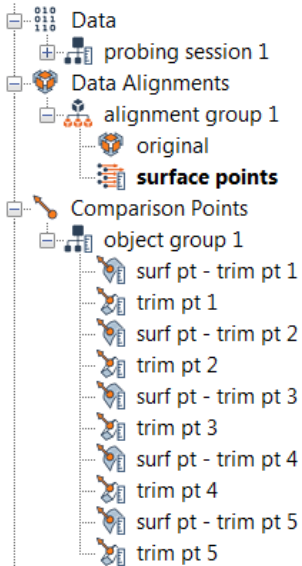
Tips

When probing comparison points, visual and audio feedback are provided. If the probe is out of the measurement zone, the display color of the comparison point disc is red and the farther the probe is from the comparison point, the slower the sound is. Moving towards the comparison point will accelerate the sound. Once the measurement zone is reached, the display color of the comparison point disc will change to green and the sound will change to a higher pitch which eventually stops, indicating the location of the probing device is good enough to acquire data.

Results

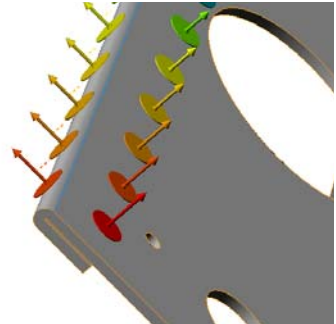
A probing session Data object is added to the Tree View and a measured status icon indicates the measured status of the trimmed edge comparison points and their corresponding back points.

The trimmed edge comparison points and their backpoints are displayed in the 3D scene with a solid measurement zone. The color of the comparison points corresponds to the defined color scale.



Creating hemmed edge comparison points to inspect the hemmed edges of a sheet metal part

A hemmed edge is typically found in sheet metal parts as the result of a hemming process which consists in folding an edge until it is flush to itself. A hemmed edge comparison point is a measurement object. It is created to obtain the deviation on a hemmed edge, at specific coordinates. The geometry of the part can be obtained by extracting it from an aligned Data object, or by probing the measured component of the hemmed edge comparison point.



Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Create hemmed edge comparison points to capture the nominal component.

Required: The project contains a Reference object. If the project also contains a Data object, the Data object must be aligned to the Reference object.

Steps

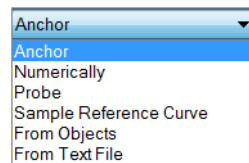
1. Choose *Measure > Comparison Points > Create > Hemmed Edge Points*.

The Create Hemmed Edge Comparison Points dialog box opens.

2. Choose a **Method** among the following.

The creation of hemmed edge comparison points using the most common methods follows.

All creation methods are shown to the right.

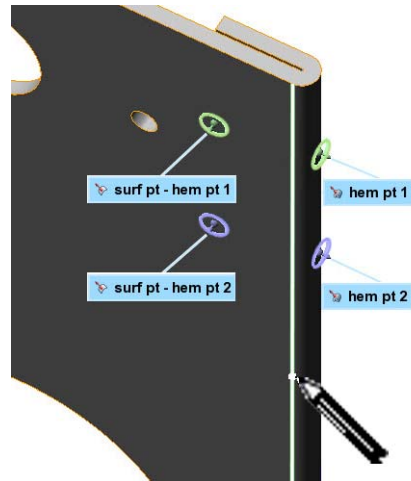


Anchor

This method allows creating hemmed edge comparison points by anchoring on a Reference object curve.

1. In the **Method** list, select **Anchor**.
2. Specify the **Radius** of the measurement zone for each comparison point. See [Understanding key parameters](#) on page 401.
3. Specify how the **Hemmed edge radius** is measured, either by extracting the radius from the Reference object, or by manually entering it in the **Custom** field.
4. Click **Anchor**.
5. In the 3D scene, click the tangency curves of the hemmed edge to anchor the hemmed edge comparison points.
6. Right-click to exit the anchor mode.

Creation method	
Method:	Anchor
Parameters	
Radius:	2.500
Hemmed edge radius:	
<input checked="" type="radio"/>	Extract from reference object
<input type="radio"/>	Custom: 2.000



Sample Reference Curve

This method allows creating uniformly spaced hemmed edge comparison points by sampling curves of Reference objects using a step.

1. In the **Method** list, select **Sample Reference Curve**.
2. Specify the **Sampling step**. This parameter sets the distance between comparison points along the reference curve.
3. Specify the **Radius** of the measurement zone for each comparison point. See [Understanding key parameters](#) on page 401.
4. Specify how the **Hemmed edge radius** is measured, either by extracting the radius from the Reference object, or by manually entering it in the **Custom** field.
5. Select the **Range**:
 - **Whole Curve**: this option, selected by default, enables creating comparison points along the whole curve.
 - **Partial Curve**: this option enables creating comparison points along a portion of the curve.
6. In the 3D scene, click the reference curve to select it. A preview is displayed to show where the comparison points will be located.
7. Click **Confirm**.

Creation method

Method: Sample Reference Curve

Parameters



Sampling step: 10.000

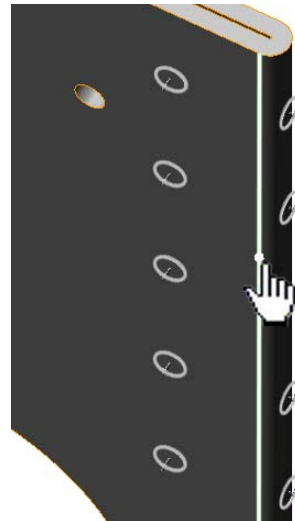
Radius: 2.500

Hemmed edge radius:

Extract from reference object

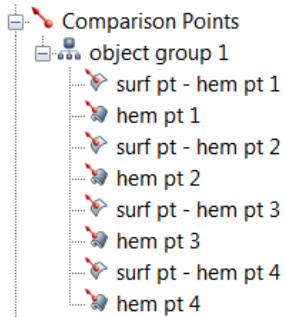
Custom: 2.000

Range:  

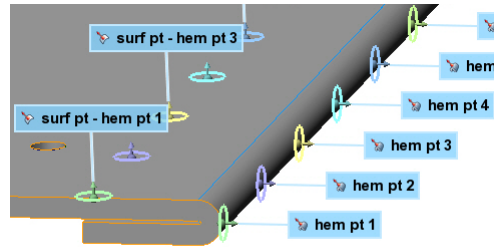


Results

The new comparison points, with a nominal component, are added to the Tree View under the Comparison Points branch.



The comparison points are displayed in the 3D scene.



Notes

By default, when creating hemmed edge comparison points, a back point is created. The back point is a surface comparison point offset from the part edge.



The back point locates the actual part surface before extracting the edge measurement, ensuring that deformation of the actual part shape does not prevent edge inspection.

If back points are not desired, the option can be disabled under the **More** section of the creation dialog box.

Measuring the geometry on the part

For an existing hemmed edge comparison point, the corresponding geometry on the part can be obtained by extraction on an aligned Data object or by probing a physical part. If proceeding by probing, the digitizing device should be aligned to the Reference object. In PolyWorks | Inspector, the geometry obtained on the part is called the *measured component* of the measurement object.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Extract the geometry

When there is a Data object available in the project at hemmed edge comparison point creation, the measured component is extracted automatically. Otherwise, when the Data object becomes available and is aligned, a manual extraction operation must be performed using the hemmed edge comparison point.

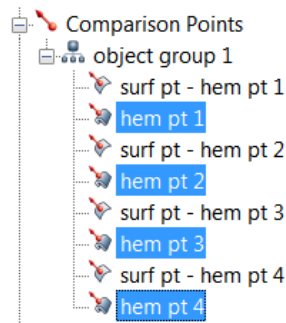
Note that when a hemmed edge comparison point is extracted, the corresponding back point is also automatically extracted.

Objective: Extract the corresponding geometry on the part to obtain the measured component of the comparison point.

Required: The project contains a Reference object, an aligned Data object, and a hemmed edge comparison point with a nominal component.

Steps

1. Select the hemmed edge comparison points in the Tree View.



2. Choose *Measure > Extract Measured*.



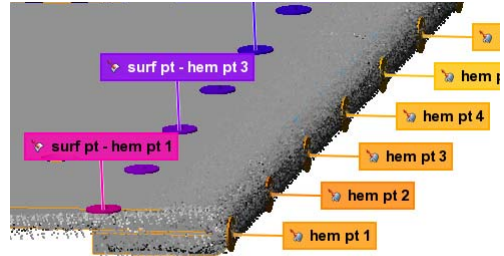
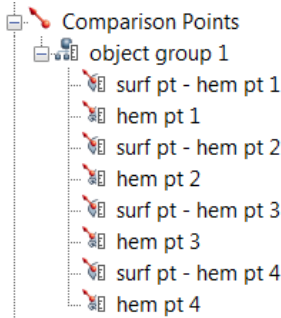
Tips

If a hemmed edge comparison point cannot be extracted, make sure that there is enough data to perform the extraction. Also, modifying the extraction parameters could make a difference. In the Tree View, select the hemmed edge comparison point and choose *Edit > Object Properties*, then click the **Measurement** tab to access the extraction parameters.

Results

A measured status icon indicates the measured status of the hemmed edge comparison points and their corresponding back points.

The hemmed edge comparison points and their back points are displayed in the 3D scene with a solid measurement zone. The color of the comparison points corresponds to the defined color scale.



Probe the geometry

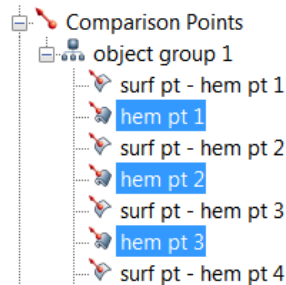
The geometry of the hemmed edges on the sheet metal part can be obtained by probing the measured component of a hemmed edge comparison point.

Objective: Probe the corresponding geometry on the part to obtain the measured component of the comparison point.

Required: The project contains a Reference object as well as a hemmed edge comparison point with a nominal component and its back point. The probing device is aligned to the Reference object.

Steps

1. Select the hemmed edge comparison points in the Tree View.



- Choose *Measure > Comparison Points > Define Measured*.

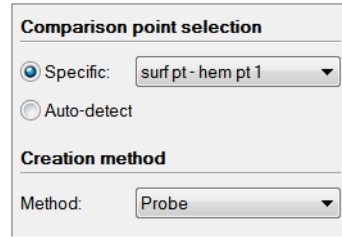


Alternatively, right-click the selected comparison points in the Tree View and choose *Probe Measured*.

The Defined Measured Comparison Point Components dialog box opens.

- Specify the probing mode:

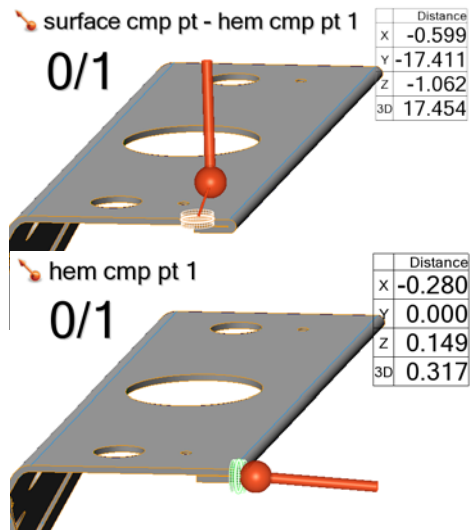
- **Specific:** Allows probing the selected comparison points following the Tree View order.
- **Auto-detect: Allows** probing the comparison point closest to the probing device. Only the selected comparison points will be detected.



Regardless of the probing mode, you will always be prompted to probe the back point first and then the hemmed edge comparison point.

- Click **Probe**.

Two points must be probed: first probe the back point, then the corresponding hemmed edge comparison point.



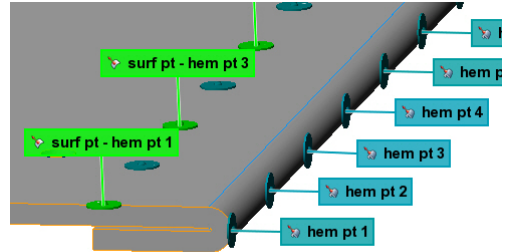
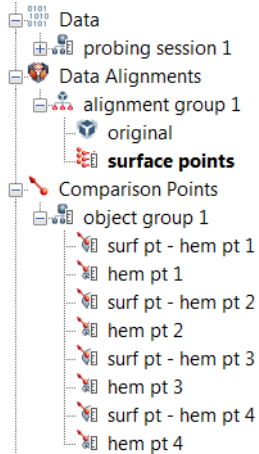
Tips

When probing comparison points, visual and audio feedback are provided. If the probe is out of the measurement zone, the display color of the comparison point disc is red and the farther the probe is from the comparison point the slower the sound is. Moving towards the comparison point will accelerate the sound. Once the measurement zone is reached, the display color of the comparison point disc will change to green and the sound will change to a higher pitch, which eventually stops indicating the location of the probing device is good enough to acquire data.

Results

A probing session Data object is added to the Tree View and a measured status icon indicates the measured status of the hemmed edge comparison points and their corresponding back points.

The hemmed edge comparison points, and their backpoints, are displayed in the 3D scene with a solid measurement zone. The color of the comparison points corresponds to the defined color scale.



Gauges



Calipers



Profile gauges



Flush & gap gauges



Airfoil gauges

Measuring part dimensions with gauges

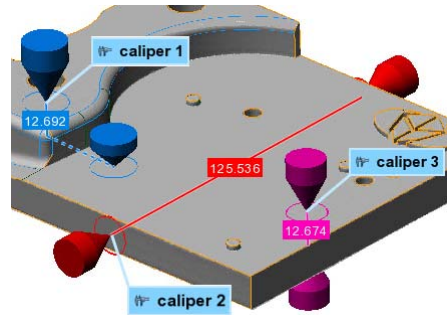
In PolyWorks|Inspector, common physical measurement tools such as calipers, radius gauges, and step gauges are simulated using Gauge objects. Gauges are used to measure the deviations of the part geometry with respect to the nominal geometry.

There are four types of gauges:



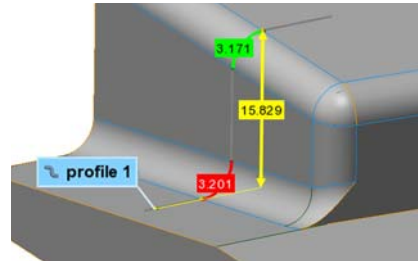
Caliper

A gauge that is similar to a physical caliper used to measure distances.



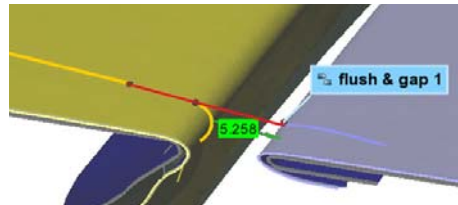
Profile gauge

Profile gauges are used to measure the radii, distances, or heights on the fillets or rounds of a part on a cross-sectional plane.



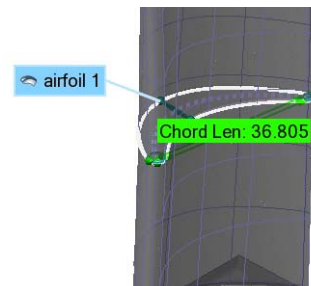
Flush & gap gauge

Flush & gap gauges are used to measure the flush and gap between parts of an assembly on a cross-sectional plane.



Airfoil gauge

Airfoil gauges are used to measure airfoil profiles on a cross-sectional plane.



Gauges, except for calipers, are measured on linked cross-sections which are automatically created on gauge creation. Calipers calculate distances either on a 3D surface or at locations on a cross-section.

Display options

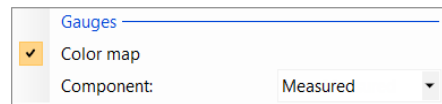
There are display options for the visualization of the results.

1. Click **Object Display Options**.



2. Under **Gauges**:

- Select **Color map** to display the deviation of the measured geometry to the nominal geometry. Note that the color map display can be customized. For information, see [Changing the color map display](#) on page 201.
- Select the component that will be displayed: **Nominal** or **Measured**.



Measuring object dimensions with calipers

A caliper is a measurement instrument widely used in the manufacturing industry to measure the distance between two opposite sides of an object by adjusting its tips to fit the points to be measured. The gap between the tips reports the distance between the sides directly, if the caliper has a built-in scale, or indirectly, if another measurement instrument must be used to measure the gap.

Introducing calipers

In PolyWorks | Inspector, the caliper tool is a software simulation of a physical caliper. It has similar behavior and applications. For example, it has two customizable tips.

Calipers are used to inspect distances using the nominal geometry and the geometry of the part in order to produce results determined by geometry controls. The nominal geometry and the part geometry are captured by intersecting the surface of aligned Reference and Data objects with the caliper tips.

Calipers are available on the *Measure > Gauges* menu or on the **Gauges** toolbar.



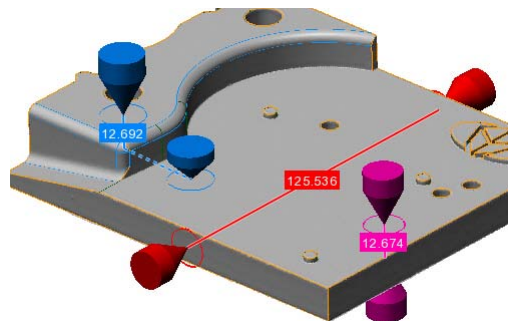
Caliper types

There are two types of calipers:



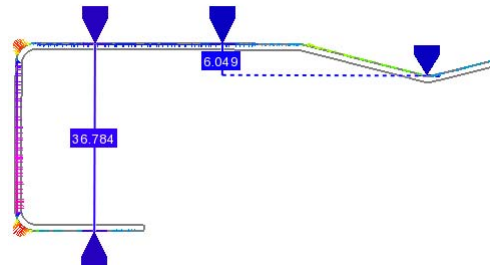
Standard caliper

Calculates distances on a 3D surface.



Cross-section caliper

Calculates distances on cross-sections.

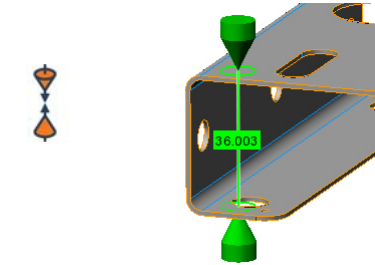


Axis types

There are two axis types for both types of calipers:

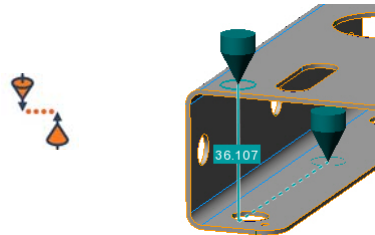
- **Single axis**

Allows creating calipers using one single axis for both endpoints.



- **Offset axes**

Allows creating calipers using one axis for each endpoint.

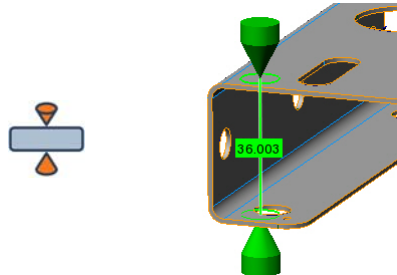


Caliper tip directions

Calipers can have one or both tips facing inward along the direction of measurement, outward along the direction of measurement, or be anchored.

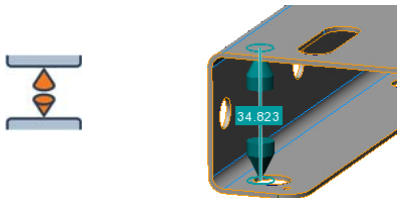
- **Inward**

An inward tip points towards the other tip.



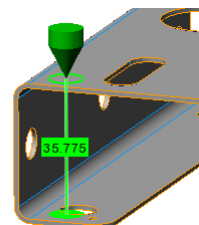
- **Outward**

An outward tip points away from the other tip.



- **Anchored**

An anchored tip is fixed and it does not move from where it is created.



Display options

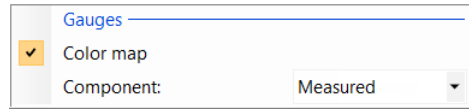
There are display options for the visualization of the results.

1. Click **Object Display Options**.



2. Under **Gauges**:

- Select **Color map** to display the deviation of the measured geometry to the nominal geometry. Note that the color map display can be customized. For information, see [Changing the color map display](#) on page 201.
- Select the component that will be displayed: **Nominal** or **Measured**.



Creation methodology

A standard dialog box with common and contextual parameters is used for the creation of calipers.

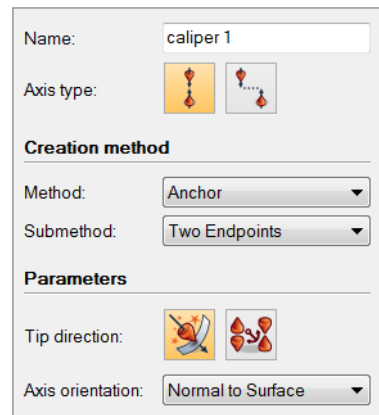
1. Choose *Measure > Gauges > Create > [any caliper type]*.



A creation dialog box opens.

2. Specify a **Name**, if desired.
3. In the **Axis type** list, click **Single Axis** or **Offset Axes**.
4. In the **Method** list, select a creation method.
5. In the **Submethod** list, select a creation submethod.

This list is not offered for some methods.

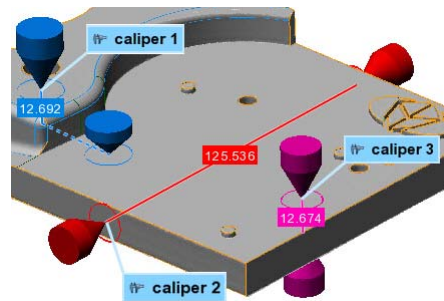


6. In the **Parameters** section, configure the caliper creation.
7. Click the appropriate button to launch the creation.

Creating standard calipers to measure distances on objects

It is possible to measure distances between surfaces using the standard caliper tool, which simulates a physical caliper. While standard calipers created using default settings may meet many needs, they are fully configurable in the creation dialog box or post creation using their property sheet.

A caliper is a measurement object. It is created to obtain a) the nominal geometry of the Reference object, and b) the geometry of the part by extracting it from an aligned Data object using an existing caliper. When both geometries have been obtained, geometry controls are applied to produce the desired results.



Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Start inspecting a distance on a part by creating a standard caliper to capture the nominal component.

Required: The project contains a Reference object.

Steps

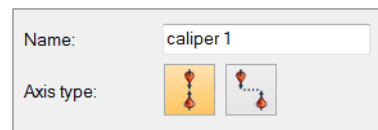
1. Choose *Measure > Gauges > Create > Standard Calipers*.

The Create Calipers dialog box opens.



2. Select the **Axis type**:

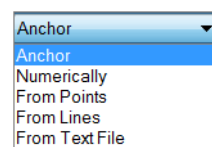
- **Single Axis** allows creating calipers that use one single axis for both endpoints.
- **Offset Axes** allows creating calipers using one axis for each endpoint.



3. Choose a **Method** from those that follow.

The creation of calipers using the most frequently used methods follows.

All of the possible creation methods are shown to the right.



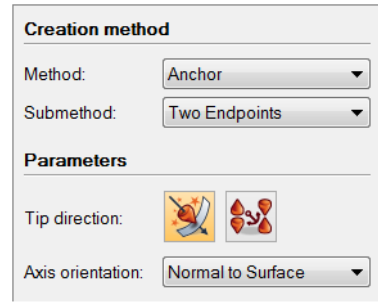
Anchor

This method allows anchoring caliper endpoints interactively.

1. In the **Method** list, select **Anchor**.
2. In the **Submethod** list, select **Two Endpoints**.

This allows creating a caliper by specifying the two endpoints.

If using offset axes, this is the only submethod.



3. Select a **Tip direction**:
 - **Automatic** specifies the tip direction to automatically point toward the surface where the endpoints are located.
 - **Custom** allows configuring the tip direction to face inward, outward, or be anchored in space.

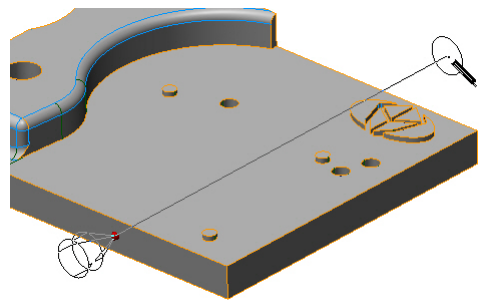
4. Specify the **Axis orientation**.

The caliper axis can be normal to the surface, follow a system axis, a custom vector, or be defined by the endpoints.

5. Click **Anchor**.

An interactive anchoring mode is launched:

- Press the SPACEBAR to interrupt the mode and rotate the model.
- Pick the endpoints of the caliper on the Reference object.
- Repeat if desired.
- Right-click to exit the mode.



Numerically

This method allows entering numerical values for the coordinates of the caliper endpoints.

1. In the **Method** list, select **Numerically**.
2. In the **Submethod** list, select **Two Endpoints**.

This allows creating a caliper by specifying the two endpoints. If using offset axes, this is the only submethod.

3. Select a **Tip direction**:

- **Automatic** specifies the tip direction to automatically point toward the surface where the endpoints are located.
- **Custom** allows configuring the tip direction to face inward, outward, or be anchored in space.

4. Under **1st endpoint** and **2nd endpoint**, enter the (x, y, z) coordinates of the endpoints.

5. If using offset axes, specify the **Axis orientation**.

The caliper axis can be normal to the surface, follow a system axis, a custom vector, or be defined by the endpoints.



6. Click **Create**.

Creation method

Method:

Submethod:

Parameters

Tip direction:  

1st endpoint		2nd endpoint	
x:	<input type="text" value="0.000"/>	x:	<input type="text" value="0.000"/>
y:	<input type="text" value="0.000"/>	y:	<input type="text" value="0.000"/>
z:	<input type="text" value="0.000"/>	z:	<input type="text" value="0.000"/>

From Text File

This method allows creating calipers by importing information from a text file. While the file must contain the coordinates of one or both endpoints, other information such as vectors, nominal lengths, radii, and names can also be imported.

1. In the **Method** list, select **From Text File**.

2. In the **Submethod** list, select **Two Endpoints**.

This allows creating a caliper by specifying the two endpoints. If using offset axes, this is the only submethod.

3. Select a **Tip direction**:

- **Automatic** specifies the tip direction to automatically point toward the surface where the endpoints are located.
- **Custom** allows configuring the tip direction to face inward, outward, or be anchored in space.

4. If offset axes are used, specify the **Axis orientation**.

The caliper axis can be normal to the surface, follow a system axis or a custom vector, or be defined by the endpoints.

5. Click **Browse**.

A file browser opens.

6. Navigate to a text file containing the (x, y, z) coordinates of the endpoints.

7. In the import dialog box, under **Text files**, select a **Template** that corresponds to the file structure.

A preview of the file is presented to verify that the chosen template is correct.

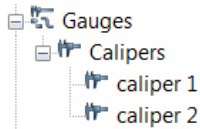
X	Y	Z
28.840626	25.400000	96.970459
61.032288	12.700000	98.412498
95.929634	5.818786	125.000000
95.929634	5.818786	0.000000

Import x,y,z as Northing, Easting, Elevation

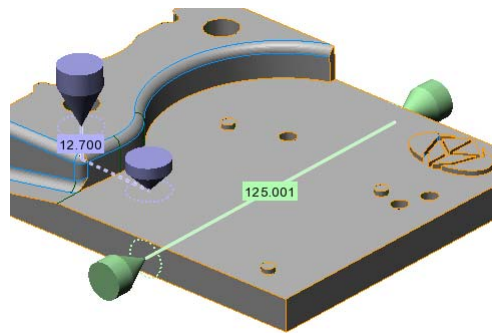
8. Click **OK**.

Results

The new calipers, with a nominal component, are added to the Tree View under the Gauges > Calipers branch.



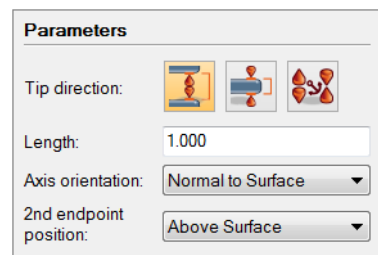
A caliper with a single axis and one with offset axes are displayed in the 3D scene.



Notes

- By default, if a Data object is available and aligned when the caliper is created, the measured component is extracted automatically.
- The **One Endpoint** submethod allows creating calipers by specifying only the first endpoint. The second endpoint is automatically computed using the specifications in the **Parameters** section.

The parameters are slightly different than the ones for the **Two Endpoints** submethod, but the creation workflow is the same.



Extracting the geometry of the part

For an existing caliper, the corresponding geometry of the part can be obtained by extraction on an aligned Data object. In PolyWorks|Inspector, the geometry obtained on the part is called the *measured component* of the measurement object.

When there is a Data object available in the project at caliper creation, the measured component is extracted automatically. Otherwise, when the Data object becomes available and is aligned, a manual extraction operation must be performed using the caliper.

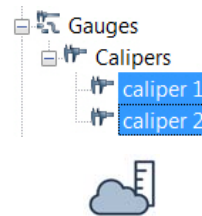
Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Objective: Finish inspecting the surface location for an existing caliper by extracting the corresponding geometry of the part to obtain the measured component of the caliper.

Required: The project contains a Reference object, an aligned Data object, and a caliper with a nominal component.

Steps

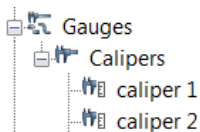
1. In the Tree View, select the calipers.



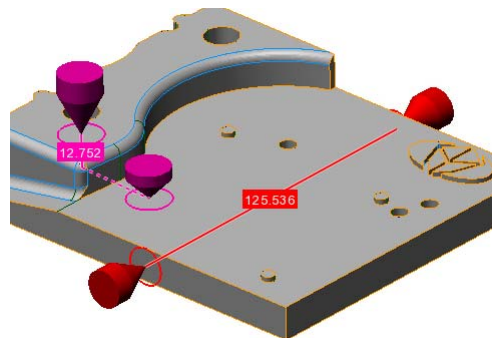
2. Choose *Measure > Extract Measured*.

Results

A measured status icon indicates the measured status of the calipers.



A caliper with a single axis and one with offset axes are displayed in the 3D scene. The color of the calipers corresponds to the defined color scale and it reflects, by default, the deviation of the measured component from the nominal component.



Notes

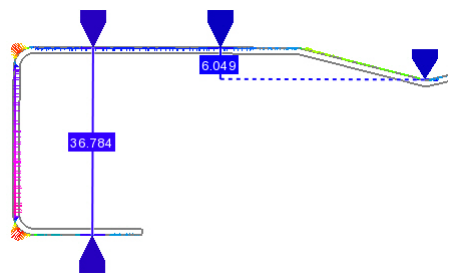
- The extraction parameters of a caliper are saved to the **Measurement** tab of its property sheet. They can be edited post creation by selecting the caliper in the Tree View, choosing *Edit > Object Properties*, opening the tab, and making other specifications.
- Caliper endpoints can be moved inward or outward after creation. To do so, select the caliper in the Tree View and choose *Measure > Gauges > Drag Caliper Endpoints*.



Creating cross-section calipers to measure distances on cross-sections

Cross-sections allow the 2D analysis of a part at specific locations. A cross-section caliper allows further 2D analysis by measuring distances using only the contours of cross-sections.

A cross-section caliper is a measurement object. It is created to obtain a) the nominal geometry of the Reference object, and b) the geometry of the part by extracting it from an aligned Data object. When both geometries have been obtained, geometry controls are applied to produce the desired results.



Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Start inspecting a distance on a cross-section by creating a cross-section caliper to capture the nominal component.

Required: The project contains a Reference object and a cross-section with a nominal component.

Steps

1. In the Tree View, select the cross-section and choose *View > Object Navigator > Cross-Sections*.

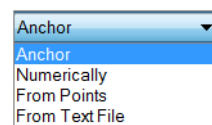
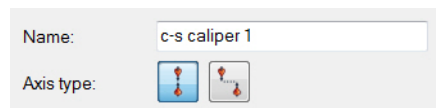
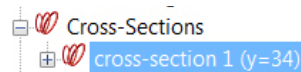
The Cross-Section Navigator dialog box opens, and the selected cross-section is displayed in 2D in the 3D scene.

2. Choose *Measure > Gauges > Create > Cross-Section Calipers*.

The Create Cross-Section Calipers dialog box opens.

3. Select an **Axis type**:
 - **Single Axis** allows creating calipers using one single axis for both endpoints.
 - **Offset Axes** allows creating calipers using one axis for each endpoint.
4. Choose a **Method** from those that follow.

The creation of cross-section calipers using the most frequently used methods follows. All of the possible creation methods are shown to the right.



Anchor

This method allows anchoring cross-section caliper endpoints interactively.

1. In the **Method** list, select **Anchor**.
2. In the **Submethod** list, select **Two Endpoints**.

This allows creating a caliper by specifying the two endpoints. If using offset axes, this is the only submethod.

3. Select a **Tip direction**:
 - **Automatic** specifies the tip direction to automatically point toward the surface where the endpoints are located.
 - **Custom** allows configuring the tip direction to face inward, outward, or to be anchored in space.

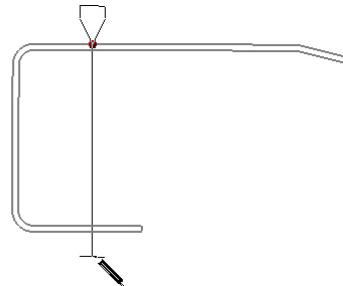
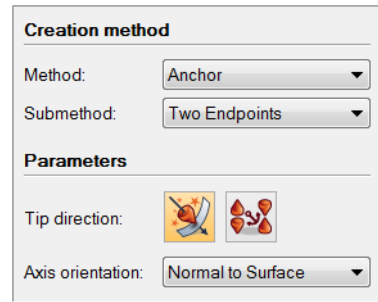
4. Specify the **Axis orientation**.

The caliper axis can be normal to the surface, follow a standard axis, a custom vector or be defined by the endpoints.

5. Click **Anchor**.

An interactive anchoring mode is launched:

- Pick the endpoints of the caliper on the nominal component of the cross-section.
- Repeat if desired.
- Right-click to end the anchoring operation.



Numerically

This method allows entering numerical values for the coordinates of the cross-section caliper endpoints.

1. In the **Method** list, select **Numerically**.
2. In the **Submethod** list, select **Two Endpoints**.

This allows creating a caliper by specifying the two endpoints. If using offset axes, this is the only submethod.



3. Select a **Tip direction**:
 - **Automatic** specifies the tip direction to point automatically toward the surface where the endpoints are located.
 - **Custom** allows configuring the tip direction to face inward, outward, or to be anchored in space.

Creation method

Method:

Submethod:

Parameters

Tip direction:  

1st endpoint		2nd endpoint	
x:	<input type="text" value="0.000"/>	x:	<input type="text" value="0.000"/>
y:	<input type="text" value="0.000"/>	y:	<input type="text" value="0.000"/>
z:	<input type="text" value="0.000"/>	z:	<input type="text" value="0.000"/>

4. Under **1st endpoint** and **2nd endpoint**, enter the (x, y, z) coordinates of the endpoints.
5. If using offset axes, specify the **Axis orientation**.

The caliper axis can be normal to the surface, follow a system axis, a custom vector or be defined by the endpoints.

6. Click **Create**.

From Text File

This method allows creating calipers by importing information from a text file. While the file must contain the coordinates of one or both endpoints, other information such as vectors, nominal lengths, radii, and names can also be imported.

1. In the **Method** list, select **From Text File**.
2. In the **Submethod** list, select **Two Endpoints**.

This allows creating a caliper by specifying the two endpoints. If using offset axes, this is the only submethod.

3. Select a **Tip direction**:
 - **Automatic** specifies the tip direction to point automatically toward the surface where the endpoints are located.
 - **Custom** allows configuring the tip direction to face inward, outward, or to be anchored in space.

4. If using offset axes, specify the **Axis orientation**.

The caliper axis can be normal to the surface, follow a system axis, a custom vector or be defined by the endpoints.

5. Click **Browse**.

A file browser opens.

6. Navigate to a text file containing the (x, y, z) coordinates of the endpoints.

7. In the import dialog box, under **Text files**, select a **Template** that corresponds to the file structure.

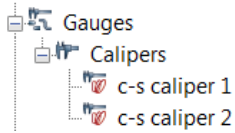
A preview of the file is presented, which allows verifying that the chosen template is correct.

X	Y	Z
15.660246	34.000000	0.000000
15.660246	34.000000	37.000000
40.774338	34.000000	0.000000
81.116917	34.000000	6.241749

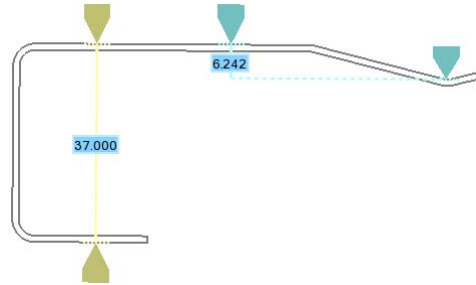
8. Click **OK**.

Results

The new cross-section calipers, with a nominal component, are added to the Tree View under the Gauges > Calipers branch.



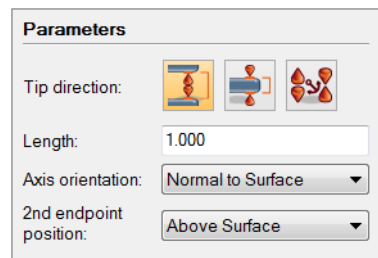
Single axis and offset axis cross-section calipers are displayed in the 3D scene.



Notes

- By default, if the cross-section has a measured component when the cross-section caliper is created, the measured component of the caliper is extracted automatically.
- The **One Endpoint Submethod** allows creating cross-section calipers by only specifying the first endpoint. The second endpoint is automatically computed using specifications in the **Parameters** section.

The parameters are slightly different than the ones for the **Two Endpoints** submethod, but the creation workflow is the same.



Extracting the geometry of the part

For an existing cross-section caliper, the corresponding geometry of the part can be obtained by extraction on an aligned Data object. In PolyWorks | Inspector, the geometry obtained on the part is called the *measured component* of the measurement object.

When there is a Data object available in the project at caliper creation, the measured component is extracted automatically. Otherwise, when the Data object becomes available and is aligned, a manual extraction operation must be performed using the cross-section caliper.

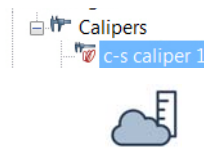
Objective: Finish inspecting the distance for an existing cross-section caliper by extracting the corresponding geometry of the part to obtain the measured component of the caliper.

Required: The project contains a cross-section caliper with a nominal component and its linked cross-section, and an aligned Data object.

Steps

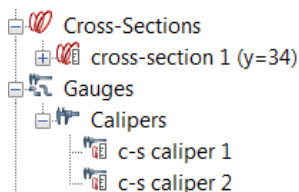
1. Select a cross-section caliper in the Tree View.
2. Choose *Measure > Extract Measured*.

Extracting the cross-section calipers triggers the extraction of measured component of their linked cross-section.

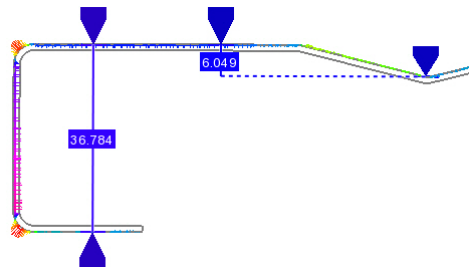


Results

A measured status icon indicates the measured status of the cross-section calipers and the linked cross-section.



Single axis and offset axis cross-section calipers are displayed in the 3D scene. The color of the calipers corresponds to the defined color scale and reflects, by default, the deviation of the measured component from the nominal component.



Notes

- The extraction parameters of a caliper are saved to the **Measurement** tab of its property sheet. They can be edited post creation by selecting the caliper in the Tree View, choosing *Edit > Object Properties*, opening the tab, and making other specifications.
- A caliper's endpoints can be moved inward or outward after creation. To do so, select the caliper in the Tree View and choose *Measure > Gauges > Drag Caliper Endpoints*.

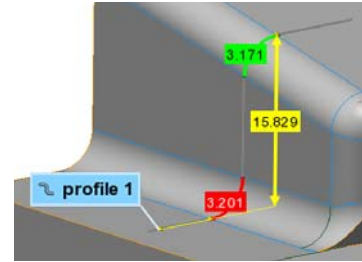


Measuring object radii with profile gauges

The profile gauge is a useful tool to inspect radii on a part on a cross-sectional plane.

Introducing profile gauges

The profile gauge is used to inspect a profile along a cross-sectional plane using the nominal geometry and the geometry of the part in order to produce results determined by geometry controls.



Several profile gauge types are available that allow inspecting one radius at a profile, or two radii and the distance between them.

Profile gauges are available on the *Measure > Gauges* menu or on the **Gauges** toolbar.



Profile gauge families

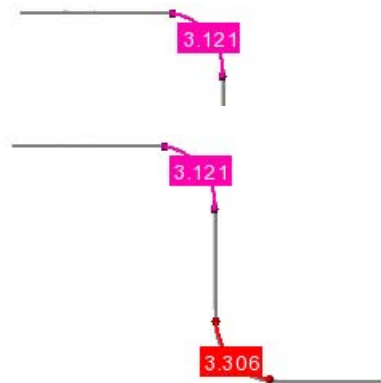
There are three families of profile gauges.

One radius

This family type computes measurements only for one radius.

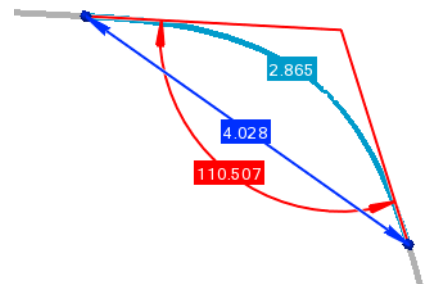
Two radii

This family type computes measurements for two radii.



Design lines

This family type detects one variable radius representing a feature line. Feature lines are characteristic lines created into the side of a car to give it visual distinction.



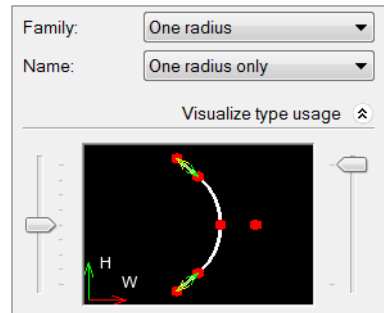
Profile gauge types

There are several types of profile gauges offered for each family, and certain types obtain more accurate results in certain applications. To help choose the correct type of profile gauge, the **Visualize type usage** section in the creation dialog box offers an interactive display of the different scenarios where the selected type works.

The selection of gauges for each family meets the needs of most users. In addition, users can program their own gauges using special creation tools; this advanced subject is not presented in this document.

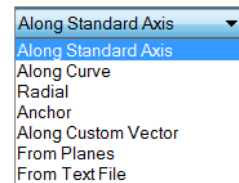
Visualizing type usage

1. In the creation dialog box, expand the **Visualize type usage** area.
2. A display of the current gauge type is shown.
3. Use the vertical scroll bar to interactively change the display, showing the possible scenarios supported by the selected gauge type. If the two radii families are selected, two scroll bars are available, one for each radius.



Creation methods

Profile gauge measurements are calculated on a linked standard cross-section, which is created at the same time as the profile gauge. This is why profile gauges share the same creation methods as standard cross-sections. The methods most frequently used are presented in this document.



Creation methodology

1. Choose *Measure > Gauges > Create > Profile Gauges*.

The Create Profile Gauge dialog box opens.



2. Specify an object **Name**, if desired.

3. Specify the gauge **Family** and **Name**.

Use the **Visualize type usage** to display the possible scenarios supported by the selected profile gauge type.

4. In the **Method** list, select a creation method.

5. Specify the required parameters according to the creation method.

Method-specific parameters may include browsing for a text file or entering point coordinates manually.

6. Click **Preview** to view the measurement zone of the profile gauge.

7. Click the appropriate button to launch or confirm the creation of the gauges.

The dialog box contains the following fields and options:

- Name:** profile 1
- Type**
 - Family:** One radius
 - Name:** One radius only
 - Visualize type usage (dropdown arrow)
- Creation method**
 - Method:** Along Standard Axis
- Axis**
 - X
 - Y
 - Z
- Mode**
 - Individual
 - Multiple
- Position**
 - x: 0.000
 - Anchor Point (button)
- Options** (dropdown arrow)
- Preview (button)

Display options

Display options are available for the visualization of the results.

1. Click **Object Display Options**.



2. Under **Gauges**, in the **Component** list, choose **Nominal** or **Measured**.

The dialog box shows the following settings:

- Gauges** (selected)
- Color map
- Component:** Measured

PolyWorks | Inspector also offers a visualization mode called Profile Gauge Navigator that displays a single profile gauge in 2D mode, making it easier to understand the results. See [Reviewing profile gauge measurement results](#) on page 563.

Creating profile gauges

A profile gauge is a measurement object. It is created to obtain, along a cross-sectional plane, a) the nominal geometry on the Reference object, and b) the geometry of the part, by extracting it from an aligned Data object, or by probing the geometry on the part. When both geometries have been obtained, geometry controls are applied to produce the desired results.

Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Start inspecting a profile along a cross-sectional plane by creating a profile gauge to capture the nominal component.

Required: A Reference object. If there is a Data object in the project, it must be aligned to the corresponding Reference object.

Steps

1. Choose *Measure > Gauges > Create > Profile Gauges*.

The Create Profile Gauges dialog box opens.



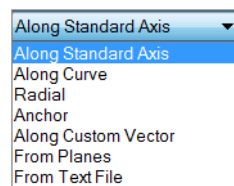
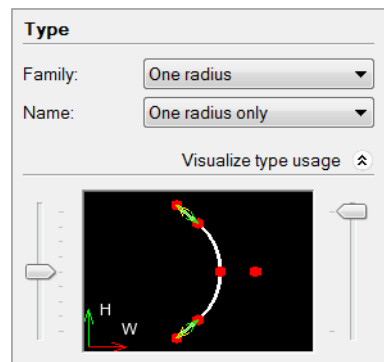
2. Choose a gauge **Family** type and **Name**.

*To better understand how to choose the type, the **Visualize type usage** section can be expanded. In this section, all operations that the selected type can perform are illustrated. This is for visualization purposes only, and does not affect the creation process.*

3. Choose a **Method** from those that follow.

The creation of profile gauges, using the most frequently used methods, follows.

All of the creation methods are shown to the right.



Along Curve

This method allows creating profile gauges perpendicular to a curve. It involves specifying the type of curve to use, and then specifying the location of the gauge along the curve.

1. In the **Method** list, select **Along Curve**.
2. Under **Curve**, select **From reference object curves** to use the curves of a Reference object.
3. Under **Mode**, select **Individual**.

*If multiple profile gauges are needed, the **Multiple** mode allows gauges to be spaced evenly at a specified step along the curve.*

4. Under **Position**, click **Anchor Point**.
An interactive anchoring mode is launched.

5. Anchor a point along the curve by picking a point in the 3D scene. Right-click to exit the anchoring mode.

*If **At intersection with plane** is selected, only the location of the intersection of the curve and the chosen standard plane can be picked.*

The measurement zone is displayed.

6. To resize the measurement zone to include only the area of interest, drag the blue borders.
7. Click **Confirm**.

Creation method

Method:

Curve

From reference object curves
 From polyline

Mode

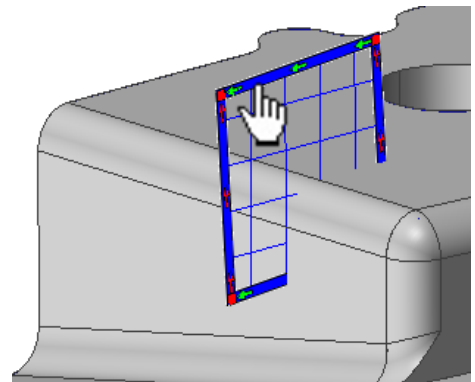
Individual
 Multiple

Position

At intersection with plane:

XY YZ ZX

z:



Tips

Profile gauges can be created along a whole curve or just using a portion of the curve. When the **Mode** is set to **Multiple**, two anchoring modes become available:

- **Whole curve:** Profile gauges are created along the entire length of the curve.
- **Partial curve:** Profile gauges are created between the specified start and end point on the curve. Click a first point on the curve to indicate the start point and click a second point to indicate the end point.

Along Standard Axis

This method allows creating profile gauges perpendicular to a standard system axis, such as the X axis. It is recommended to have a properly oriented coordinate system to obtain the anticipated results.

1. In the **Method** list, select **Along Standard Axis**.

2. Under **Axis**, select **X**, **Y**, or **Z**.

3. Under **Mode**, select **Individual**.

*If multiple profile gauges are needed, the **Multiple** mode creates gauges evenly spaced at a specified step, either within a range or by slicing whole objects.*

4. Under **Position**, specify the coordinate of a point on the selected standard axis.

*The value can be entered in the text box or by clicking **Anchor Point**, which allows picking a point on the Reference object in the 3D scene.*

5. Click **Preview**.

A preview of the measurement zone of the profile gauge is displayed in the 3D scene.

The preview is automatically displayed when anchoring a point.

Right-click to exit the anchoring mode.

6. To resize the measurement zone to include only the area of interest, drag the blue borders.

7. Click **Confirm**.

Creation method

Method:

Axis

X Y Z

Mode

Individual Multiple

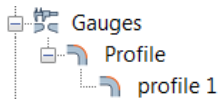
Position

x:

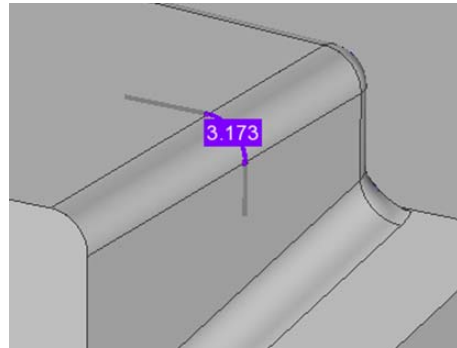
Options

Results

The new profile gauge, with a nominal component, is added to the Tree View under the Gauges > Profile branch.

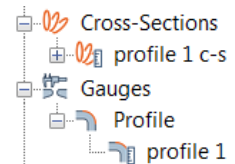


The profile gauge is displayed in the 3D scene.



Notes

- By default, if a Data object is available and aligned when the profile gauge is created, the measured component is extracted automatically.
- Since a profile gauge is a 2D analysis, a linked cross-section is automatically extracted on profile gauge creation to compute measurements.
- The measurement zone of a profile gauge can be edited post creation by following these steps:

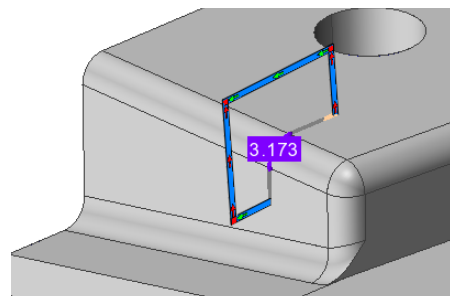


1. Select the profile gauge in the Tree View.
2. Choose *Measure > Gauges > Edit Measurement Zones*.



The measurement zone is displayed in the 3D scene and can be edited by dragging the borders.

3. Press ESC to exit the editing mode.



Measuring the geometry on the part

For an existing profile gauge, the corresponding geometry on the part can be obtained by extraction on an aligned Data object or by probing a physical part. If proceeding by probing, the digitizing device should be aligned to the Reference object. In PolyWorks|Inspector, the geometry obtained on the part is called the *measured component* of the measurement object.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Extract the geometry

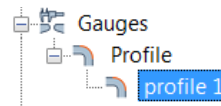
When there is an aligned Data object available in the project at profile gauge creation, the measured component is extracted by default. Otherwise, when the Data object becomes available and is aligned, a manual extraction operation must be performed using the profile gauge.

Objective: Finish inspecting the profile for an existing profile gauge by extracting the gauge's measured component.

Required: A profile gauge and an aligned Data object.

Steps

1. In the Tree View, select the profile gauge.

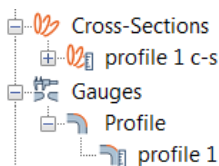


2. Choose *Measure > Extract Measured*.



Results

A measured status icon indicates the *measured* status of the profile gauge as well as its linked cross-section.



The profile gauge is displayed in the 3D scene.



Notes

- If the profile gauge is created using the wrong family or name, it is possible to change these parameters as well as other gauge creation parameters. Select the gauge and choose *Edit > Object Properties*. The object's property sheet opens. Open the **Profile** tab and make the desired specifications.

- It is possible to view in 2D mode the selected profile gauges using the Profile Gauge Navigator. To access the navigator, choose *View > Object Navigator > Profile Gauges*. See [Reviewing profile gauge measurement results](#) on page 563.

Probe the geometry

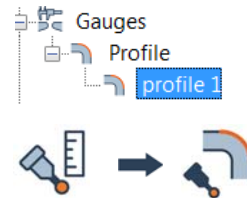
The geometry for a profile on the part can be obtained by probing the measured component of a profile gauge. The probed points are projected onto the plane defined by the measurement zone of the gauge. Subsequently, the projected points are joined by a polyline to define the measured component of the linked cross-section.

Objective: Finish inspecting the profile for an existing profile gauge by probing the corresponding part geometry.

Required: A Reference object, a profile gauge, a probing device aligned with the Reference object, and the part.

Steps

- In the Tree View, select the profile gauges to probe.

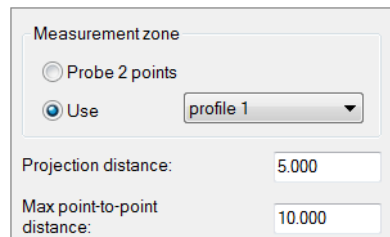


- Choose *Measure > Gauges > Probe Measured Profile*.

The Probe Measured Profile Gauge Components dialog box opens.

- In the **Measurement zone** area, select **Use**.

The selected profile gauges are available in the adjacent list.



- Specify a **Projection distance**.

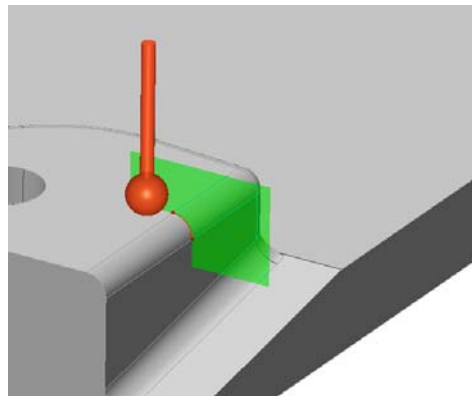
Probed points within this distance from the measurement zone are considered as valid points to compute the measured component.

- Specify a **Max point-to-point distance**.

Specifies the maximum distance between points to be joined together by the polyline.

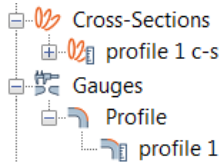
- Click **Probe**.

Use the visual and audio guidance to move the probe to the desired location. Once within the profile projection distance, probing is allowed.

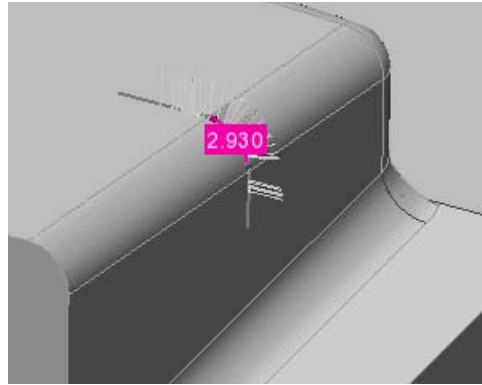


Results

A measured status icon indicates the *measured* status of the profile gauge as well as its linked cross-section.



The profile gauge is displayed in the 3D scene.

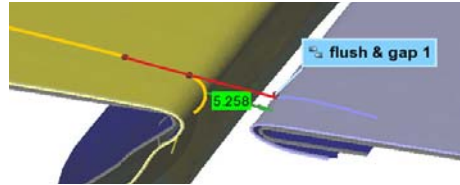


Notes

- Measured profile gauges can also be probed without having an existing profile gauge by using the **Probe 2 points** method. In this case, a measurement zone must first be defined. As well, no alignment is necessary for the probing device since no Reference object is required.

Measuring flush and gap with flush & gap gauges

In manufacturing, when performing an assembly, it can be necessary to measure the flush and the gap existing between the assembled parts to assure the assembly meets the design requirements.



Introducing flush & gap gauges

The flush & gap gauge is used to inspect the assembly on a cross-sectional plane using the nominal geometry and the geometry of the part in order to produce results determined by geometry controls.

Flush & gap gauges are available on the *Measure > Gauges* menu or on the **Gauges** toolbar.



Flush & gap gauges are organized by family, and each family can have several types of flush & gap gauges. This selection of gauges meets the needs of most users. Users can program their own gauges using special creation tools; this advanced subject is not covered in this document.

Flush & gap gauge families

There are four families of flush & gap gauges: **Side B is a line**, **Side B is made of points**, **Side A and B are fillets**, and **Universal**.

The **Universal** family is the most versatile one, enclosing almost every possible scenario where both sides are fillets. Since it offers more flexibility, it is the only family that is addressed in detail. While the types offered in the **Universal** family have wider applications, the ones offered for the **Side A and B are fillets** family have more specific applications.

Flush & gap gauge types

The **Universal** family offers three standard flush and gap gauge types:

- **AutoDirections Universal**

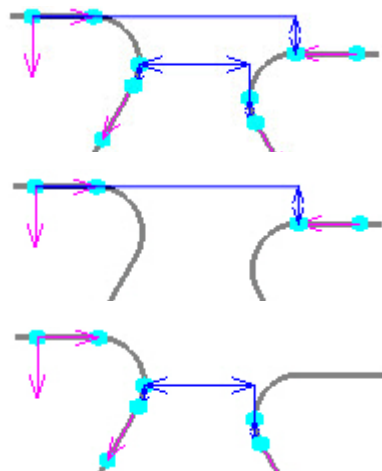
This type of flush and gap gauge computes both the gap and the flush whenever A and B side are fillets.

- **AutoDirections Universal Flush Only**

This type of flush and gap gauge computes only the flush whenever A and B side are fillets.

- **AutoDirections Universal Gap Only**

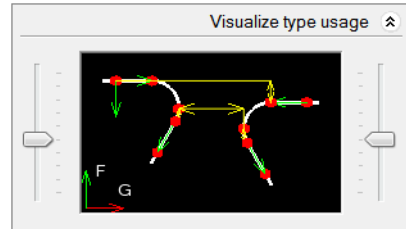
This type of flush and gap gauge computes only the gap whenever A and B side are fillets.



Visualizing type usage

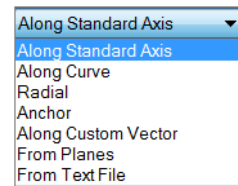
Each flush & gap gauge family and combination of types address specific needs of measurement. To help choose the correct family and type, the **Visualize type usage** section in the creation dialog box offers an interactive display of the different scenarios where the selected type works.

1. In the creation dialog box, expand the **Visualize type usage** area.
2. A display of the current gauge type is shown.
3. Use the vertical scroll bar on each side to interactively change the display, showing the possible scenarios supported by the selected gauge type.



Creation methods

Flush & gap gauge measurements are calculated along a linked standard cross-section, which is created at the same time as the flush & gap gauge. This is why flush & gap gauges use the same creation methods as standard cross-sections. The methods most frequently used are presented in this document.



Creation methodology

1. Choose *Measure > Gauges > Create > Flush & Gap Gauges*.

The Create Flush & Gap Gauges dialog box opens.



2. Specify an object **Name**, if desired.
3. Specify the gauge **Family** and **Name**.
*Use the **Visualize type usage** to display the possible scenarios supported by the selected flush & gap gauge type.*
4. Specify a creation **Method**.
5. Specify the required parameters according to the creation method.
Method-specific parameters may include browsing for a text file or entering point coordinates manually.
6. Click **Preview** to view the measurement zone of the flush and gap gauge.
7. Click the appropriate button to launch the gauge creation.

Name:

Type

Family:

Name:

Visualize type usage ▾

Creation method

Method:

Axis

X Y Z

Mode

Individual
 Multiple

Position

x:

Options ▾

Display options

Display options are available for the visualization of the results.

1. Click **Object Display Options**.
2. Under **Gauges**, in the **Component** list, choose **Nominal** or **Measured**.



Gauges

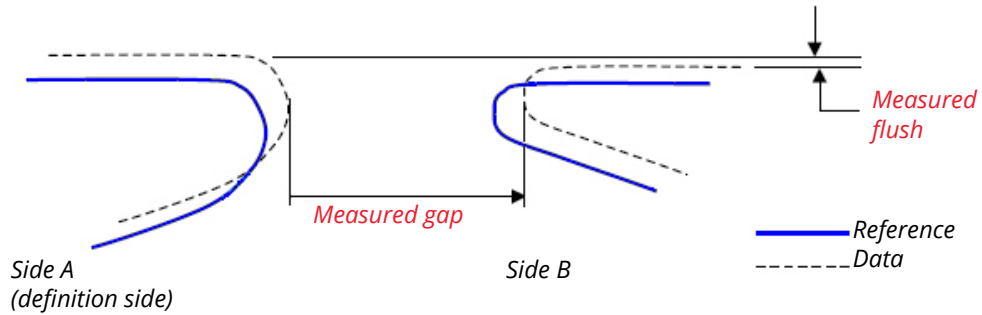
Color map

Component:

PolyWorks | Inspector also offers a visualization mode called Flush & Gap Gauge Navigator that displays a flush & gap gauge in 2D mode, making it easier to read the results. See [Reviewing flush & gap gauge measurement results](#) on page 565.

Typical flush and gap analysis

The figure below illustrates the flush and the gap measured on the part.



Creating flush & gap gauges

A flush & gap gauge is a measurement object. It is created to obtain, along a cross-sectional plane, a) the nominal geometry on the Reference object, and b) the geometry of the part, by extracting it from an aligned Data object, or by probing the geometry of the part. When both geometries have been obtained, geometry controls are applied to produce the desired results.

Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Start inspecting the flush & gap on an assembly by creating a flush & gap gauge to capture the nominal component.

Required: A Reference object assembly, or separate Reference objects properly aligned to each other.
If there are Data objects in the project, they must be aligned to the corresponding Reference objects.

Steps

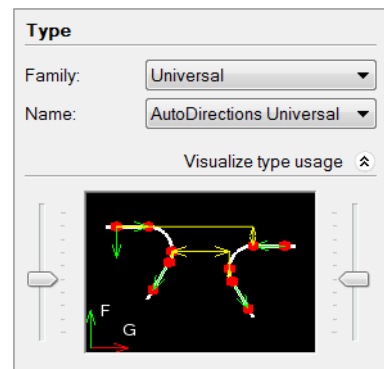
1. Choose *Measure > Gauges > Create > Flush & Gap Gauges*.

The Create Flush & Gap Gauges dialog box opens.



2. Choose the **Family** type and the name.

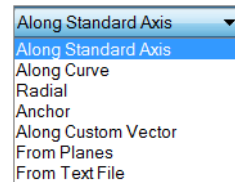
Use the **Visualize type usage** section to better understand how to choose the type. The operations that the selected type performs are illustrated. This is only for visualization purposes and does not affect the creation process.



3. Choose the creation **Method** from the methods that follow.

The creation of flush & gap gauges, using the methods most often used, follows.

All of the creation methods are shown to the right.



Along Curve

This method allows creating flush & gap gauges perpendicular to a curve. It involves specifying the type of curve to use, and then specifying the location of the gauge along the curve.

1. In the **Method** list, select **Along Curve**.
2. Under **Curve**, select **From reference object curves** to use the curves of a Reference object.
3. Under **Mode**, select **Individual**.

*The **Multiple** mode allows creating evenly spaced flush & gap gauges at a specified step either within a range or by slicing whole objects.*

Creation method

Method:

Curve

From reference object curves
 From polyline

Mode

Individual
 Multiple

4. Under **Position**, click **Anchor Point**.
An interactive anchoring mode is launched.

5. Anchor a point along the curve by picking a point on the Reference object in the 3D scene. Right-click to exit the anchoring mode.

Position

At intersection with plane:

XY YZ ZX

z:

*If **At intersection with plane** is selected, only the point where the curve and the specified standard plane intersect can be picked.*

The measurement zone is displayed.

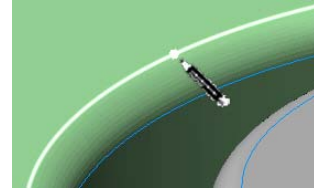
6. To resize the measurement zone to include only the area of interest, drag the blue borders.
7. Click **Confirm**.

Tips

Flush & gap gauges can be created along a whole curve or just using a portion of the curve. When the **Mode** is set to **Multiple**, two anchoring modes become available:

- **Whole Curve:**

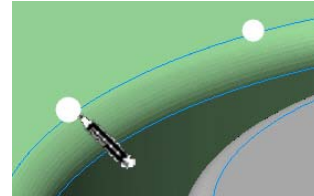
Flush & gap gauges are created along the entire length of the curve.



- **Partial Curve:**

Flush & gap gauges are created between the specified start and end points on the curve.

Click a first point to indicate the start point, and click a second point to indicate the end point on the curve.



Along Standard Axis

This method allows creating flush & gap gauges perpendicular to a standard axis. It is recommended to have a properly oriented coordinate system to obtain the anticipated results.

1. In the **Method** list, select **Along Standard Axis**.
2. Under **Axis**, select **X**, **Y**, or **Z**.
3. Under **Mode**, select **Individual**.

*The **Multiple** mode allows creating evenly spaced flush & gap gauges at a specified step either within a range or by slicing whole objects.*

Creation method

Method: Along Standard Axis

Axis

X
 Y
 Z

Mode

Individual
 Multiple

4. Under **Position**, specify the coordinate of a point on the selected standard axis.

*The value can be entered in the text box or by clicking **Anchor Point**, which allows picking a point on the Reference object in the 3D scene.*

5. Click **Preview**.

Position

x:

Anchor Point

Options ⌵

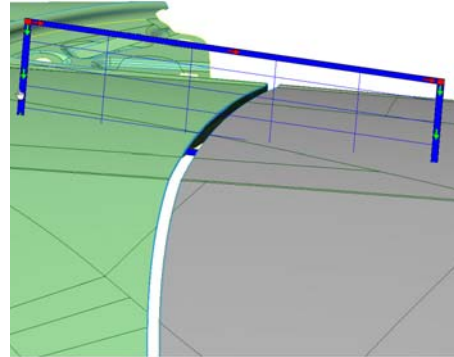
Preview

A preview of the measurement zone of the flush & gap gauge is displayed in the 3D scene.

The preview is automatically displayed when anchoring a point.

Right-click to exit the anchoring mode.

6. To resize the measurement zone to include only the area of interest, drag the blue borders.
7. Click **Confirm**.



From Text File

It is possible to create the flush & gap gauges from predefined points by importing them from a text file.

1. In the **Method** list, select **From Text File**.
2. Specify the **File contents**:
 - **Ordered Points**: the file contains a list of ordered consecutive points.
 - **Points & Gap Vectors**: the file contains a set of points and gap vectors, and optionally flush vectors.

Creation method	
Method:	From Text File ▼
File contents:	Ordered Points ▼

- Under **Position**, specify the path of the text file to be used. If the file content has been set to **Ordered Points**, additional information must be specified:

- **Number of gauges per curve segment:** specifies the number of gauges to create along each curve segment (line joining two consecutive points).
- **Distance between gauges:** specifies the spacing between gauges along the curve.

Position

Text file: ...

Number of gauges per curve segment:

Distance between gauges:

- Click **Preview**.

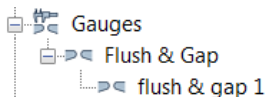
A preview of the measurement zone of the flush & gap gauge is presented in the 3D scene.

- To resize the measurement zone to include only the area of interest, drag the blue borders.

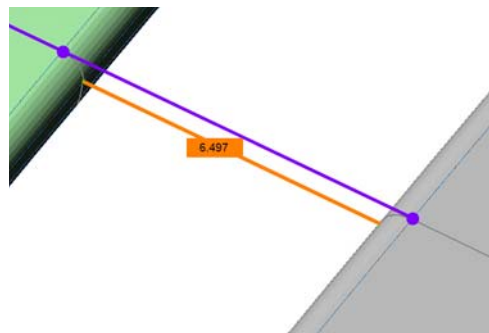
- Click **Confirm**.

Results

The new flush & gap gauge, with a nominal component, is added to the Tree View under the Gauges > Flush & Gap branch.

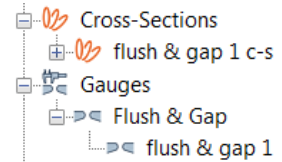


The flush & gap gauge is displayed in the 3D scene.



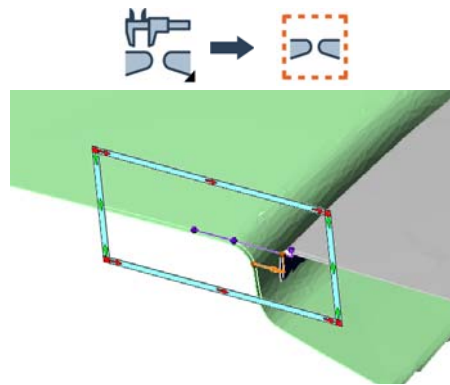
Notes

- By default, if Data objects are available and aligned when the flush & gap gauge is created, the measured component is extracted automatically.
- Since a flush & gap gauge is a 2D analysis, a linked cross-section is automatically extracted on flush & gap gauge creation to compute measurements.



- The measurement zone of a flush & gap gauge can be edited post creation by following these steps:

1. Select the flush & gap gauge in the Tree View.
2. Choose *Measure > Gauges > Edit Measurement Zones*.
3. The measurement zone is displayed in the 3D scene and can be edited by dragging the borders.
4. Press ESC to exit the editing mode.



Measuring the geometry on the part

For an existing flush & gap gauge, the corresponding geometry of the part can be obtained by extraction on an aligned Data object or by probing a physical part. If proceeding by probing, the digitizing device must be aligned to the Reference object. In PolyWorks|Inspector, the geometry obtained on the part is called the *measured component* of the measurement object.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Extract the geometry

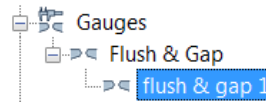
When there is an aligned Data object available in the project at flush & gap gauge creation, the measured component can be extracted automatically. Otherwise, when the Data object becomes available and is aligned, a manual extraction operation must be performed using the flush & gap gauge.

Objective: Finish inspecting the flush and gap on a part assembly for an existing flush & gap gauge by extracting the gauge's measured component.

Required: A flush & gap gauge, and properly aligned Data objects.

Steps

1. In the Tree View, select the flush & gap gauge.

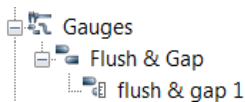


2. Choose *Measure > Extract Measured*.

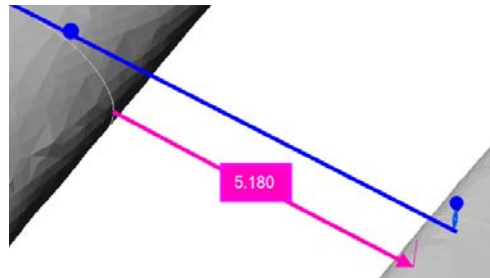


Results

A measured status icon indicates the measured status of the flush & gap gauge.



The flush & gap gauge is displayed in the 3D scene.



Notes

- If the flush & gap gauge is created using the wrong family or name, it is possible to change these parameters and other gauge creation parameters. Select the gauge and choose *Edit > Object Properties*. The object's property sheet opens. Open the **Flush & gap** tab and make the desired specifications.
- It is possible to view in 2D mode the selected flush & gap gauges using the Flush & Gap Gauge Navigator. To access the navigator, choose *View > Object Navigator > Flush & Gap Gauges*. See [Reviewing flush & gap gauge measurement results](#) on page 565.

Probe the geometry

The geometry for a flush and gap on the part can be obtained by probing the measured component of a flush & gap gauge. The probed points are projected onto the plane defined by the measurement zone of the gauge. Subsequently, the projected points are joined by polylines to define the measured component of the linked cross-section.

Objective: Obtain the gauge's measured component by probing the corresponding part geometry.

Required: A Reference object, a flush & gap gauge, a probing device aligned with the Reference object, and the part.

Steps

1. In the Tree View, select the flush & gap gauges to probe.
2. Choose *Measure > Gauges > Probe Measured Flush & Gap*.
The Probe Measured Flush & Gap Gauge Components dialog box opens.
3. In the **Measurement zone** area, select **Use**.
The selected flush & gap gauges are available in the adjacent list.
4. Specify a **Projection distance**.
Probed points within this distance from the measurement zone are considered as valid points to compute the measured component.



Measurement zone

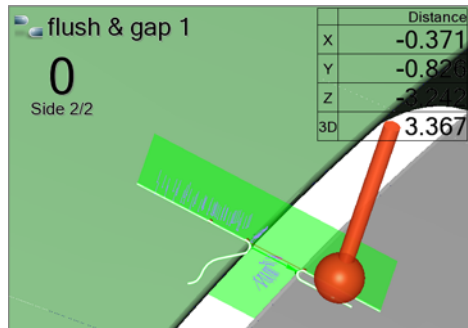
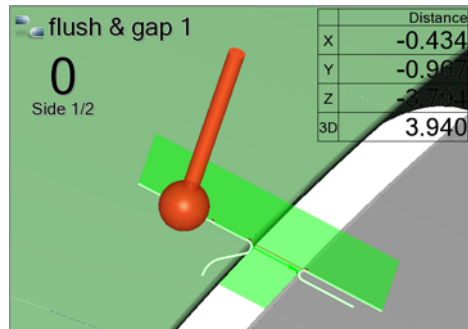
Probe 2 points

Use flush & gap 1

Projection distance:

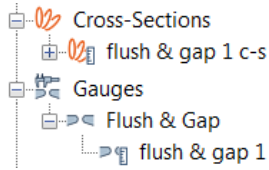
Max point-to-point distance:

5. Specify a **Max point-to-point distance**.
Specifies the maximum distance between points to be joined together by the polyline.
6. Click **Probe**.
Use the visual and audio guidance to move the probe to the desired location. Once within the projection distance, probing is allowed.
Measuring the flush and gap requires probing two sides and each side is probed in a separate step.

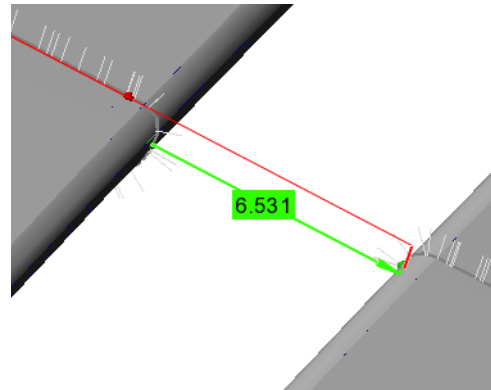


Results

A measured status icon indicates the *measured* status of the flush & gap gauge as well as its linked cross-section.



The flush & gap gauge is displayed in the 3D scene.

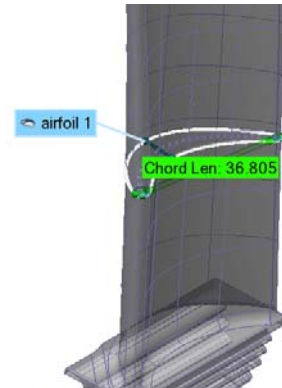


Notes

- Measured flush & gap gauges can also be probed without having an existing flush & gap gauge by using the **Probe 2 points** method. In this case, a measurement zone must first be defined. Moreover, no alignment is necessary for the probing device since no Reference object is required.

Measuring airfoil geometry with airfoil gauges

From domestic fans to turbines, blades play an important role in the manufacturing field. They make it possible to use fluid and gas dynamic principles to either absorb energy from fluids and gases, or produce energy. As a result, the design of their cross-sectional shape, called airfoil, must be accurately calculated and inspected.



Introducing airfoil gauges

The airfoil gauge is used to inspect standard airfoil characteristics on a cross-sectional plane using the nominal geometry and the geometry of the part in order to produce results determined by geometry controls.

Airfoil gauges are available on the *Measure > Gauges* menu or on the **Gauges** toolbar.



Airfoil gauges can measure the chord, the camber line, the leading and trailing edge radii and the maximum thickness, and more.

Airfoil gauge families

There are two families of airfoil gauges.

- **Aerodynamic**

Measurements are based on the aerodynamic chord that is defined as the vector that joins the limits of the camber line.



- **Tangential**

Measurements are based on the tangential chord that is defined as the vector that joins the two tangential points.



For both families, only the **Classic** name is available, which results in **Aerodynamic - Classic** or **Tangential - Classic**, depending upon the family chosen.

Airfoil edges

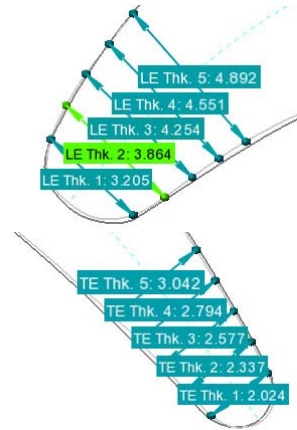
Analyzing airfoils implies analyzing their edges, called leading edge (LE) and trailing edge (TE).

- **Leading edge**

Defined by the area of maximum curvature at the front of the airfoil.

- **Trailing edge**

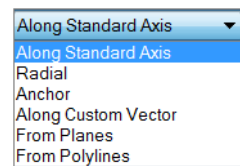
Defined by the area of maximum curvature at the rear of the airfoil.



An airfoil typically has both edges. However, some sections can only present the leading edge or the trailing edge. PolyWorks | Inspector allows analyzing the edges simultaneously, or individually, depending on the scenario. The shape of the edges can be either circular or cutback.

Creation methods

Airfoil gauge measurements are calculated on a linked standard cross-section, which is created at the same time as the airfoil gauge. This is why airfoil gauges share most of the same creation methods as standard cross-sections. The methods most frequently used are presented in this document.



Creation methodology

1. Choose *Measure > Gauges > Create > Airfoil Gauges*.

The Create Airfoil Gauges dialog box opens.

2. Specify a **Name**, if desired.
3. Specify the family **Type**.
4. Specify the **Edges** to analyze on the airfoil profile.
5. Specify the **Edges shapes**.
6. Specify the creation **Method**.
7. Specify the required parameters according to the creation method.
8. Click **Preview** to view the measurement zone of the airfoil gauge.
9. Click the appropriate button to launch the gauge creation.



Name:

Type

Family:

Name:

Edges:

Edge shapes:

Creation method

Method:

Axis

X Y Z

Mode

Individual Multiple

Position

x:

Display options

Display options are available for the visualization of the results.

1. Click **Object Display Options**.
2. Under **Gauges**, in the **Component** list, choose either **Nominal** or **Measured**.



Gauges

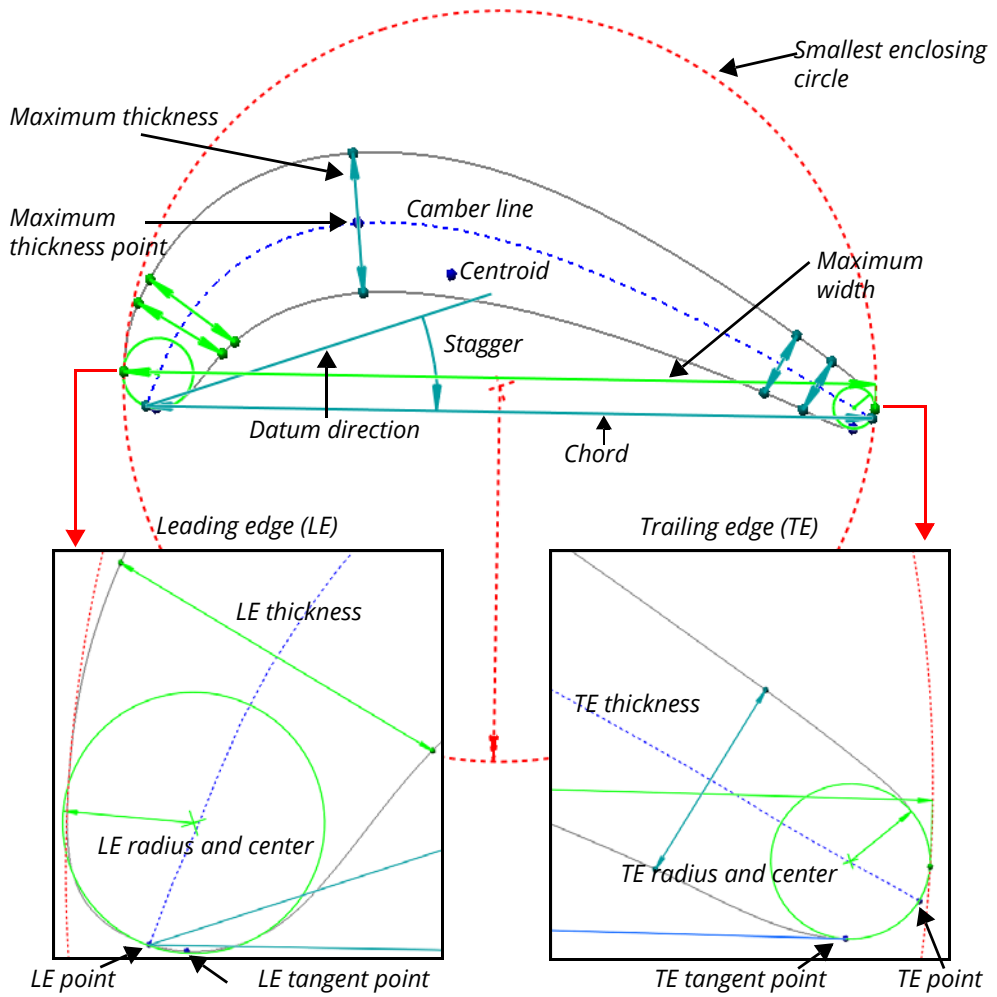
Color map

Component:

PolyWorks | Inspector also offers a visualization mode called Airfoil Gauge Navigator which displays the airfoil gauge in 2D mode, making it easier to read the results. See [Reviewing airfoil gauge measurement results](#) on page 567.

Typical airfoil analysis

This figure shows an example of airfoil geometry captured by an airfoil gauge and the standard airfoil characteristics that can be calculated and reported.



Creating airfoil gauges

An airfoil gauge is a measurement object. It is created to obtain, on a cross-sectional plane, a) the nominal geometry of the Reference object, and b) the geometry of the part by extracting it from an aligned Data object. When both geometries have been obtained, geometry controls are applied to produce the desired results.

Getting the nominal geometry

In PolyWorks|Inspector, the nominal geometry of a measurement object is called its *nominal component*, and the geometry obtained from the part is called its *measured component*.

Objective: Start inspecting the standard characteristics of an airfoil by creating an airfoil gauge to capture the nominal component.

Required: A Reference object that represents an airfoil.

Steps

1. Choose *Measure > Gauges > Create > Airfoil Gauges*.

The Create Airfoil Gauges dialog box opens.

2. Choose the **Family** type:

- **Aerodynamic:** measurements are based on the aerodynamic chord.
- **Tangential:** measurements are based on the tangential chord.

The **Name** offered for both families is **Classic**.

3. Specify the **Edges** to analyze:

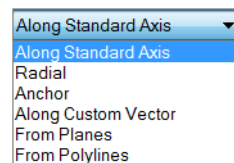
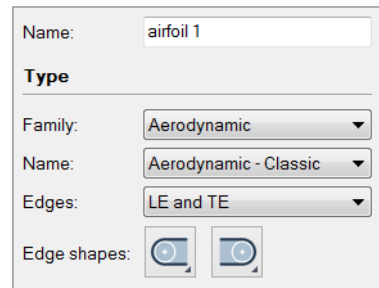
- **LE and TE:** leading edge and trailing edge.
- **LE:** leading edge only.
- **TE:** trailing edge only.

4. Set the **Edge shapes** of the edges to analyze:

- **Circular**
- **Cutback**

5. Choose a creation **Method** from the ones that follow.

The creation of airfoil gauges using the most frequently used methods follows. To the right, all of the possible creation methods are shown.



Along Standard Axis

This method allows creating airfoil gauges perpendicular to a standard axis. It is recommended to have a properly oriented coordinate system to obtain the anticipated results.

1. In the **Method** list, select **Along Standard Axis**.

Creation method

Method: Along Standard Axis

Axis

X Y Z

Mode

Individual Multiple

2. Under **Axis**, select **X, Y, or Z**.

3. Under **Mode**, select **Individual**.

*The **Multiple** mode allows creating evenly spaced airfoil gauges at a specified step either within a range or by slicing whole objects.*

4. Under **Position**, specify the coordinate of a point on the selected standard axis.

*The value can be entered in the text box or by clicking **Anchor Point**, which allows picking a point on the Reference object in the 3D scene.*

Position

x:

Anchor Point

Options ⌵

Preview

5. Click **Preview**.

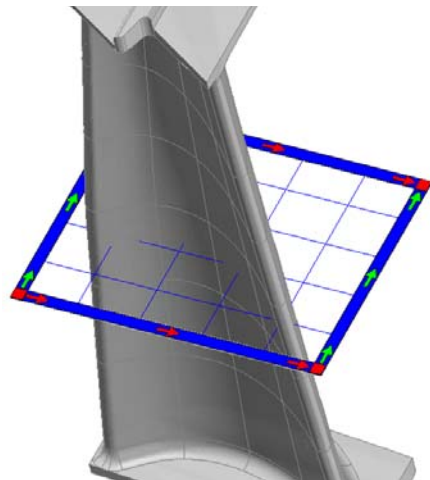
A preview of the measurement zone of the airfoil gauge is displayed in the 3D scene.

The preview is automatically displayed when anchoring a point.

Right-click to exit the anchoring mode.

6. Drag the blue borders to resize the measurement zone in order to include only the area of interest.

7. Click **Confirm**.



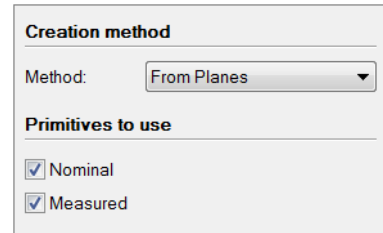
From Planes

This method allows creating airfoil gauges at the position of existing plane primitives.

1. In the Tree View, select the plane(s) to use.



2. In the **Method** list, select **From Planes**.



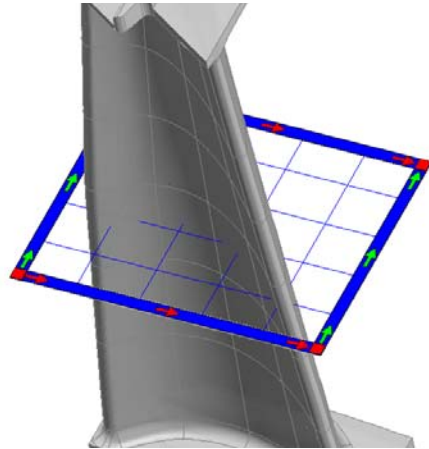
3. Specify the **Primitives to use**.

*By default, both of the primitive types, **Nominal** and **Measured**, are selected. If a selected plane has a nominal and a measured primitive, an airfoil gauge is created at the location of each primitive.*

4. Click **Preview**.

A preview of the measurement zone of the airfoil gauge is displayed in the 3D scene.

5. Drag the blue borders to resize the measurement zone in order to include only the area of interest.
6. Click **Confirm**.

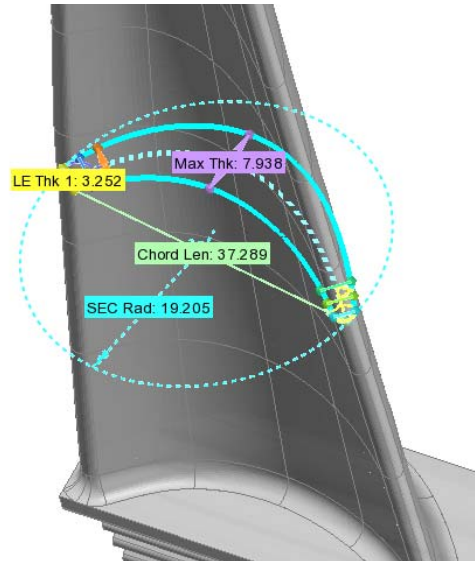


Results

The new airfoil gauge, with a nominal component, is added to the Tree View under the Gauges > Airfoils branch.

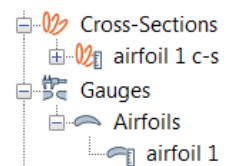


The airfoil gauge is displayed in the 3D scene.



Notes

- If a Data object is available and aligned when the airfoil gauge is created, then the measured component is extracted.
- An airfoil gauge uses a cross-section to compute the measurements since it is a 2D analysis. Because of this, a linked cross-section is created automatically on airfoil gauge creation.



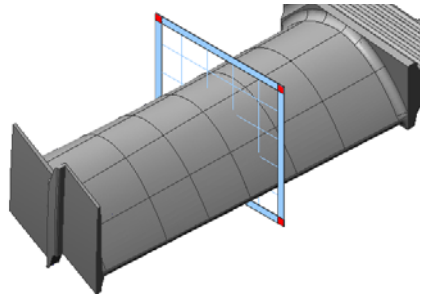
- The measurement zone of an airfoil gauge can be edited post creation by following these steps:

1. Select the airfoil gauge in the Tree View.

Choose *Measure > Gauges > Edit Measurement Zones*.



2. The measurement zone is displayed in the 3D scene and can be edited by dragging the borders.
3. Press ESC to exit the editing mode.



Extracting the geometry of the part

For an existing airfoil gauge, the corresponding geometry of the part can be obtained by extraction on an aligned Data object. In PolyWorks|Inspector, the geometry obtained on the part is called the *measured component* of the measurement object.

The measured component of an airfoil gauge can be automatically extracted at creation when there is an aligned Data object available in the project. Otherwise, a manual extraction can be performed later when the Data object is acquired and properly aligned.

Note that it is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Objective: Finish inspecting the airfoil for an existing airfoil gauge by extracting the gauge's measured component.

Required: An airfoil gauge and a properly aligned Data object representing an airfoil part.

Steps

1. In the Tree View, select the airfoil gauge.



2. Choose *Measure > Extract Measured*.

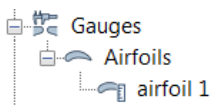


Tip

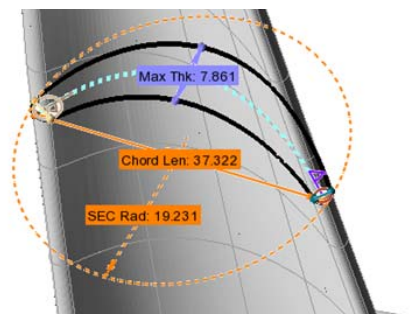
It is possible to view in 2D mode the selected airfoil gauges using the Airfoil Gauge Navigator. To do so, choose *View > Object Navigator > Airfoil Gauges*. For more information, see [Reviewing airfoil gauge measurement results](#) on page 567.

Results

A measured status icon indicates the measured status of the airfoil gauge.



The airfoil gauge is displayed in the 3D scene.



Notes

If the airfoil gauge is created using the wrong family or if the information related to the edges to analyze was specified incorrectly, it is possible to change these parameters and other gauge creation parameters. Select the gauge and choose *Edit > Object Properties*. The object's property sheet opens. Open the **Airfoil** tab and make the desired specifications.

Additional tools



Build/Inspect



Basic measurements



PolyWorks | Talisman™



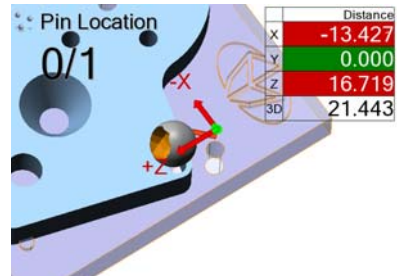
Creating Level Planes

Orient to Gravity

Measuring displacement during real-time assembly

When moving a physical part, it is very useful to track its deviation to stable CAD surfaces or feature(s) to get the part into the closest possible position.

The Build/Inspect mode offers visual and audio guidance for tasks such as lining up to a hole or axis, leveling a part, or securing it in a fixture. Deviations can be measured, in real time, to a center point, axis, or surface. It is possible to measure the probe position to one or to many features simultaneously, using set tolerances for each feature. If necessary, the deviation can be recorded for later reference.



Objective: Move a part into position over a fixed assembly hole using real-time guidance.

Required: A CAD model or feature(s), and a probing device aligned to the Reference object.

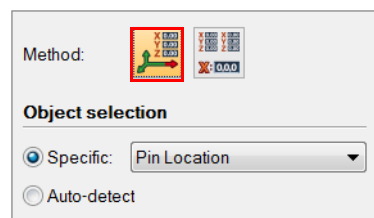
Steps

1. In the Tree View, select a destination feature.
2. Choose *Measure > Build/Inspect*.

The Build/Inspect dialog box and the Build/Inspect Objects pane open.



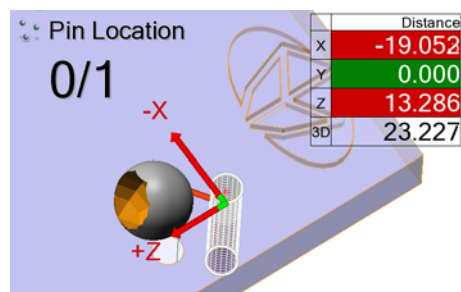
3. In the **Method** list, select **Single Distance** to show guidance to just one feature.
4. Under **Object selection**, select **Specific** to choose which feature to be guided to.



Distance arrows, digital readout values, and audio feedback in the 3D scene indicate the current distance of the probe from the destination feature.

5. Adjust the part, using the guidance, to bring it closer to the destination.

Build/Inspect feedback is displayed in red and green, based on positional tolerances.



- Specify the **Distance Type**, **Compensation**, and **Tolerance** values in the Build/Inspect Objects pane.

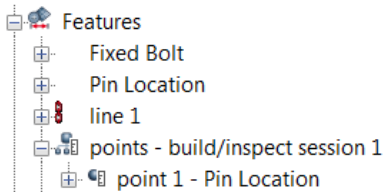
Distance Type	Compensation	Tol	TolX	TolY
Axis	None	XYZ	2.000	100.000

The **Inverted** compensation method can be useful when building or inspecting a fixture and only the CAD model of the part is available.

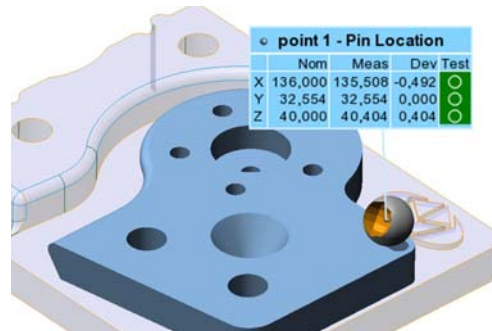
- If desired, acquire a point to record the final position.

Results

If a point is collected, it is added to the Tree View, under the Features branch. A measured status icon indicates the measured status.

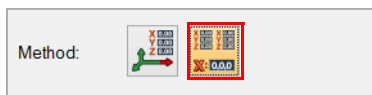


The physical part location is optimized (a representation of the part is shown in blue), and if a point was collected, it is displayed in the 3D scene.



Notes

It is possible to measure several features simultaneously by selecting the **Multiple Distances** method.

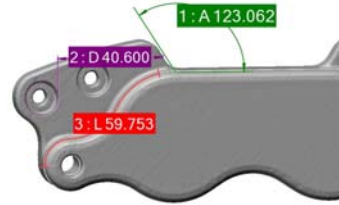


Each feature can be assigned an individual Digital Readout that allows the user to track the probe position with regards to more than one destination.

		Fixed Bolt	Pin Location
3D	X		-0.540
	Y		0.000
	Z		0.752
	3D	64.465	0.926

Taking basic measurements interactively

To quickly measure simple dimensions on a model, cross-section, or feature, use the Basic Measurements tool. Basic measurements include various distances, radii, curve lengths, and angles. Calculations are performed by simply anchoring on the object in the 3D scene. Basic measurements are not updatable, but offer a quick and easy way to get a measurement.



Objective: Measure an angle interactively.

Required: The project contains a Reference and/or Data object.

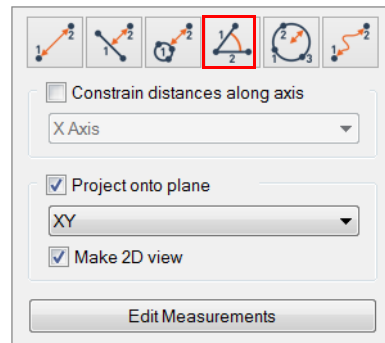
Steps

1. Choose *Measure > Interactive Measurements > Basic*.

The Create Basic Measurements dialog box opens.



2. Specify the **Angle** measurement type to measure a basic angle on the part.
3. If the part is aligned to a coordinate system, select **Project onto plane** and choose a system plane or plane feature to constrain the angle measurement.



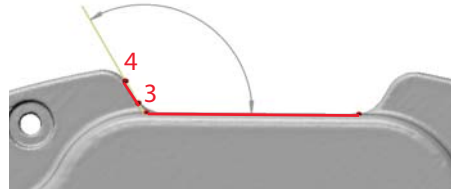
4. In the 3D scene, define one side of the angle by anchoring two or more points on the model.

Press and hold SHIFT to pick the closest vertex.



5. Right-click to confirm the first side.

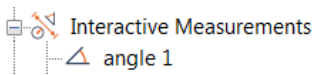
- Define the second side of the angle by anchoring two or more points on the model.
Press and hold SHIFT to pick the closest vertex.



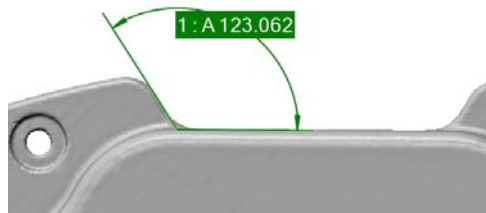
- Right-click to end the measurement.

Results

The angle measurement is added to the Tree View under the Interactive Measurements branch.

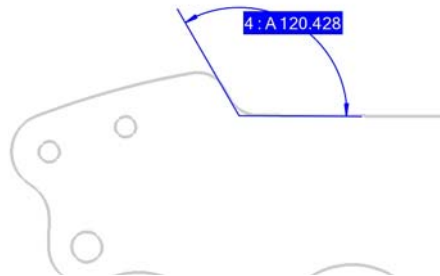


The angle measurement is displayed in the 3D scene.



Notes

It is possible to perform basic measurements on cross-sections and features as well. This can make anchoring more accurate.



Adding quantitative or visual checks

Inspection processes may require at some point quantitative checks using conventional metrology tools such as thread gauges, or visual checks such as the quality or presence of weld spots.

Quantitative checks imply physical measurements and are typically reported in terms of numerical values, while visual checks imply a qualitative test and are typically reported in terms of pass/fail values.



These checks can be added to the inspection project using the custom measurement objects.

Creating custom measurement objects

Objective: Create custom measurement objects to add quantitative or visual checks to the inspection project.

Required: The project contains a Reference and/or Data object.

Steps

1. Choose *Measure > Custom Measurements > Create*.

The Create Custom Measurements dialog box opens.



2. Specify the **Type**:

- **Numerical:** Quantitative checks reported in terms of numerical values.
- **Qualitative:** Visual checks reported in terms of pass/fail values.

3. Choose one of the following creation methods:

Anchor

This method allows anchoring a point interactively at a representative location of the measurement.

1. In the **Method** list, select **Anchor**.
2. Click **Anchor**.

The interactive anchoring mode is launched.

- In the 3D scene, anchor a point at a representative location.



Press and hold SHIFT when anchoring to pick the closest vertex.

- Right-click to exit the anchoring mode.

Numerically

This method allows defining the representative location of the point using specific coordinates.

- In the **Method** list, select **Numerically**.
- Specify the **Position** coordinates to define the location.
- Click **Create**.

Position:

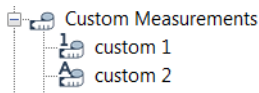
X:

Y:

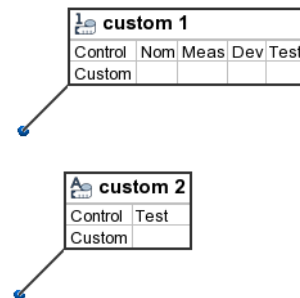
Z:

Results

The custom measurement object is added to the Tree View, under the Custom Measurements branch.



The custom measurement object is displayed in the 3D scene.



Notes

The position and orientation (if any) of the custom measurement object are used during Play Inspection sequences to position and orient the 3D scene.

The next step consists in defining the nominal and measured components for the **Numerical** type, or only the measured component for the **Qualitative** type.

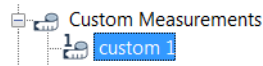
Defining the nominal component for the numerical type

Objective: Define the nominal component of a numerical type custom measurement object.

Required: The project contains a numerical custom measurement object.

Steps

1. In the Tree View, select the numerical custom measurement object for which to define the nominal component.



2. Choose *Measure > Custom Measurements > Define Nominal*.

*Alternatively, right-click the selected object in the Tree View and choose *Define Nominal*.*

The Define Nominal Custom Measurement Components dialog box opens.

3. Enter the value that defines the nominal component of the custom measurement object.

A screenshot of a dialog box titled 'Control'. It contains a label 'Nominal:' followed by an empty text input field.

4. Click **Create**.

Results

The nominal component of the numerical type custom measurement object is defined.

A screenshot of the Tree View showing 'custom 1' selected. A tooltip table is displayed over the selection, showing the configuration for the nominal component.

custom 1				
Control	Nom	Meas	Dev	Test
Custom	10.000			

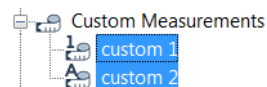
Defining the measured component

Objective: Define the measured component of custom measurement objects.

Required: The project contains a custom measurement object.

Steps

1. In the Tree View, select the custom measurement objects to measure.



2. Choose *Measure > Custom Measurements > Define Measured*.

*Alternatively, right-click the selected objects in the Tree View and choose *Define Measured*.*

The Define Measured Custom Measurement Components dialog box opens. It is contextual to the type of custom measurement object:

Numerical type

1. Enter the **Measured** numerical value that defines the measurement.
2. Click **Create**.

Qualitative type

1. In the **Test** list, select the qualitative value that defines the measurement. The following values are offered:
 - Pass
 - Warning
 - Fail
2. Click **Create**.

Results

The measured component of custom measurement objects is defined.

custom 1				
Control	Nom	Meas	Dev	Test
Custom	10.000	10.200	0.200	○

custom 2	
Control	Test
Custom	○

Notes

- Tolerances can be adjusted using the Geometry Controls pane. See [Editing control tolerances](#) on page 501.

Geometry Controls - custom 1						
Control	Nominal	Measured	Tolerance	Deviation	Test	
Custom	10.000	10.200	±1.000 ▼	0.200	Pass	

- In the PolyWorks Metrology Suite, the nominal and measured components of custom measurements are independent from the units set for the project. This means that if the project units are modified, custom measurement values are not converted.
- It is also possible to measure an object using a scripted measurement method (see [Scripted measurement methods](#) on page 698).

Measuring when away from the workstation

When performing probing operations away from the computer, which is common for laser tracker applications, it may be helpful to have access to functions and operations, like capturing a point or changing the type of feature that will be probed, without having to go back to the computer. It is possible to perform these operations by using Talisman, which is the PolyWorks Metrology Suite mobile application.

Connecting to Talisman

Talisman gives remote control of the PolyWorks Metrology Suite such as: accessing the PolyWorks | Inspector 3D scene and performing 3D transformations; connecting to a probing or laser-scanning measurement device; configuring measurement modes and parameters; and data acquisition.

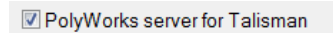
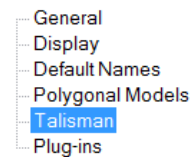
The Talisman application runs on mobile devices and communicates with the PolyWorks Metrology Suite via a Wi-Fi connection.

Objective: Have remote access to a PolyWorks | Inspector project using Talisman.

Required: The workstation and the mobile device must be connected to the same network and a PolyWorks | Inspector project must be open on the workstation.

Steps

1. In the Workspace Manager, choose *Tools > Options*.
2. In the Workspace Manager Options dialog box, open the **Talisman** page.
3. Select the **PolyWorks server for Talisman** parameter.
This enables communication between Talisman and the PolyWorks Metrology Suite. Unless the user configuration is reset, this only needs to be done once.
4. Click **OK**.
5. Start the Talisman application on the mobile device.
6. Within the Talisman application, select the project in order to view and perform desired operations.



Creating level planes

Some laser trackers have an electronic level that calculates the inclination of the tracker with respect to the axis of gravity. PolyWorks | Inspector allows the user to obtain the necessary information from the tracker's level to create a measured plane primitive. The plane is oriented with respect to gravity, which results in a plane that is perfectly horizontal.

Objective: Create a level plane.

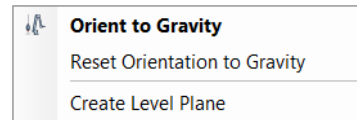
Required: A laser tracker with an electronic level.

Steps

1. Connect to the laser tracker in PolyWorks | Inspector.
2. Make sure that the laser tracker is initialized.
3. On the **Probing Device Control** toolbar, click the arrow next to the **Orient to Gravity** button.
4. Choose **Create Level Plane**.

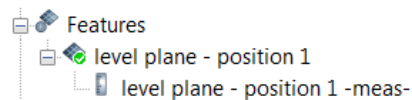


The laser tracker performs a series of movements from which it will output a plane feature that is level.



Results

The plane is added to the Tree View, under the **Features** branch.



Notes

Once the plane is created, a new coordinate system can be defined using this level plane by choosing *Tools > Coordinate Systems > Create Cartesian*.

Orienting to gravity

When laser trackers have an electronic level, it is possible to orient them to gravity. The level inside the tracker computes the inclination of the tracker with respect to the axis of gravity so that acquired points are compensated by taking into account the computed inclination.

Objective: Orient the laser tracker to gravity.

Required: A laser tracker with an electronic level.

Steps

1. Connect to the laser tracker in PolyWorks | Inspector.
2. Make sure that the laser tracker is initialized.
3. On the **Probing Device Control** toolbar, click the **Orient to Gravity** button.



The laser tracker performs a series of movements from which it measures its inclination angles.

Results

A plumb bob icon appears to the left of the active device position to indicate that the laser tracker is oriented to gravity.



Geometry controls



Dimensional and GD&T controls

Controlling measurements using geometry controls

When inspecting using measurement objects, namely features, cross-sections, comparison points, and gauges, the main purpose is to control whether the geometric characteristics of the objects are within the specified tolerance. This is done by adding controls on the geometric characteristics of measurement objects. These geometry controls can be of two types.

Dimensional controls – All measurement objects can have dimensional controls. These controls are used to control, using a tolerance, various geometric characteristics of measurement objects, like the diameter of a circle, the 3D distance of a comparison point, or the flush value of a flush & gap gauge. Each dimensional control reports a measured value and a deviation, is toleranced, and can report a nominal value.

GD&T controls (features only) – Only feature objects can have GD&T controls. These special controls are used to control precise geometrical dimensions, as specified by the ASME Y14.5 and ISO standards.

PolyWorks | Inspector offers the Geometry Controls pane that is the unique access through which the controls of all measurement objects are managed.

The Geometry Controls pane allows doing the following for a measurement object:

- View all dimensions and review the nominal, measured, and deviation values; note that distance controls and most GD&T controls typically only have a measured value, which is also the deviation.
- Choose the dimensions to tolerance (become dimensional controls).
- Create GD&T controls for features, as defined by the specified GD&T standard.
- Edit the tolerance for the controls.

By default, new controls are displayed in the annotation of features in the 3D scene, and are included in report tables created from all measurement objects.

To review and analyze the results of the geometry controls, it is recommended that the Control Reviewer tool is used. For information, see [Reviewing measurement results using the Control Reviewer](#) on page 533.

The Geometry Controls pane

To open the Geometry Controls pane:

1. Choose *Measure > Geometry Controls*.

Alternatively, click the **Geometry Controls** button on the Main Objects toolbar.



The Geometry Controls pane is displayed.

Column headers

Measurement context

best-fit to ref 1 Best-fit On

Control	Nominal	Measured	Tolerance	Deviation	Test
∅ 25.397±1.000 mating	25.397	25.395	±1.000	-0.002	Pass
∅ 1.000 A B C		0.078	1.000	0.078	Pass
Diameter	25.397	25.591	±1.000	0.194	Pass
X	70.999	71.114	±1.000	0.115	Pass
Y	20.000	20.000	±1.000	0.000	Pass
Z	40.000	40.066	±1.000	0.066	Pass

List of controls

Contextual toolbar

It offers the list of controls for the selected measurement object, a contextual toolbar, column headers, and a measurement context header.

If no measurement object is selected in the Tree View, the list of controls appears empty and the buttons in the contextual toolbar are grayed out.

Column headers

Identify the values contained in the list of controls. It is possible to show/hide columns by right-clicking the column header and selecting the appropriate item. Columns can also be reordered by dragging them to a new location.

List of controls

Displays controls for the selected measurement object in the Tree View.

Measurement context

Displayed when the selected object in the Tree View is a cross-section or a cross-section feature. Two uneditable text boxes are displayed, offering the following content:

- Data Alignment: displays the name of the data alignment to which the cross-section is bound (or the linked cross-section in the case of cross-section features).
- Best-Fit Activation State: displays the state of the best-fit alignment for the selected cross-section (or the linked cross-section in the case of cross-section features).

Contextual toolbar

It offers creation and editing functionalities. The toolbar varies depending on the measurement objects selected in the Tree View, as buttons can be missing or disabled. Some buttons display a dialog box, while others display a shortcut menu. Refer to the following section for more information.

The contextual toolbar

Different functions are offered, depending on the context:



Add/Remove Dimensional Controls

Available for all measurement objects. It allows activating or deactivating the mode for adding/removing dimensional controls.



Add GD&T Control

Available only when the selected measurement object is a feature. It displays the list of GD&T tools that can be added to the selected feature, according to the specified GD&T standard, along with their respective symbols.



Set Controls using Template

It displays a shortcut menu that provides a list of the available control templates for the selected measurement object in the Tree View. By default, existing GD&T controls are preserved.



Edit Controls

It either displays the Edit Controls dialog box, when selecting a dimensional control, or the editing window, when selecting a GD&T control. A multiple selection can be performed.



Remove Controls

It removes the selected controls. A multiple selection can be performed.



Set as Default

It saves the current dimensional control specifications as a control template and sets it as the default template for the selected measurement object type.



Save as Control Template

It saves the current dimensional control specifications as a control template for the selected measurement object type.

Using the Geometry Controls

Adding and removing controls

The Geometry Controls pane allows viewing all dimensions (for example, radius, diameter, or X, Y, Z coordinates) and choosing which one will be used as a control. Check boxes and a different background color indicate whether each dimension is effectively controlled or not.

Objective: Specify dimensions to control for a measurement object.

Required: A measurement object.

Steps

1. In the Tree View, select a measurement object.

Use a multiple selection to specify controls for same-type measurement objects simultaneously.



2. Choose *Measure > Geometry Controls*.

The Geometry Controls pane opens.



3. In the pane, click **Add/Remove Dimensional Controls** to display all available dimensional controls for the feature type.



4. Select the desired controls in the expanded list.

Their current results are displayed in the pane.

	Control	Nominal	Measured
<input checked="" type="checkbox"/>	Diameter	25.400	25.553
<input checked="" type="checkbox"/>	X	71.000	71.032
<input checked="" type="checkbox"/>	Y	25.400	25.399
<input checked="" type="checkbox"/>	Z	40.000	40.038
<input type="checkbox"/>	Radius	12.700	12.776
<input type="checkbox"/>	3D Distance		0.050
<input type="checkbox"/>	Normal Dista...		-0.001

5. Click **Add/Remove Dimensional Controls** again to show only the effective controls.

Results

The added controls, and their current results, are displayed in the Geometry Controls pane.

Control	Nominal	Measured	Tolerance	Deviation	Test
Diameter	25.400	25.553	±1.000 ▼	0.153	Pass
Radius	12.700	12.776	±1.000 ▼	0.076	Pass
X	71.000	71.032	±1.000 ▼	0.032	Pass
Y	25.400	25.399	±1.000 ▼	-0.001	Pass
Z	40.000	40.038	±1.000 ▼	0.038	Pass

Notes

For features and gauges, if the nominal component is missing, its value can be entered directly in the cell under the **Nominal** column. The same applies for the **Length**, **Perimeter**, and **Area** dimensions of cross-sections.

The Geometry Controls pane is also used to add and encode Geometric Dimensioning and Tolerancing (GD&T) controls. See [Adding and encoding GD&T controls](#) on page 508 for more information.

Editing control tolerances

A tolerance, consisting of upper and lower tolerance limits, is assigned to each individual control. Values can be edited manually, or be changed by assigning a tolerance template that contains predefined tolerance limits.

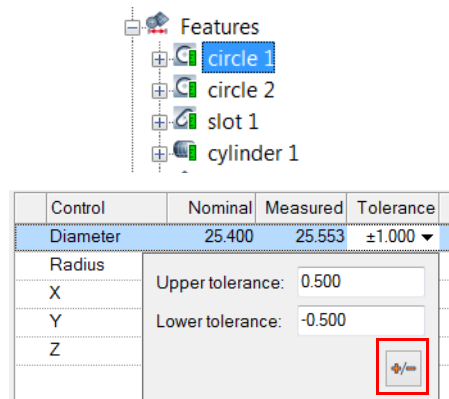
Standard tolerance templates are provided. However, new tolerance templates can be created on the **Objects > Tolerances** page of the PolyWorks|Inspector Options dialog box. Choose *Tools > Options* to open the options dialog box.

Objective: Edit the tolerance of a geometry control.

Required: A measurement object.

Steps

- In the Tree View, select a measurement object.
Use a multiple selection to specify tolerances for same-type measurement objects simultaneously.
- Click the arrow in the **Tolerance** field.
- Edit the tolerance limits using one of the following methods:
 - Manually: Enter tolerance limit values in the dialog box displayed.
 - Using a template: Click the **Assign Tolerance** button in the dialog box displayed and select a tolerance template.



Results

The new tolerance values are displayed in the pane.

Control	Nominal	Measured	Tolerance
Diameter	25.400	25.553	±0.500 ▼
Radius	12.700	12.776	±1.000 ▼
X	71.000	71.032	±1.000 ▼
Y	25.400	25.399	±1.000 ▼

Notes

Tolerances can be edited for multiple controls simultaneously by selecting the controls in the pane and clicking **Edit Controls**.



Using control templates

When a measurement object is created, its controls and their tolerance are specified by the object's default control template. In the Geometry Controls pane, new controls can be added and different tolerances can be specified for a measurement object. It is possible to save the current specifications as a new user control template for that type of object, which can then be used to set the controls of other objects of the same type. It can also be designated as the default control template for the particular type of object.

Control templates provided with the PolyWorks Metrology Suite are called standard templates and have the STD prefix in template lists, while control templates created by users have theUSR prefix. Standard templates cannot be deleted.

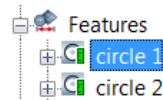
Saving a control template

Objective: Save a control template from the current controls of a measurement object.

Required: A measurement object.

Steps

1. In the Tree View, select a measurement object.



2. Choose *Measure > Geometry Controls*.

The Geometry Controls pane opens.



3. In the pane, modify the controls if required.
See [Adding and removing controls](#) on page 500 and [Editing control tolerances](#) on page 501.

- Click **Save as Control Template**.

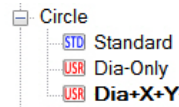


To save as the default control template for the type of measurement object, click **Set as Default**.

- Specify the name of the new control template and click **Save**.

Results

A user control template is saved for the type of measurement object. It can be used to set the contents of other measurement objects of the same type. The default template is in bold.



Setting controls using a template

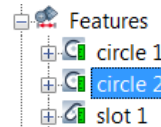
Objective: Set controls for a measurement object using a control template.

Required: A measurement object.

Steps

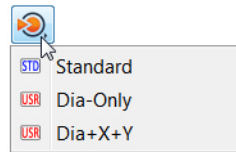
- In the Tree View, select a measurement object.

Use a multiple selection to set the controls for same-type measurement objects simultaneously.



- In the Geometry Controls pane, click **Set Controls using Template**.

- Choose the template.



Results

The control template is used to set the controls of the selected measurement object.

Managing control templates

The template editor allows personalizing control templates, either to create a new control template, set a control template as the default for a measurement object type, edit the contents of a user control template (USR) or delete it from the list of templates.

Managing existing templates

Objective: For a type of measurement object, delete a user template from the list of templates and change the default control template.

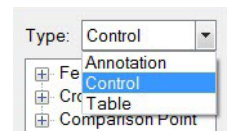
Required: More than two control templates for the measurement object type.

Steps

1. Choose *Tools > Edit Templates*.

The Edit Templates dialog box opens.

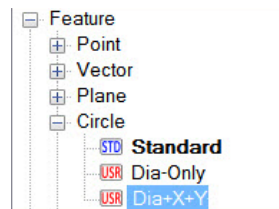
2. In the **Type** list, select **Control**.



3. Browse for the desired object type.

4. Select the control template to designate as the default control template.

The control template Dia+X+Y is selected as an example.



5. To make the selected template the default template, click **Set as default**.



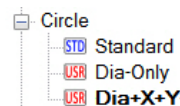
6. To delete an existing user template (USR), select it and click **Delete**.



Results

The new default control template is indicated in bold.

New circles will use the new default control template.



Creating and editing a user control template

Objective: For a type of measurement object, create a new user template and edit its contents.

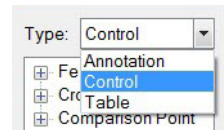
Required: PolyWorks | Inspector is open.

Steps

1. Choose *Tools > Edit Templates*.

The Edit Templates dialog box opens.

2. In the **Type** list, select **Control**.



3. Browse for the desired object type.

4. To create a new user template, click **Create**.

The Create User Template dialog box opens.

5. Specify the **Name** of the new user template.

6. Specify the **Base Template** for the new user template.

If a template is selected upon creation, it will be preselected as the base template by default.

7. Click **Create**.

*The newly created user template can be set as default by selecting the **Set as default** parameter.*

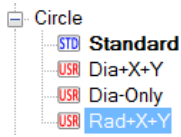
8. Edit the content of the newly created control template, directly in the pane to the right, by selecting the desired controls and changing the tolerance values for the controls.

9. Click **Apply**.

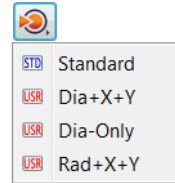
Control	Tolerance
<input checked="" type="checkbox"/> Radius	±1.000 ▾
<input checked="" type="checkbox"/> X	±1.000 ▾
<input checked="" type="checkbox"/> Y	±1.000 ▾
<input type="checkbox"/> ∅ Mating Diameter	±1.000 ▾
<input type="checkbox"/> Position	1.000 ▾

Results

The new user control template is added to the list of templates.



It can now be assigned to measurement objects.



Notes

Edit the content of a user control template at any time by selecting it in the list, making the required modifications in the pane, then clicking **Apply**.

Controlling features

When inspecting using features, the geometric characteristics of the objects can be controlled with two types of controls: dimensional and GD&T.

- **Dimensional controls:** They allow controlling, using tolerances, all characteristics of a feature relating to size, location, and orientation, such as radius, origin, length, and angle. When nominal and measured values are present for a dimensional control, a deviation is calculated, and using the specified tolerance, a Pass/Fail status can be determined.
- **GD&T controls:** These controls are used to control precise geometrical dimensions, as specified by the ASME Y14.5 and ISO standards. The measurement envelope, datum constraints, measured results, and Pass/Fail statuses are displayed for each encoded control.

Adding dimensional controls

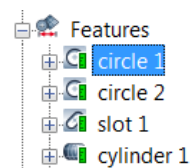
Objective: Add dimensional controls to a feature.

Required: A feature measurement object.

Steps

1. In the Tree View, select a feature.

Use a multiple selection to modify same-type features simultaneously.



- Choose *Measure > Geometry Controls*.

The Geometry Controls pane opens.



- In the pane, click **Add/Remove Dimensional Controls** to display all available dimensional controls for the feature type.



- Select each desired control in the expanded list.

Their current results are displayed in the pane.

- Click **Add/Remove Dimensional Controls** again to show only the effective controls.

	Control	Nominal	Measured
<input checked="" type="checkbox"/>	Diameter	25.400	25.553
<input checked="" type="checkbox"/>	X	71.000	71.032
<input checked="" type="checkbox"/>	Y	25.400	25.399
<input checked="" type="checkbox"/>	Z	40.000	40.038
<input type="checkbox"/>	Radius	12.700	12.776
<input type="checkbox"/>	3D Distance		0.050
<input type="checkbox"/>	Normal Dista...		-0.001

Results

The added controls and their current results are displayed in the Geometry Controls pane.

Control	Nominal	Measured	Tolerance	Deviation	Test
Diameter	25.400	25.553	±1.000 ▼	0.153	Pass
Radius	12.700	12.776	±1.000 ▼	0.076	Pass
X	71.000	71.032	±1.000 ▼	0.032	Pass
Y	25.400	25.399	±1.000 ▼	-0.001	Pass
Z	40.000	40.038	±1.000 ▼	0.038	Pass

Notes

If a nominal feature component exists, the nominal values are displayed in the **Nominal** column. If the component is missing, its nominal values can be entered directly in the fields.

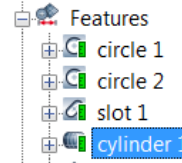
Adding and encoding GD&T controls

Objective: Add and encode a GD&T control.

Required: A feature measurement object.

Steps

1. In the Tree View, select a feature.



2. Choose *Measure > Geometry Controls*.

The Geometry Controls pane opens.

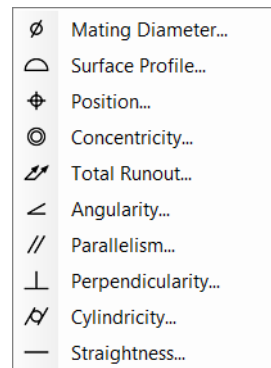


3. In the pane, click **Add GD&T Control**.

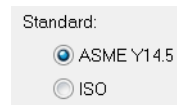


4. In the list, select the desired GD&T control.

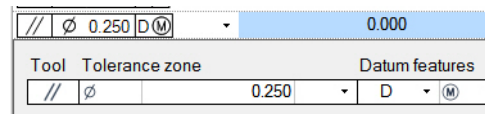
Only GD&T controls appropriate for the selected feature type, and the specified standard, are displayed. In this example, the ASME Y14.5 GD&T standard is specified.



*The GD&T standard can be specified on the **Objects > Features > GD&T** page of the PolyWorks | Inspector options.*



5. Enter the required information to define the GD&T control. The items offered depend on the specified GD&T tool and standard.



Datum Features must be defined before using them for encoding.

6. Press the ENTER key to confirm.

Results

The GD&T control is displayed in the Geometry Controls pane. If the feature contains a nominal and a measured component, the deviation is automatically calculated.

Control	Nominal	Measured	Tolerance	Deviati...	Test
∅ 25.400±1.000 mating	25.400	25.233	±1.000	-0.167	Pass
⊕ ∅ 0.200 A B C		0.226	0.250	0.226	Pass
⊕ ∅ 1.000 A B		0.090	0.150	0.090	Pass
∥ ∅ 0.250 D (M)		0.000	0.250	0.000	Pass
Diameter	25.400	25.553	±1.000	0.153	Pass

Notes

When a GD&T control contains information (for example, an error or a warning), feedback is provided with different symbols in the **Control** column. Hovering the mouse over the symbol displays information about an invalid or incomplete GD&T encoding.

⊕ ∅ 1.000 A	1.000 (B: 0.000)	0.000	0.000	0.000	Pass
⊕ ∅ 1.000	0.162 1.000 (B: 0.0...	0.162	0.250 (B: 0.0...	0.000	0.0

Defining datum features

Objective: Define a feature as a datum for use in GD&T controls.

Required: A feature.

Steps

1. In the Tree View, select a feature.
2. Choose *Edit > Object Properties*.

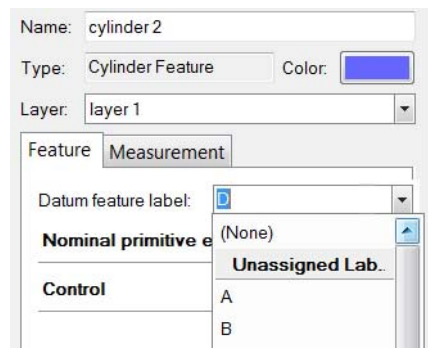
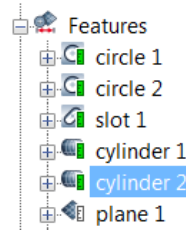
The Properties dialog box opens.

The property sheet of a feature can also be accessed by double-clicking on it.

3. Click the **Feature** tab.
4. Select a datum label in the **Datum feature label** list.

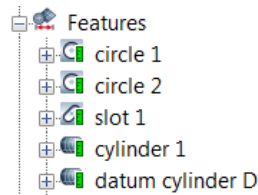
Labels that are already used are not available in the list. Custom datum labels can be created by entering them manually.

5. Click **Apply**.



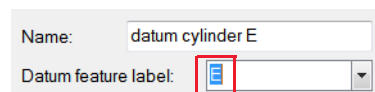
Results

The datum label is added to the feature, and the feature is renamed by default.



Notes

Datum labels can also be assigned to features during feature creation. While in the feature creation interface, select a datum label in the **Datum feature label** list.



Defining datum targets

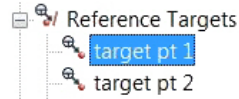
Objective: Define reference targets as a datum target for use in GD&T controls.

Required: A reference target point, line or area.

Steps

1. In the Tree View, select a reference target.

Surface reference target points, reference target lines and reference target areas can be used as datum targets.



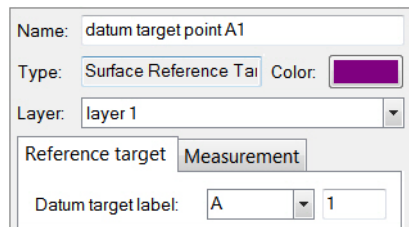
2. Choose *Edit > Object Properties*.

Alternatively, right-click the selected reference target in the Tree View and choose *Properties*.

The Properties dialog box opens.

3. On the **Reference target** tab, select a datum label in the **Datum target label** list.

Labels that are already used are not available in the list. Custom datum labels can be created by entering them manually.

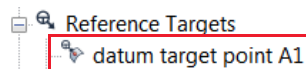


4. A default index value, starting at 1, is automatically populated in the index field to the right. If desired, manually enter a custom index number.

5. Click **Apply**.

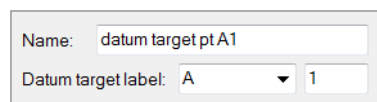
Results

The datum label is added to the reference target and is renamed by default.



Notes

Datum labels can also be assigned to reference targets on creation. While in the creation interface, select a datum label in the **Datum target label** list.



When used in the Datum Reference Frame of a GD&T control, all index numbers of the specified datum are used. For example, if reference targets A1, A2, and A3 exist, and Datum A is specified in the Datum Reference Frame, all three reference targets are used.

Defining composite datums

Objective: Create a composite datum from two or more datum features for use in GD&T controls.

Required: Two or more datums of the same feature type.

Steps

1. Choose *Measure > Features > Composite Datum Features*.

The Composite Datum Features dialog box opens.

In the **Creation** section:

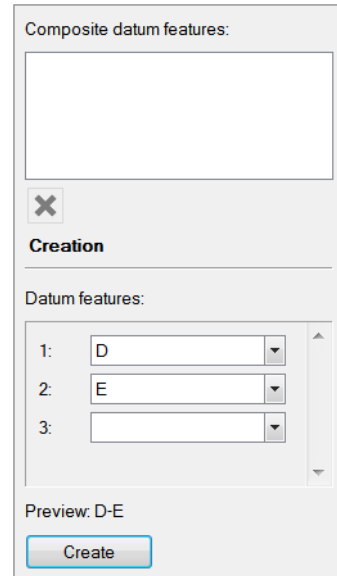
2. Select the first datum feature in the first list.

3. Select a second datum feature of the same feature type in the second list, which appears after specifying the first one.

Only usable datum features, such as datum features of the same feature type, are displayed in the list.

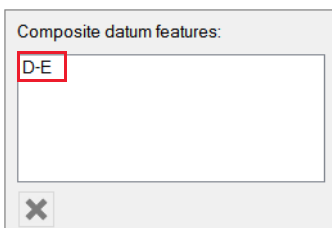
4. Continue until the composite datum contains all the desired datum features.

5. Click **Create**.

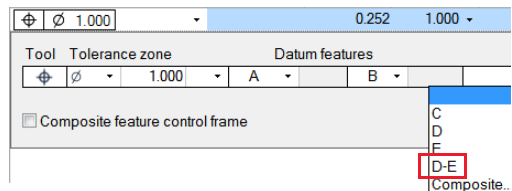


Results

The composite datum feature appears in the **Composite datum features** list.



The composite datum feature is now available for use in GD&T controls.



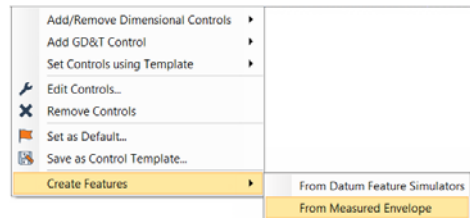
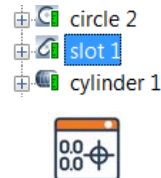
Creating features from measurement envelopes

Objective: Create features representing the GD&T measurement envelope.

Required: A valid and measured GD&T control.

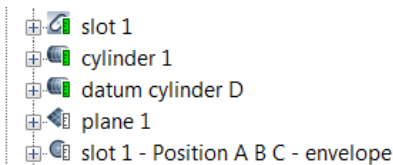
Steps

1. In the Tree View, select a feature.
2. Choose *Measure > Geometry Controls*.
The Geometry Controls pane opens.
3. In the pane, select a GD&T control.
4. Right-click in any field and choose *Create Features > From Measured Envelope*.

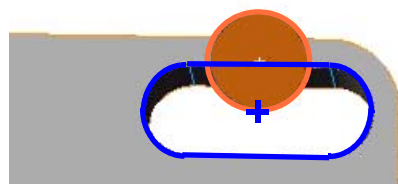


Results

A new feature is added to the Tree View that represents the shape and size of the measurement envelope.



The measured envelope is displayed in the 3D scene at the location of the measured slot with respect to the DRF alignment.

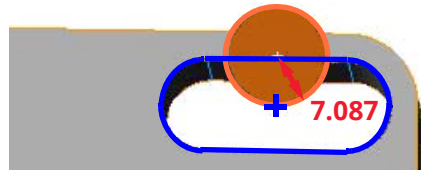


Notes

The measurement envelope feature describes the measured value of the GD&T control, the envelope shape, and the measured location of the feature as constrained by the Datum Reference Frame.

Geometry Controls - slot 1

Control	Nom	Meas	Tol	De
M D A B D-E		7.087	1.000	7.0



Controlling cross-sections

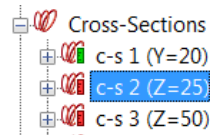
The Geometry Controls pane can be used to specify toleranced dimensional controls for a cross-section. Dimensional controls allow controlling, using tolerances, the characteristics of a cross-section relating to size, form, and deviation such as Min and Max deviations, line profile, length, and area. Results can be obtained when the nominal and measured components are present. For a control, typically the nominal and measured values are used to obtain a deviation and, using the tolerance and deviation, a Pass/Fail status is determined.

Objective: Specify controls for a cross-section.

Required: A cross-section.

Steps

- In the Tree View, select a cross-section.
Use a multiple selection to modify cross-sections simultaneously.



- Choose *Measure > Geometry Controls*.

The Geometry Controls pane opens.



- In the pane, click **Add/Remove Dimensional Controls** to display all available dimensional controls for the object type.



- Select the desired controls in the expanded list.

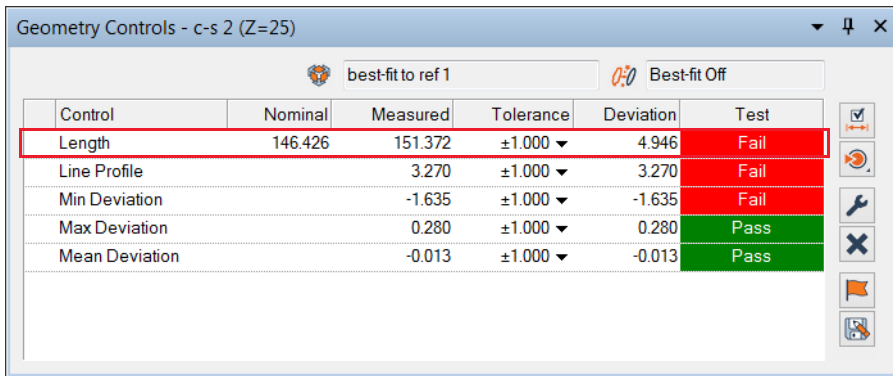
Current values are displayed in the pane.

- Click **Add/Remove Dimensional Controls** again to show only the effective controls.

	Control	Nominal	Measured
<input checked="" type="checkbox"/>	Line Profile		3.270
<input checked="" type="checkbox"/>	Min Deviation		-1.635
<input checked="" type="checkbox"/>	Max Deviation		0.280
<input checked="" type="checkbox"/>	Mean Deviation		-0.013
<input type="checkbox"/>	Length	146.426	151.372
<input type="checkbox"/>	Perimeter	146.426	

Results

The added controls and their current results are displayed in the Geometry Controls pane.



Control	Nominal	Measured	Tolerance	Deviation	Test
Length	146.426	151.372	±1.000	4.946	Fail
Line Profile		3.270	±1.000	3.270	Fail
Min Deviation		-1.635	±1.000	-1.635	Fail
Max Deviation		0.280	±1.000	0.280	Pass
Mean Deviation		-0.013	±1.000	-0.013	Pass

Controlling comparison points

The Geometry Controls pane can be used to specify toleranced dimensional controls for a comparison point. Dimensional controls allow controlling, using tolerances, the characteristics of a comparison point that relate to its location in space. When the nominal and measured components are present, results can be obtained. For a control, typically the nominal and measured values are used to obtain a deviation and, using the tolerance and deviation, a Pass/Fail status is determined.

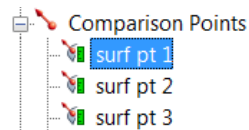
Objective: Specify controls for a comparison point.

Required: A comparison point.

Steps

1. In the Tree View, select a comparison point.

Use a multiple selection to modify comparison points simultaneously.



2. Choose *Measure > Geometry Controls*.

The Geometry Controls pane opens.



3. In the pane, click **Add/Remove Dimensional Controls** to display all available dimensional controls for the object type.



- Select the desired controls in the expanded list.

Current values are displayed in the pane for the new controls.

- Click **Add/Remove Dimensional Controls** again to show only the effective controls.

	Control	Nominal	Measured
<input checked="" type="checkbox"/>	Surface Distance		-0.051
<input type="checkbox"/>	Surface Point X	51.295	51.295
<input type="checkbox"/>	Surface Point Y	25.400	25.349
<input type="checkbox"/>	Surface Point Z	47.172	47.172
<input type="checkbox"/>	3D Distance		-0.051
<input type="checkbox"/>	3D Point X	51.295	51.295
<input type="checkbox"/>	3D Point Y	25.400	25.349
<input type="checkbox"/>	3D Point Z	47.172	47.172

Results

The added controls, and their current results, are displayed in the Geometry Controls pane.

Control	Nominal	Measured	Tolerance	Deviation	Test
Surface Distance		-0.051	±1.000	-0.051	Pass
3D Distance		-0.051	±1.000	-0.051	Pass

Controlling calipers

The Geometry Controls pane can be used to specify toleranced dimensional controls for a caliper. Dimensional controls allow controlling, using tolerances, the characteristics of a caliper that relate to its length and location in space. When the nominal and measured components are present, results can be obtained. For a control, typically the nominal and measured values are used to obtain a deviation and, using the tolerance and deviation, a Pass/Fail status is determined.

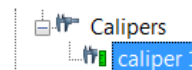
Objective: Specify controls for a caliper.

Required: A caliper.

Steps

- In the Tree View, select a caliper.

Use a multiple selection to modify calipers simultaneously.



- Choose *Measure > Geometry Controls*.

The Geometry Controls pane opens.



- In the pane, click **Add/Remove Dimensional Controls** to display all available dimensional controls for the object type.



- Select the desired controls in the expanded list.

Current values are displayed in the pane for the new controls.

	Control	Nominal	Measured
<input checked="" type="checkbox"/>	Length	26.825	26.773
<input type="checkbox"/>	1st Tip Endpoint 3D ...		0.180
<input type="checkbox"/>	1st Tip Endpoint X	74.128	74.128
<input type="checkbox"/>	1st Tip Endpoint Y	20.056	20.056
<input type="checkbox"/>	1st Tip Endpoint Z	8.000	8.180
<input type="checkbox"/>	2nd Tip Endpoint 3D ...		0.128

- Click **Add/Remove Dimensional Controls** again to show only the effective controls.

Results

The added controls, and their current results, are displayed in the Geometry Controls pane.

Control	Nominal	Measured	Tolerance	Deviation	Test
Length	26.825	26.773	±1.000	-0.052	Pass
1st Tip Endpoint 3D Di...		0.180	±1.000	0.180	Pass
2nd Tip Endpoint 3D Di...		0.128	±1.000	0.128	Pass

Controlling profile gauges

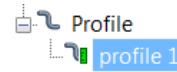
The Geometry Controls pane can be used to specify toleranced dimensional controls for a profile gauge. Dimensional controls allow controlling, using tolerances, the characteristics of a profile gauge that relate to its size and deviation from the nominal component. The exact controls available depend on the profile family type. For a control, typically the nominal and measured values are used to obtain a deviation and, using the tolerance and deviation, a Pass/Fail status is determined.

Objective: Specify controls for a profile gauge.

Required: A profile gauge.

Steps

- In the Tree View, select a profile gauge.
Use a multiple selection to modify profile gauges simultaneously.
- Choose *Measure > Geometry Controls*.
The Geometry Controls pane opens.
- In the pane, click **Add/Remove Dimensional Controls** to display all available dimensional controls for the object type.
- Select the desired controls in the expanded list.
Current values are displayed in the pane for the new controls.
- Click **Add/Remove Dimensional Controls** again to show only the effective controls.



	Control	Nominal	Measured
<input checked="" type="checkbox"/>	A Rad	3.179	3.255
<input type="checkbox"/>	A1 Normal Dist		-0.022
<input type="checkbox"/>	A2 Normal Dist		-0.034
<input type="checkbox"/>	AE Dist		0.256
<input type="checkbox"/>	A Highest Pt Dist		1.474

Results

The added controls, and their current results, are displayed in the Geometry Controls pane.

Control	Nominal	Measured	Tolerance	Deviation	Test
A Rad	3.179	3.255	±1.000	0.076	Pass
AE Dist		0.256	±1.000	0.256	Pass

Controlling flush & gap gauges

The Geometry Controls pane can be used to specify toleranced dimensional controls for a flush & gap gauge. Dimensional controls allow controlling, using tolerances, the characteristics of a flush & gap gauge that relate to its size and deviation from the nominal component. The exact controls available depend on the flush & gap family and name. For a control, typically the nominal and measured values are used to obtain a deviation and, using the tolerance and deviation, a Pass/Fail status is determined.

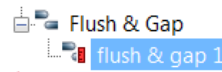
Objective: Specify controls for a flush & gap gauge.

Required: A flush & gap gauge.

Steps

1. In the Tree View, select a flush & gap gauge.

Use a multiple selection to modify flush & gap gauges simultaneously.



2. Choose *Measure > Geometry Controls*.

The Geometry Controls pane opens.



3. In the pane, click **Add/Remove Dimensional Controls** to display all available dimensional controls for the object type.



4. Select the desired controls in the expanded list.

Current values are displayed in the pane for the new controls.

	Control	Nominal	Measured
<input checked="" type="checkbox"/>	Flush	0.007	-0.190
<input checked="" type="checkbox"/>	Gap	6.473	5.230
<input type="checkbox"/>	A Lat Dist		0.482
<input type="checkbox"/>	A Normal Dist		0.397
<input type="checkbox"/>	B Lat Dist		0.757

5. Click **Add/Remove Dimensional Controls** again to show only the effective controls.

Results

The added controls, and their current results, are displayed in the Geometry Controls pane.

Control	Nominal	Measured	Tolerance	Deviation	Test
Flush	0.007	-0.190	±1.000	-0.197	Pass
Gap	6.473	5.230	±1.000	-1.243	Fail
A Lat Dist		0.482	±1.000	0.482	Pass
B Lat Dist		0.757	±1.000	0.757	Pass

Controlling airfoil gauges

The Geometry Controls pane can be used to specify toleranced dimensional controls for an airfoil gauge. Dimensional controls allow controlling, using tolerances, the characteristics of an airfoil gauge that relate to its size, form, and deviation from the nominal component. The exact controls available depend on the airfoil family. For a control, typically the nominal and measured values are used to obtain a deviation and, using the tolerance and deviation, a Pass/Fail status is determined.

Objective: Specify controls for an airfoil gauge.

Required: An airfoil gauge.

Steps

1. In the Tree View, select an airfoil gauge.

Use a multiple selection to modify airfoil gauges simultaneously.



2. Choose *Measure > Geometry Controls*.

The Geometry Controls pane opens.



3. In the pane, click **Add/Remove Dimensional Controls** to display all available dimensional controls for the object type.



- Select the desired controls from the expanded list.

Current values are displayed in the pane for the new controls.

- Click **Add/Remove Dimensional Controls** again to show only the effective controls.

	Control	Nominal	Measured
<input checked="" type="checkbox"/>	Max Width	41.117	41.204
<input checked="" type="checkbox"/>	Stagger	-8.631	-8.911
<input checked="" type="checkbox"/>	Max Thk	8.113	8.130
<input checked="" type="checkbox"/>	LE Thk 1	3.139	3.194
<input checked="" type="checkbox"/>	TE Thk 1	2.216	2.112
<input type="checkbox"/>	CL Len	48.093	48.052
<input type="checkbox"/>	Chord Len	40.085	40.146
<input type="checkbox"/>	Area	274.968	272.275

Results

The added controls and their current results are displayed in the Geometry Controls pane.

Control	Nominal	Measured	Tolerance	Deviation	Test
Chord Len	40.085	40.146	±1.000 ▼	0.061	Pass
Max Width	41.117	41.204	±1.000 ▼	0.087	Pass
Stagger	-8.631	-8.911	±1.000 ▼	-0.280	Pass
Max Thk	8.113	8.130	±1.000 ▼	0.017	Pass
LE Thk 1	3.139	3.194	±1.000 ▼	0.055	Pass
TE Thk 1	2.216	2.112	±1.000 ▼	-0.104	Pass

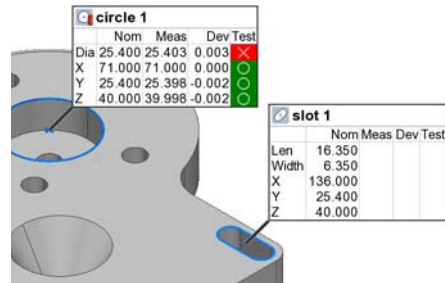
Reporting measurement results



Annotations

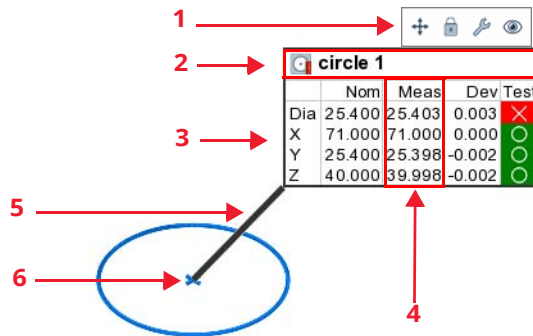
Annotations are graphical notes displayed in the 3D scene in order to identify objects and provide information about them. By default, each object shown in the 3D scene has an annotation attached to it.

Annotations differ slightly from one type of object to another.



Annotation overview

Annotations are typically in a table format. Here is a typical example of an annotation:



- | | | |
|---|-------------------------|--|
| 1 | Annotation mini toolbar | A toolbar displayed when hovering the mouse over an annotation. For more information, see The Annotation mini toolbar on page 525. |
| 2 | Header | Displays the object name, preceded by an icon that represents the object type and the measurement status of the object (measured or unmeasured). |
| 3 | Lines | List all the object controls, one per line. |
| 4 | Columns | Present attributes of the controls, such as the nominal, measured and deviation values, as well as the pass/fail test. |
| 5 | Attachment line | A line that links the annotation to the corresponding object. |
| 6 | Attachment point | The location where the annotation is attached to the object. |

The Annotation mini toolbar

The annotation mini toolbar is displayed when hovering the mouse over an annotation. It offers the following buttons:



Position Annotations



Enables or disables the interactive mode that allows repositioning annotations in the 3D scene. For more information, see [Repositioning annotations in the 3D scene](#) on page 526.

Lock/Unlock Annotation Position



Locks or unlocks the position of the current annotation. When the annotation is locked, its position remains unchanged with respect to the object. Its repositioning is only possible using the Position Annotations tool.

Edit Annotation



Opens a dialog box that allows specifying the type of information that will be displayed in the current annotation or modifying the appearance of this annotation. For more information, see [Editing an object annotation](#) on page 530.

Hide Annotation



Hides the current annotation. Hover the mouse over the attachment point of the annotation to display a button that allows displaying the annotation.

Repositioning annotations in the 3D scene

To maximize the clarity of report snapshots, it is possible to reposition annotations in the 3D scene prior to taking a snapshot. This operation can be made for a specific annotation or for all annotations simultaneously.

Objective: Reposition object annotations in the 3D scene.

Required: The project contains an object with a visible annotation.

Steps

1. Choose *Tools > Annotations > Position*.



Alternatively, mouse over the annotation and click the **Position Annotation** button on the Annotation mini toolbar.



The positioning annotation mode is activated.

The Annotation Display Options toolbar is displayed in the lower part of the 3D scene.

See [The Annotation Display Options toolbar](#) on page 528.



2. Reposition the annotations using one of the following methods.

Interactively

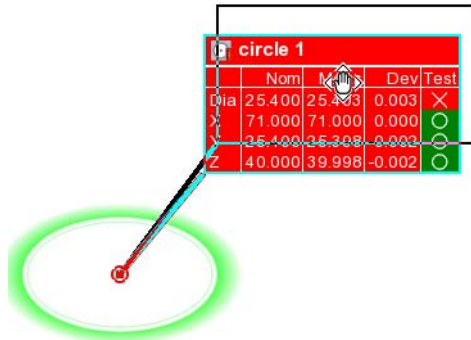
This method allows repositioning annotations individually in the 3D scene by dragging them to the desired location.

1. Left-click an annotation.

The annotation is displayed in red, and a hand symbol is displayed when hovering the mouse over the annotation.

2. Hold the left mouse button and drag to the desired location.

- Press Spacebar to temporarily exit the mode and perform rotations/translations.
- Press ESC to exit the mode or click the first button on the mini toolbar.



Automatically

This method allows repositioning annotations simultaneously in the 3D scene based on the automatic repositioning modes available.

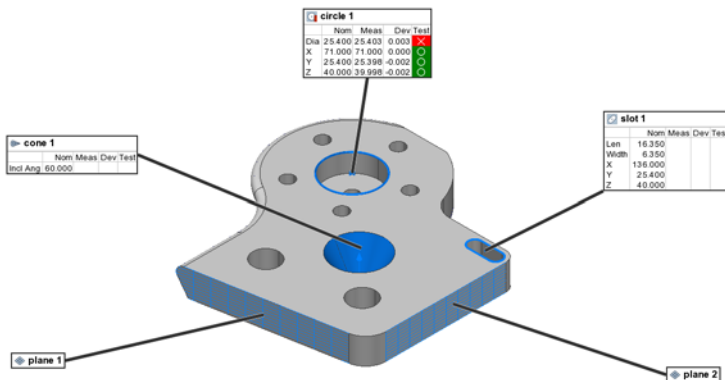
1. On the Annotation Display Options toolbar, click the arrow next to the annotation positioning button.
2. Choose a positioning mode from the drop-down list to position all annotations either:
 - **Close to object:** annotations are automatically positioned as close as possible to their related object.
 - **Close to border:** annotations are automatically positioned along the top, bottom, left, and right boundaries of the 3D Scene pane in a way that minimizes annotation overlap and the intersection of annotation attachment lines.



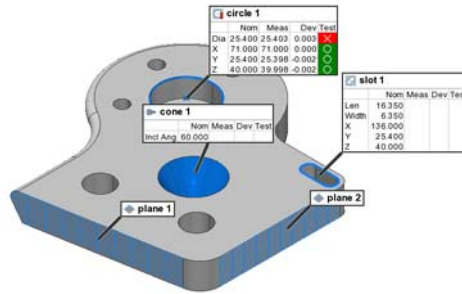
Results

The annotations are now repositioned in the 3D scene. When they have been repositioned automatically, two choices are offered and the difference between both is shown below:

- **Close to border**



- **Close to object**



Notes

- The *View* menu offers tools used to hide, hide interactively, and restore annotations.
- When an annotation is repositioned by the user, it is automatically locked, which means that the length and the orientation of the attachment line will not change, even when performing a transformation in the 3D scene (for example, zoom, rotation, and translation). The annotation can be unlocked using the appropriate button on the mini toolbar, or by right-clicking the object in the Tree View, and choosing *Annotation > Unlock Position*.

The Annotation Display Options toolbar



Sets the annotation positioning mode to close to object/border



Menu button that allows positioning the annotations close to the object or along the boundaries of the 3D Scene pane. In both cases, the annotation overlap and the intersection of annotation attachment lines are minimized.



Options related to annotation positioning mode

Offers options that allow controlling the display and positioning of annotations. It also allows specifying the minimal length of the attachment line, in pixels.



Locks/unlocks the position of all annotations

Menu button that allows locking or unlocking the position of all annotations.



Hides or restores all annotations

Displays or hides all annotations.

Editing an object annotation

On creation, the information of object annotations is determined using a default annotation template. The content of an annotation can be changed by applying another template, or by adding and removing items manually. It is also possible to modify the appearance of annotations.

Setting object annotation content

Annotations are configurable. It is possible to add or remove information items displayed on creation, as well as to modify the order in which the items appear.

Objective: Edit the content of an object annotation.

Required: The project contains an object.

Steps

1. Choose *Tools > Annotations > Edit*, then left-click an object annotation in the 3D scene.

*Alternatively, mouse over the annotation, and click the **Edit Annotation** button on the Annotation mini toolbar.*

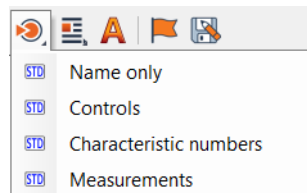


The annotation editor opens, displaying the object annotation.

2. Set the object annotation content using one of the following methods.

From a Template

1. On the annotation editor toolbar, click **Set Contents using Template**.
2. On the menu, select the annotation template to apply.

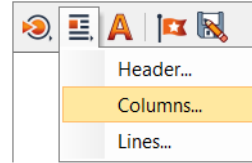


Manually

1. On the annotation editor toolbar, click **Edit Contents**.
2. On the menu, select the section of the annotation to modify.

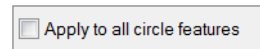
In this example, **Columns** is selected. The Edit Columns dialog box opens.

*Alternatively, right-click in an empty area of the annotation editor and click **Edit Header**, **Edit Columns** or **Edit Lines** to access the dialog box that allows editing the appropriate content.*



Notes

The changes made can be applied to all same-type objects by clicking the check box at the bottom of the Edit Annotation dialog box.



Results

The annotation content is set as specified.

Modifying the appearance of annotations

It is also possible to modify the appearance of annotations.

Objective: Edit the appearance of an annotation.

Required: The project contains an object with an annotation.

Steps

1. Choose *Tools > Annotations > Edit*, then left-click an object annotation in the 3D scene.

*Alternatively, mouse over the annotation, and click the **Edit Annotation** button on the annotation mini toolbar.*



The annotation editor opens, displaying the object annotation.

2. On the annotation editor toolbar, click **Edit Format**.

The **Display > Annotations > Formatting** page of the PolyWorks | Inspector Options dialog box opens.



3. Edit the formatting options of annotations.
4. Click **Apply**.

The formatting is applied to all annotations that appear in the 3D scene.

Results

The annotation appearance is set as specified.

Reviewing measurement results using the Control Reviewer

The Control Reviewer offers a condensed view of the most important information of an inspection project: geometry controls. It displays a table of the controls, and it offers operations to filter, sort, and group them in views. Its content is also synchronized with the 3D scene. Unlike 2D formatted reports, the 3D scene can be manipulated to better view the information.

The Control Reviewer can be used to:

- Review metrology results in an effective way
- Structure inspection projects
- Enhance the reporting workflow

The Control Reviewer pane

To open the Control Reviewer pane:

1. Choose *Report > Control Reviewer*.

Alternatively, click the **Control Reviewer** button on the Main Objects toolbar.



The Control Reviewer pane is displayed. It offers the Controls list box, a contextual toolbar, column headers, the list of controls, and statistics.

Controls list box

Menu bar

Column headers

List of controls

Contextual toolbar

Object Name	Control	Deviation	Test
datum plane A	\angle 0.200	0.089	Pass
datum cylinder B	\perp \varnothing 0.100 A	0.118	Fail
datum cylinder B	Diameter	0.007	Pass
datum cylinder C	\oplus \varnothing 0.200 A B	0.719	Fail
datum cylinder C	\perp \varnothing 0.200 A	0.149	Pass
datum cylinder C	Diameter	0.013	Pass
circle 1	\oplus \varnothing 2.000 A B C	2.072	Fail
circle 1	X	-0.281	Pass
circle 1	Z	0.997	Pass
circle 2	\oplus \varnothing 2.000 A B C	1.203	Pass
circle 2	X	0.029	Pass

Total: 33 Measured: 33 Fail: 15 Warning: 0

Tree View Dialog Zone Control Reviewer

Statistics

Menu bar

Provides access to all Control Reviewer functionalities.

Controls list box

Specifies the controls that will be listed in the pane.

An asterisk is displayed next to the control view name when there are new changes to the specified control view that have not been applied.

Contextual toolbar

Offers quick access to common operations. The operations offered on this toolbar change according to the selected item in the Controls list box.

Column headers

Used to filter and sort controls. It is also possible to show/hide column headers as well as reorder them.

List of controls

Displays controls according to the item selected in the Controls list box. Filters can also be applied to reduce the number of controls displayed.

Statistics

Displays statistics on the controls listed in the pane:

- Total number of controls;
- Number of measured controls;
- Number of controls that are out-of-tolerance (Fail) or for which a warning is issued.

The contextual toolbar

Different tools are offered, depending on the context:



Create Control View

Enters a creation mode based on all the controls listed in the pane, and opens the Create Control View dialog box.



Create Control View from Selected Controls

Enters a creation mode based on the controls selected in the pane, and opens the Create Control View dialog box.



Apply Changes to Control View

Applies the pertinent changes to the control view that is specified in the **Controls** list box.



Clear Filters

Removes all applied filters.



Edit Context

Opens the Edit Context dialog box, which is used to edit the coordinate system and the data alignment used to display the controls currently selected.



Center on All Controls

Positions the 3D scene to show all the controls currently listed in the pane as much as possible, as well as their annotations.



Center on Selected Controls

Positions the 3D scene to show the controls currently selected as much as possible, as well as their annotations.



Restore Project View

Restores the 3D scene project view of the current control view.



Create Snapshot and Table

Creates a report snapshot and a report table for the current control view.



Create Table

Creates a report table for the current control view.



Create Snapshot

Creates a report snapshot for the current control view.

Using the Control Reviewer

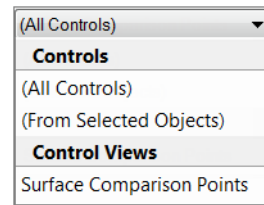
Reviewing controls

By default, all active controls of all measurement objects in the project are listed in the Control Reviewer pane, offering an overview of the results without having any report items or formatted reports created. These controls can be filtered and sorted.

Filtering controls using the Controls list

Steps

1. In the Control Reviewer pane, open the **Controls** list box.
2. Choose one of the following options:
 - **(All Controls)**: Displays all active controls of all measurement objects in the inspection project.
 - **(From Selected Objects)**: Displays all active controls of the selected measurement objects. Objects can be selected in the 3D scene as well as in the Tree View.
 - **Control Views**: User-defined control views are listed under this item. Displays the controls associated with the selected control view.



Results

Controls are listed in the pane according to the selection made in the **Controls** list box.

Object Name	Control	Deviation	Test
surf pt 1	Surface Distance	0.149	Pass
surf pt 2	Surface Distance	0.152	Fail
surf pt 3	Surface Distance	0.227	Pass
surf pt 4	Surface Distance	0.173	Pass
surf pt 5	Surface Distance	0.247	Pass
surf pt 6	Surface Distance	0.183	Pass

Filtering controls using filters

Steps

1. In the Control Reviewer pane, identify the columns that have a filter icon.

Column headers with a filter icon indicate that controls in that column can be filtered.



2. Click the filter icon of a column header.

A contextual dialog box is displayed.

3. Enable the **Activate filter** parameter.

Depending on the column, controls can be filtered using a string-based filter, or a multiple selection from a predefined list of filters.

String-based filter

1. In the list, choose a filtering method:

- **Is Equal To:** Search for strings that are equal to the specified string.
- **Contains:** Search for strings that contain the specified string.

Object Name	Control	Deviation
datum plane A		
datum cylinder B	<input checked="" type="checkbox"/> Activate filter	
datum cylinder B	Contains	
datum cylinder C	Datum	
datum cylinder C		

2. Type the string that will be used to filter the controls.

Multiple-selection based filter

1. In the list, choose a filtering method:

- **Is Equal To:** Search for results that are equal to the selection.
- **Is Different From:** Search for results that are different from the selection.

2. Select the desired expressions used to filter controls.

Control	Deviation	Test
Diameter		
Diameter	<input checked="" type="checkbox"/> Activate filter	
	Is Equal To	
	<input checked="" type="checkbox"/> Diameter	
	<input type="checkbox"/> Flatness	
	<input type="checkbox"/> Length	
	<input type="checkbox"/> Line Profile	
	<input type="checkbox"/> Max Deviation	
	<input type="checkbox"/> Min Deviation	
	<input type="checkbox"/> Min/Max Deviation	
	<input type="checkbox"/> Perpendicularity	
	<input type="checkbox"/> Position	
	<input type="checkbox"/> Surface Distance	

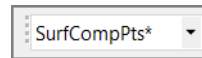
Results

The list of controls is filtered, and only the relevant controls are displayed.

<i>Object Name</i>	<i>Control</i>	Deviation	Test
datum cylinder B	Diameter	0.024	Fail
datum cylinder C	Diameter	-0.027	Pass

Notes

- When a column is filtered, its header is displayed in italics and the filter icon is colored.
- When filtering controls within a control view, an asterisk is added next to the name of the control view indicating that the changes have not been applied. Click **Apply Changes to Control View**.



Alternatively, choose Control View > Apply Changes from the Control Reviewer menu bar.

Sorting controls

Steps

- In the Control Reviewer pane, move the mouse pointer over the desired column header.
- Click a column header to sort controls.

All column headers can be sorted by increasing/decreasing values. A third click on a column header removes the sorting.

Measured	Tolerance	Deviation
Sort by: Deviation/Increasing		
-0.269	±0.100	-0.269
97.326	±1.000	-0.054
7.473	±1.000	-0.027
7.524	±0.010	0.024
-4.763	±0.010	0.037

Measured	Tolerance	Deviation
Sort by: Deviation/Decreasing		
1.898	±1.000	1.898
1.744	2.000	1.744
1.666	2.000	1.666
1.509	2.000	1.509
93.725	±0.800	0.925

Results

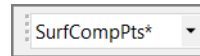
The list of controls is sorted.

Object Name	Control	Nominal	Measured	Tolerance	Deviation	Test
circle 1	\varnothing 2.000 A B C		2.260	2.000	2.260	Fail
c-s 2 (y=-3.681)	Line Profile		1.898	± 1.000	1.898	Fail
circle 4	\varnothing 2.000 A B C		1.744	2.000	1.744	Pass
circle 3	\varnothing 2.000 A B C		1.666	2.000	1.666	Pass
circle 2	\varnothing 2.000 A B C		1.509	2.000	1.509	Pass
circle 1	Z	92.800	93.725	± 0.800	0.925	Fail

Notes

When sorting controls within a control view, an asterisk is added next to the name of the control view indicating that the changes have not been applied. Click **Apply Changes to Control View**.

Alternatively, choose [Control View > Apply Changes](#) from the Control Reviewer menu bar.



Introducing control views

Historically, engineering drawings have been conceived in terms of views: top, side, front, and auxiliary views present the dimensions and measurements of a part. CAD software capable of adding PMI to 3D models also use the concept of views.

Control views can be used to review measurement results in an effective and structured manner, offering an enhanced inspection workflow. Several methods can be used to create control views. Two of them are explained in the following subsections.

Creating control views from objects in the 3D scene

A control view can be easily created from objects in the 3D scene. If desired, a report snapshot and a report table are created at the same time. By default, they are added to the active formatted report.

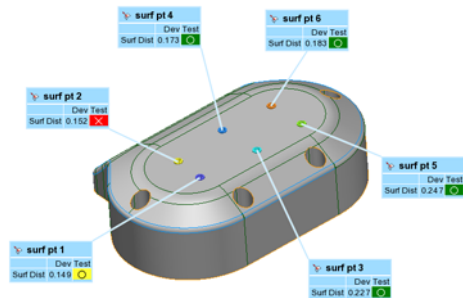
Objective: Create a control view that will include controls for objects in the 3D scene.

Required: The project must have measurement objects with controls.

Steps

1. Position the 3D scene to show all objects that will be part of the control view, adjust their orientation, and position their annotation.

The control view will include controls for all objects in the 3D scene whose annotations are displayed, as well as controls for objects whose control labels are displayed.



2. On the main toolbar, click **Create Control View from Objects in the 3D Scene**.



The Create Control View from 3D Scene dialog box opens.

3. Specify a name for the control view.

By default, report items (snapshot and report table) are created from this control view.

4. Click **Create**.

Creating control views from selected controls

A control view can be created from a selection of controls in the Control Reviewer pane.

Objective: Create a control view based on a selection of controls.

Required: The project must have measurement objects with controls.

Steps

1. In the list of controls, select the controls that will be included in the control view.

Use CTRL or SHIFT when left-clicking to select multiple controls.

If desired, controls can be filtered and sorted to list only specific controls in the pane.

Object Name	Y-	Control	Y-
surf pt 1		Surface Distance	
surf pt 2		Surface Distance	
surf pt 3		Surface Distance	
surf pt 4		Surface Distance	
surf pt 5		Surface Distance	
surf pt 6		Surface Distance	

2. In the Control Reviewer pane, click **Center on Selected Controls**.

This positions the 3D scene to show, as much as possible, the associated objects and their annotations.



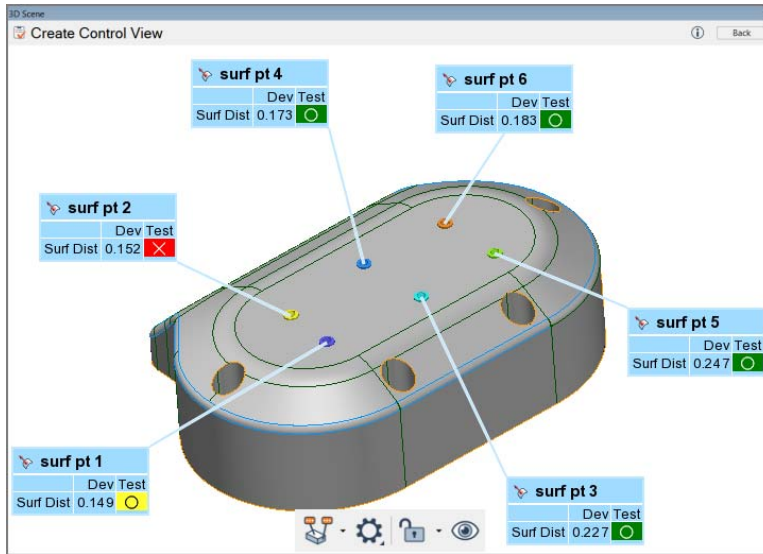
3. In the Control Reviewer pane, click **Create Control View from Selected Controls**.

The Create Control View dialog box opens.



- In the 3D scene, adjust the orientation of the objects and position of the annotations.

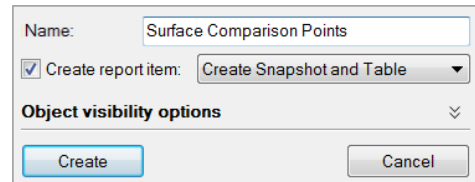
While in the creation mode, a toolbar that offers quick access to annotation display options is available. See [The Annotation Display Options toolbar](#) on page 528 for detailed information.



- In the Create Control View dialog box, specify a name for the control view.

By default, report items (snapshot and report table) are created from this control view.

- Click **Create**.



Results

- Report items linked to the control view are created and automatically included in the active formatted report.

The screenshot shows two windows: 'Control Reviewer' and 'Report Editor - report 1'.

Control Reviewer Table:

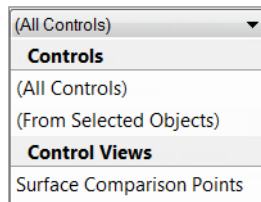
Object	Y1 Control	Y2	Nomi.	Meas.	Toler.	Devia.	Test Y.
surf pt 1	Surface Distan		0.149	±0.150	0.149		Pass
surf pt 2	Surface Distan		0.152	±0.150	0.152		Fail
surf pt 3	Surface Distan		0.227	±0.500	0.227		Pass
surf pt 4	Surface Distan		0.173	±0.500	0.173		Pass
surf pt 5	Surface Distan		0.247	±0.500	0.247		Pass
surf pt 6	Surface Distan		0.183	±0.500	0.183		Pass

Report Editor - report 1 Control View Table:

Char No.	Object Name	Control	Item	Meas	Tol	Dev	Test	Out Tol
27	surf pt 1	Surface Distance		0.149	±0.150	0.149	Pass	
28	surf pt 2	Surface Distance		0.152	±0.150	0.152	Fail	0.002
29	surf pt 3	Surface Distance		0.227	±0.500	0.227	Pass	
30	surf pt 4	Surface Distance		0.173	±0.500	0.173	Pass	
31	surf pt 5	Surface Distance		0.247	±0.500	0.247	Pass	
32	surf pt 6	Surface Distance		0.183	±0.500	0.183	Pass	

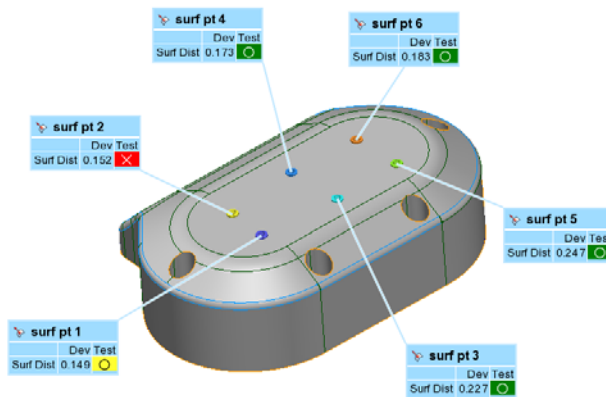
Additional information in the Control View section includes: Units: Millimat, Coordinate System: world, Organization, Operator, Email, Part name: piece 2, Part number, Piece: piece 2, and Date: 2017-02-01.

- The control view is saved and added to the **Control Views** section of the **Controls** list box.



- The control view can now be selected in the **Controls** list box. When selected, its controls are listed in the pane, and the associated 3D scene view is displayed.

Object Name	Control	Deviation	Test
surf pt 1	Surface Distance	0,149	Pass
surf pt 2	Surface Distance	0,152	Fail
surf pt 3	Surface Distance	0,227	Pass
surf pt 4	Surface Distance	0,173	Pass
surf pt 5	Surface Distance	0,247	Pass
surf pt 6	Surface Distance	0,183	Pass



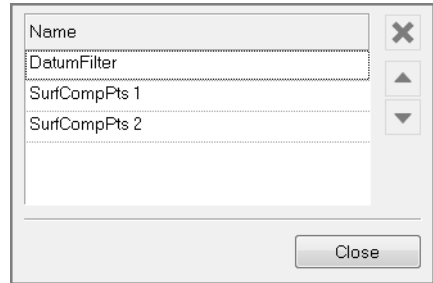
Managing control views

Control views can be deleted, reordered, and renamed.

Steps

1. In the Control Reviewer pane, choose *Control View > Manage Control Views*.

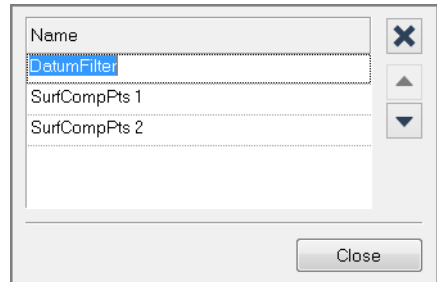
The Manage Control Views dialog box opens.



2. Select the desired control views in the control view list area.

The CTRL and SHIFT keys can be used to perform a multiple selection.

The vertical toolbar is made available.

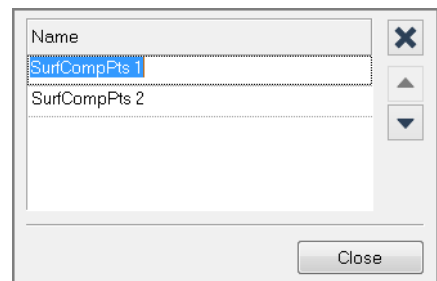


- Click **Delete** to delete the selected control views.

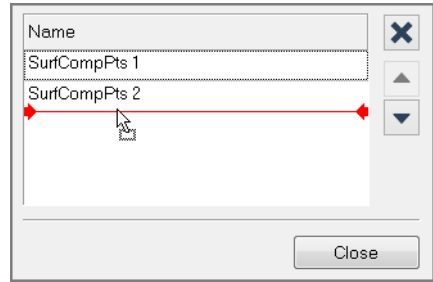
*Alternatively, select the control view from the controls list box. Choose *Control View > Delete*.*

- Click the **Move Up** and **Move Down** buttons to move the selected control views within the list.

By default, control views are listed in creation order.

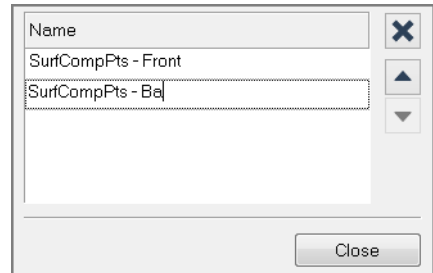


Alternatively, drag and drop control views in the control view list area to place them in the desired order.



- Click the text box to rename the control view.

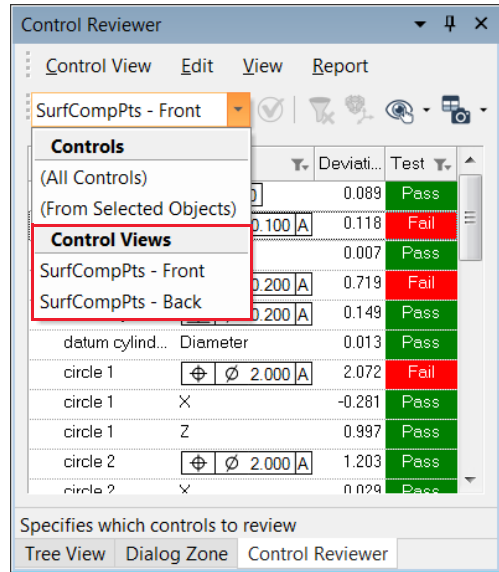
*Alternatively, choose **Control View > Properties** to rename a selected control view.*



3. Changes are automatically applied to the list of control views as soon as they are made. Click **Close**.

Results

The changes made to the selected control views in the Manage Control Views dialog box are automatically applied.



Notes

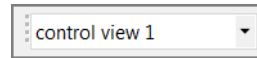
Object visibility options can be specified for each control view:

- Select a control view.
- Choose *Control View > Properties*.

Editing control views

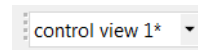
Steps

1. In the Control Reviewer pane, select a control view in the **Controls** list box.



2. Select the controls to edit. Choose *Edit* from the menu bar. Choose one of the following:
 - Edit color scale limits.
 - Modify the context of the selected controls or the controls themselves.
 - Clear filters for selected controls.
 - Add selected controls to another control view.
 - Remove the selected controls or the corresponding objects from the control view.
 - Select the corresponding objects in the Tree View for the selected controls.
 - Display the properties for the corresponding object of a selected control.

3. An asterisk is added next to the name of the control view in the **Controls** list box when changes are made in the 3D scene, to the objects, or to the controls of the current control view but the changes have not yet been applied.



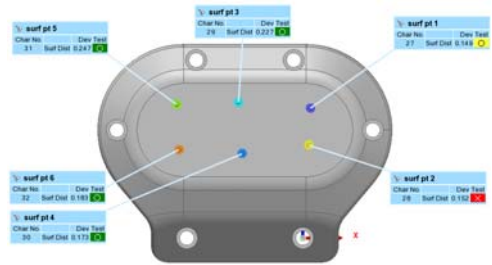
Click **Apply Changes to Control View** to apply the changes.



Alternatively, choose Control View > Apply Changes from the Control Reviewer menu bar.

Results

The control view now contains all the changes that were made.



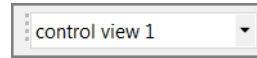
Displaying controls in the 3D scene

Controls and their associated measurement objects can be displayed in the 3D scene based on the project view embedded in a control view, or based on the content of Control Reviewer pane.

From a control view

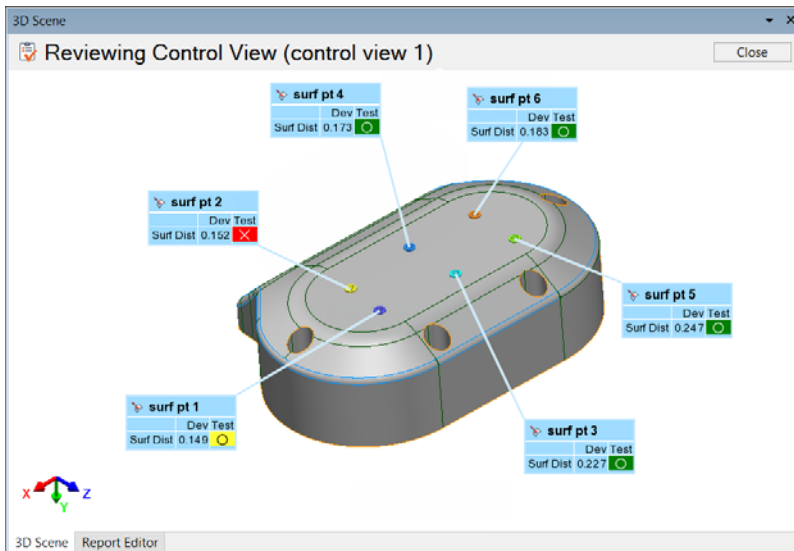
Steps

1. In the Control Reviewer pane, choose a control view from the **Controls** list box.



Results

The project view associated with the control view is shown in the 3D scene and the reviewing control view mode is launched.



Notes

- A project view contains the list of displayed objects, elements, and annotations and their position in the 3D scene.
- The 3D scene project view can be adjusted while in the reviewing control view mode.
 - If the project view associated with the control view is adjusted but not saved, an asterisk is displayed next to the name of the control view indicating that the changes have not been applied.



Click **Apply Changes to Control View** on the *Control Reviewer* toolbar to apply changes.



- Click **Restore Project View** to restore the saved project view, which corresponds to last time any changes were applied to that control view.



From the list of controls

Steps

Center on All Controls

1. If desired, filter and sort controls to display only specific controls in the list of controls.

(All Controls)	
Object Name	Control
surf pt 1	Surface Distance
surf pt 2	Surface Distance
surf pt 3	Surface Distance
surf pt 4	Surface Distance
surf pt 5	Surface Distance
surf pt 6	Surface Distance

2. Click **Center on All Controls**.



Center on Selected Controls

1. In the list of controls, select the controls that will be displayed in the 3D scene.

Use SHIFT or CTRL when left-clicking to select multiple controls.

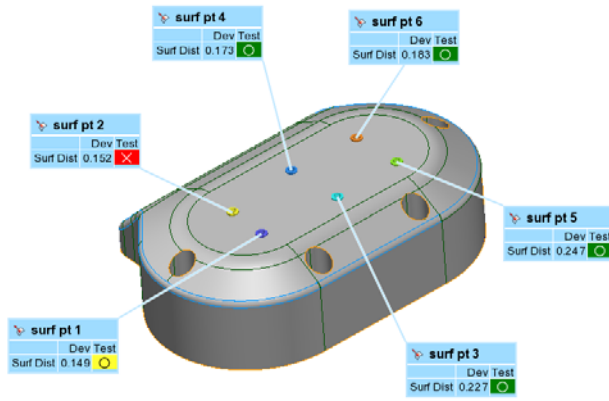
(All Controls)	
Object Name	Control
surface 1	Max Deviation
c-s 2 (y=-3.681)	Line Profile
c-s 2 (y=-3.681)	Min Deviation
c-s 2 (y=-3.681)	Max Deviation
surf pt 1	Surface Distance
surf pt 2	Surface Distance
surf pt 3	Surface Distance
surf pt 4	Surface Distance
surf pt 5	Surface Distance
surf pt 6	Surface Distance
caliper 1	Length

2. Click **Center on Selected Controls**.



Results

The 3D scene displays the controls based on the selection that was previously made.



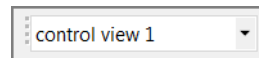
Generating snapshots and report tables from control views

Reports can also be used to analyze and communicate inspection measurements. Typically, a report consists of report tables and snapshots, complete with observations, comments, and conclusions – all formatted into a printable document.

Reporting using control views allows generating snapshots and report tables that are synchronized and linked to control views. In other words, snapshots display the project view of the control view they are associated with, and report tables display the controls of the control view they are associated with.

Steps

1. In the **Controls** list box of the Control Reviewer pane, select the control view that will be in the report.
2. Click the arrow next to **Create Snapshot and Table**, and choose one of the following:
 - **Create Table:** Creates a report table linked to the current control view.
 - **Create Snapshot:** Creates a snapshot linked to the current control view.
 - **Create Snapshot and Table:** Creates a snapshot and a report table linked to the current control view.



Results

Report items linked to the control view are created and automatically included in the active formatted report.

The screenshot shows two windows: 'Control Reviewer' and 'Report Editor - report 1'.

Control Reviewer Table:

Object	Control	Nomi.	Meas.	Toler.	Devia.	Test
surf pt 1	Surface Distan...	0.149	±0.150	0.149	Pass	
surf pt 2	Surface Distan...	0.152	±0.150	0.152	Fail	
surf pt 3	Surface Distan...	0.227	±0.500	0.227	Pass	
surf pt 4	Surface Distan...	0.173	±0.500	0.173	Pass	
surf pt 5	Surface Distan...	0.247	±0.500	0.247	Pass	
surf pt 6	Surface Distan...	0.183	±0.500	0.183	Pass	

Report Editor - report 1 Table:

Char No.	Object Name	Control	Nom	Meas.	Tol	Dev	Test	Out Tol
27	surf pt 1	Surface Distance	0.149	±0.150	0.149	Pass		
28	surf pt 2	Surface Distance	0.152	±0.150	0.152	Fail		0.002
29	surf pt 3	Surface Distance	0.227	±0.500	0.227	Pass		
30	surf pt 4	Surface Distance	0.173	±0.500	0.173	Pass		
31	surf pt 5	Surface Distance	0.247	±0.500	0.247	Pass		
32	surf pt 6	Surface Distance	0.183	±0.500	0.183	Pass		

Additional report details: Units: Millimet., Coordinate System: world, Organization: Smart, Operator: Smart, Part name: Piece, Part number: piece 2, Date: 2017-03-01, Page: 2/2.

Notes

- When creating a control view, it is possible to create both the control view and the corresponding report items.

Dialog box configuration:

- Name: Surface Comparison Points
- Create report item: Create Snapshot and Table
- Object visibility options: (collapsed)
- Buttons: Create, Cancel

- Report items of all control views can also be automatically generated by choosing *Report > Create Snapshots and Tables > From All Control Views*.

Viewing measurement results using the object navigator

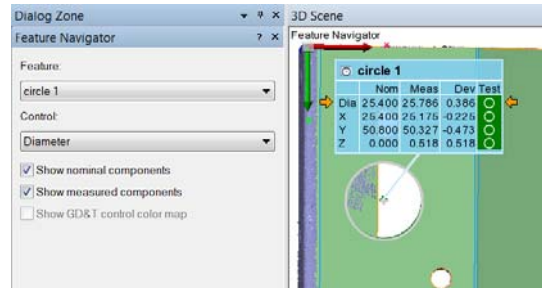
Object navigators are specialized tools that allow viewing objects one by one. In addition, they display the dimensional and GD&T controls of features, and the measurement results of gauges.

Such visualization of the measurements provides information about deviation sources and direction, which can be useful when troubleshooting.

Object navigators are available for:

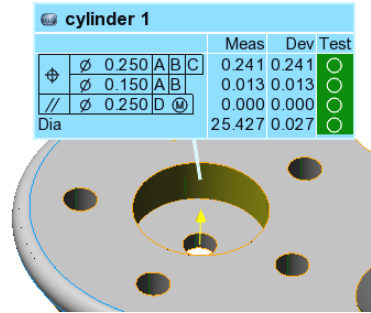
- Features
- Cross-sections
- Airfoil gauges
- Flush & gap gauges
- Profile gauges

To access the object navigators, choose one from the *View > Object Navigator* submenu.



Reviewing feature measurement results

The Feature Navigator is useful to review feature results, in order to view the dimensional and the GD&T measurements for each feature individually. This visualization of the measurements provides information about the deviation. Note that when features are created on cross-sections, features are shown on their cross-section.



Feature dimension results

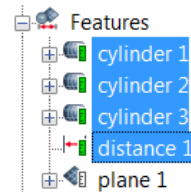
Objective: Display the result of a dimensional measurement on the model.

Required: The PolyWorks | Inspector project must have measured features with a dimensional control.

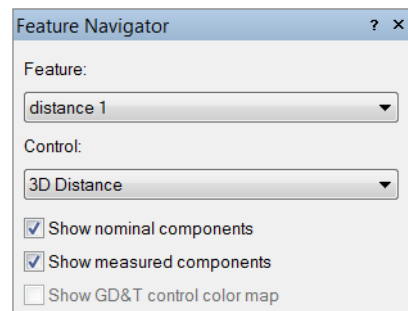
Steps

1. In the Tree View, select the features to visualize.
2. Choose *View > Object Navigator > Features* or right-click and choose *View > Feature Navigator*.

The Feature Navigator dialog box opens.

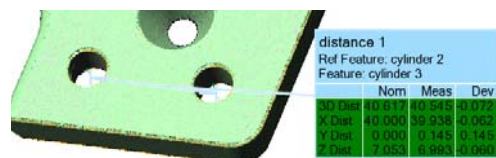


3. In the **Feature** list, select the feature to display.
4. In the **Control** list, select the dimensional control to display.
5. To display the nominal and/or measured feature, select **Show nominal components** and/or **Show measured components**.



Results

The feature dimensions for the selected feature are displayed in the 3D scene.



Feature GD&T controls

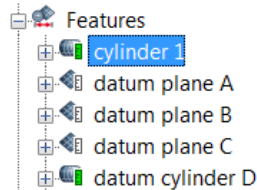
Objective: Display the result of a GD&T measurement on the model, showing the datum reference frame, alignment, and calculated features.

Required: The PolyWorks | Inspector project must have a measured feature with a GD&T control.

Steps

1. In the Tree View, select the feature to visualize.
2. Choose *View > Object Navigator > Features* or right-click and choose *View > Feature Navigator*.

The Feature Navigator dialog box opens.



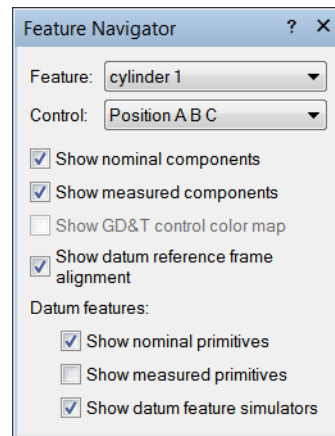
3. In the **Control** list, select the GD&T control to display.

4. To display the nominal and/or measured feature, select **Show nominal components** and/or **Show measured components**.

5. A color map can be displayed on GD&T controls that are measured using data points (for example, Surface Profile) by selecting **Show GD&T control color map**.

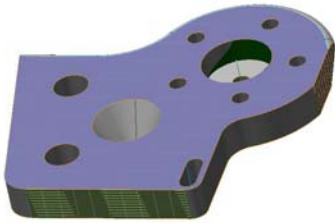
6. If **Show datum reference frame alignment** is selected, it displays the dataset and measured features in position, as defined by the datums in the datum reference frame.

7. Display the datum reference frame components by selecting the items under **Datum features**.

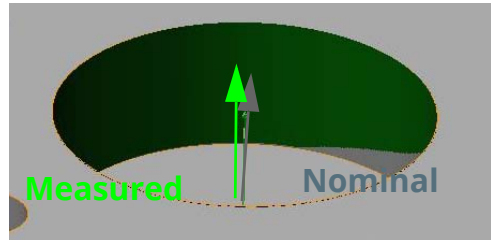


Results

The model, datums used in the datum reference frame, and feature itself are displayed in the 3D scene. The measured features are aligned using the datum reference frame.

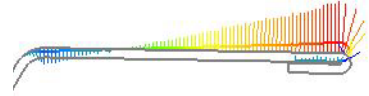


The nominal feature and the aligned and constrained measured feature are displayed, using Pass/Fail colors.



Reviewing cross-section measurement results

It is useful to review cross-section results, using the Cross-Section Navigator, in order to visualize each cross-section individually. Such visualization of the cross-sections provides information about deviation, which can be useful when troubleshooting.



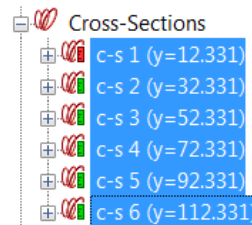
Objective: Display the results of a cross-section, showing the nominal and measured sections and deviations.

Required: The PolyWorks | Inspector project must have measured cross-sections.

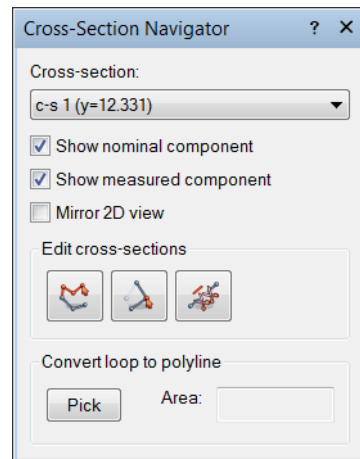
Steps

1. In the Tree View, select the cross-sections to visualize.
2. Choose *View > Object Navigator > Cross-Sections* or right-click and choose *View > Cross-Section Navigator*.

The Cross-Section Navigator dialog box opens.

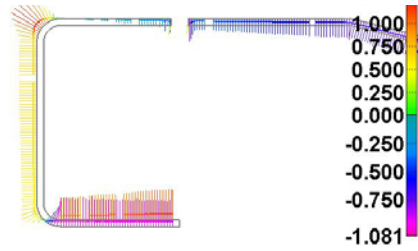


3. In the **Cross-section** list, select the cross-section to display.
4. Select **Show nominal component** and/or **Show measured component** to display the nominal and/or measured cross-section components.
5. To manually modify the cross-sections by extending, adjusting, or cutting them, click the **Extend**, the **Pick and Place Vertices**, or the **Cut Segments** button.



Results

The cross-sections and deviations are displayed in 2D in the 3D scene.



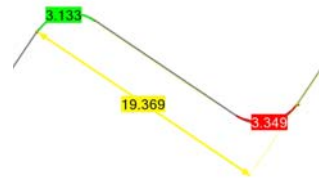
Notes

By default, measured cross-sections are displayed along with a deviation color map and deviation vectors. To hide the color map and show only the cross-section profiles, click **Hide Color Map**.



Reviewing profile gauge measurement results

It is useful to review profile gauge results, using the Profile Gauge Navigator, in order to see how the results were obtained. Such visualization provides information about deviation, which can be useful when troubleshooting.

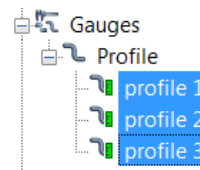


Objective: Display the result of a profile gauge, showing the measurement zone, cross-sections, and gauge dimensions.

Required: The PolyWorks | Inspector project must have measured profile gauges.

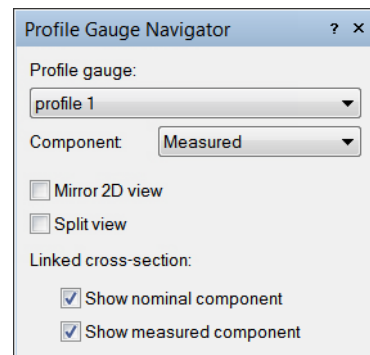
Steps

1. In the Tree View, select the profile gauges to visualize.
2. Choose *View > Object Navigator > Profile Gauges* or right-click and choose *View > Profile Gauge Navigator*.



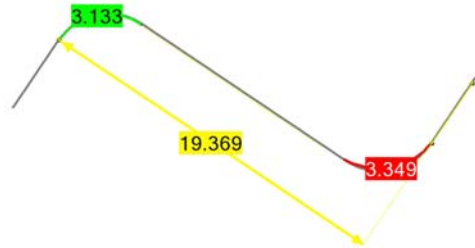
The Profile Gauge Navigator dialog box opens.

3. In the **Profile gauge** list, select the gauge to display.
4. In the **Component** list, select a component to display the **Nominal** or **Measured** component.
5. To show the components of the cross-sections used to calculate the profile gauge, select **Show nominal component** and/or **Show measured component**.



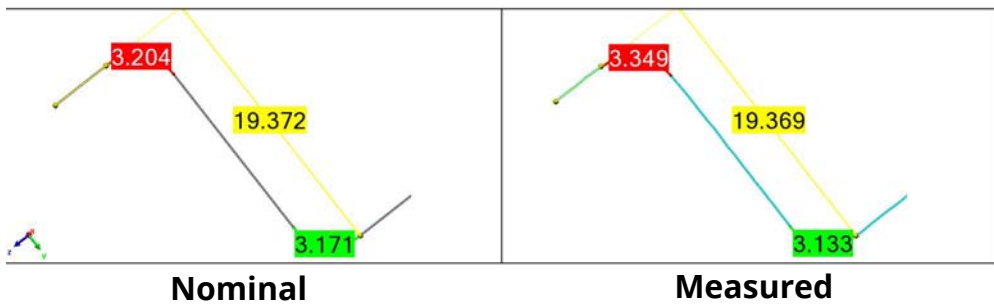
Results

The profile gauge dimensions and linked cross-sections are displayed in 2D in the 3D scene.



Notes

To view both the nominal and measured profile gauge results simultaneously, select **Split view**. The nominal cross-sections and profile gauge dimensions are displayed on the left. The measured cross-sections and profile gauge dimensions are displayed on the right.



Reviewing flush & gap gauge measurement results

It is useful to review flush & gap gauge results, using the Flush & Gap Gauge Navigator, in order to see how the results were determined. Such visualization provides information about deviation, which can be useful when troubleshooting.

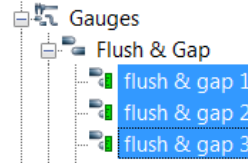


Objective: Display the result of a flush & gap gauge, showing the measurement zone, cross-sections, and gauge dimensions.

Required: The PolyWorks | Inspector project must have measured flush & gap gauges.

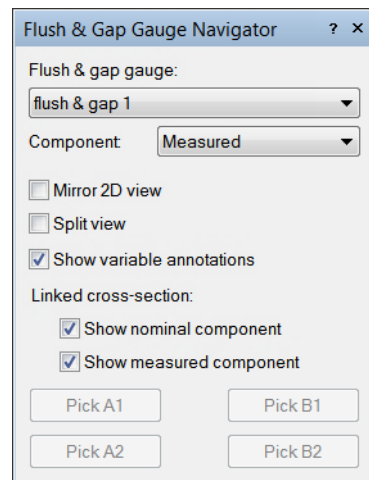
Steps

1. In the Tree View, select the flush & gap gauges to visualize.
2. Choose *View > Object Navigator > Flush & Gap Gauges* or right-click and choose *View > Flush & Gap Gauge Navigator*.



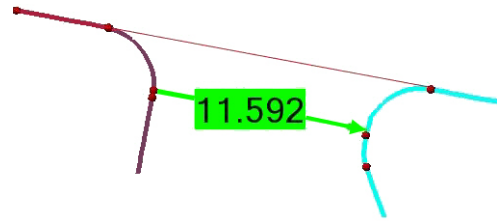
The Flush & Gap Gauge Navigator dialog box opens.

3. In the **Flush & gap gauge** list, select the gauge to display.
4. In the **Component** list, select a component to display the **Nominal** or **Measured** component.
5. Select the **Show variable annotations** to display the coordinates of the gauge contact points and other gauge key points.
6. To show the components of the cross-sections used to calculate the flush & gap gauge, select **Show nominal component** and/or **Show measured component**.



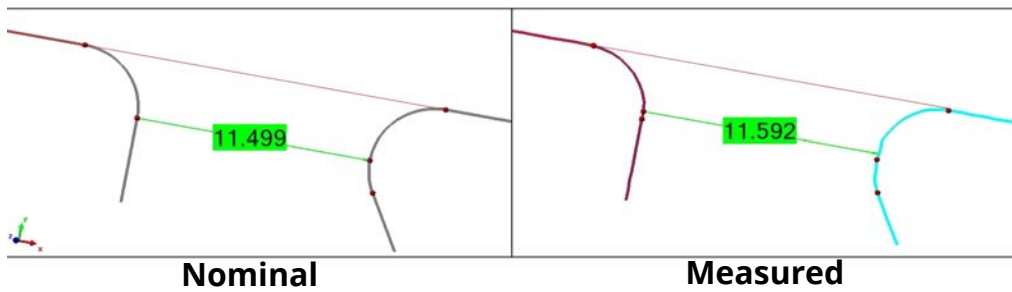
Results

The flush & gap gauge dimensions and linked cross-sections are displayed in 2D in the 3D scene.



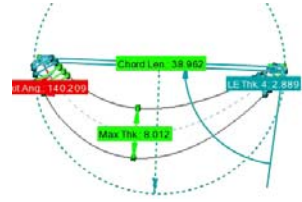
Notes

To view both the nominal and measured flush & gap gauge results simultaneously, select **Split view**. The nominal cross-sections and flush & gap gauge dimensions are displayed on the left. The measured cross-sections and flush & gap gauge dimensions are displayed on the right.



Reviewing airfoil gauge measurement results

It is useful to review airfoil gauge results, using the Airfoil Gauge Navigator, in order to examine each individual cross-section. Such visualization provides information about deviation, which can be useful when troubleshooting.

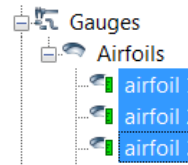


Objective: Display the result of an airfoil gauge, showing the measurement zone, cross-sections, and gauge dimensions.

Required: The PolyWorks | Inspector project must have measured airfoil gauges.

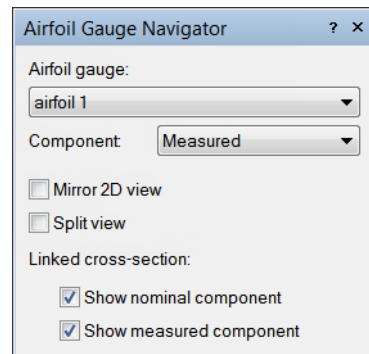
Steps

1. In the Tree View, select the airfoil gauges to visualize.
2. Choose *View > Object Navigator > Airfoil Gauges* or right-click and choose *View > Airfoil Gauge Navigator*.



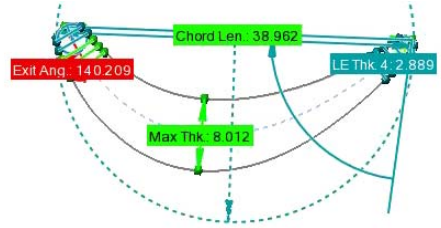
The Airfoil Gauge Navigator opens.

3. In the **Airfoil gauge** list, select the gauge to display.
4. In the **Component** list, select a component to display the **Nominal** or **Measured** component.
5. To show the components of the cross-sections used to calculate the airfoil gauge, select **Show nominal component** and/or **Show measured component**.



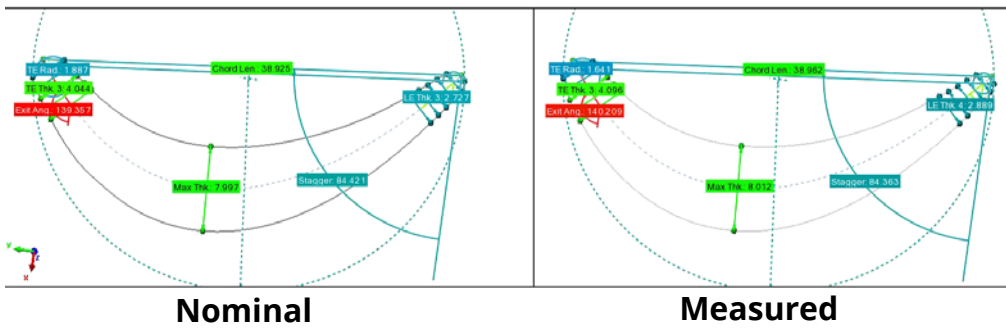
Results

The airfoil gauge dimensions and linked cross-sections are displayed in 2D in the 3D scene.



Notes

To view both the nominal and measured gauge results simultaneously, select **Split view**. The nominal cross-sections and gauge dimensions are displayed on the left. The measured cross-sections and gauge dimensions are displayed on the right.



Reporting

Producing reports is key to analyzing and communicating inspection measurements. Typically, a report consists of report tables and report snapshots, completed by observations, comments, and conclusions, all formatted into a printable document.

PolyWorks | Inspector offers tools to generate and edit such reports.

Here is the typical workflow for reporting:

1. Create report tables.



Create report tables from measurement objects in the Tree View.

They are automatically added to the active formatted report

Feature Table

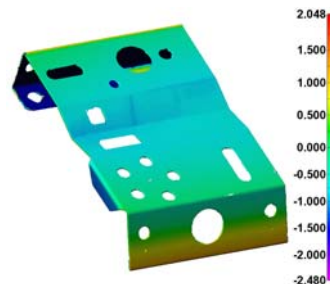
Units		Millimeters					
Coordinate System		world					
Data Alignments		best-fit 1					
Name	Control	Nom	Meas	Tol	Dev	Test	Out Tol
circle 1	Diameter	25.400	25.786 ±1.000	0.386	Pass		
circle 2	Diameter	25.400	25.538 ±1.000	0.138	Pass		
	X	177.540	177.757 ±1.000	0.217	Pass		
	Y	63.500	63.953 ±1.000	0.453	Pass		
	Z	16.933	16.747 ±1.000	-0.186	Pass		
△angle 1	3D Angle 1	90.000	91.031 ±1.000	1.031	Fail		0.031

2. Create 3D scene snapshots.



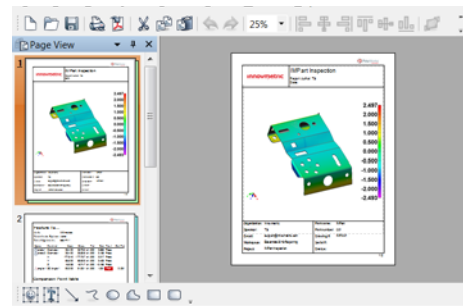
Create snapshots of the 3D scene to show inspection results.

They are automatically added to the active formatted report.



3. Prepare the formatted report.

Using the Report Editor tool, format the report to include project and piece information, comments on the results, and shapes to emphasize content.

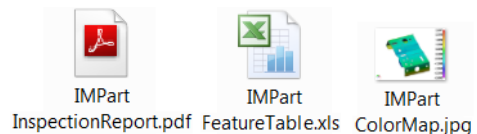


4. Export.



Export the formatted report into different formats, such as PDF.

The individual report items can be exported as well. An example is to export to Microsoft Excel.



Working with report tables

A report table is a listing of the desired information of any measurement object, such as nominal and measured values, tolerances, deviations, and pass/fail status. Tables are a very useful way to summarize inspection results.

Typically, a measurement object report table includes a title, a header area, the name of each object, and the following information for each of its controls:

Feature Table

Units	Millimeters						
Coordinate System	world						
Data Alignments	best-fit 1						
Name	Control	Nom	Meas	Tol	Dev	Test	Out Tol
circle 1	Diameter	25.400	25.786	±1.000	0.386	Pass	

- Nominal and measured values

Note that certain dimensional controls (for example, distance controls), and GD&T controls (except size controls), do not have nominal values.

- Tolerance limits
- A deviation
- A pass/fail test result
- Out-of-tolerance value

Since objects are updated by the Automatic Project Update mechanism, table results are automatically updated as well. Table contents and formatting can be edited to suit different reporting needs. Table reports can be incorporated into a formatted report or exported for use in applications like Microsoft Excel.

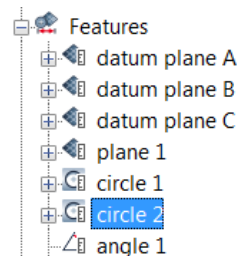
Creating report tables

Objective: Create a report table containing the current results of a measurement object.

Required: The project contains a measurement object.

Steps

1. In the Tree View, select the measurement object for which to report measurements.

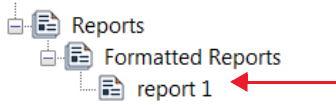


2. Choose *Report > Create Tables > From Objects*.



Results

The report table is added to the active formatted report, shown at the bottom of the Tree View.

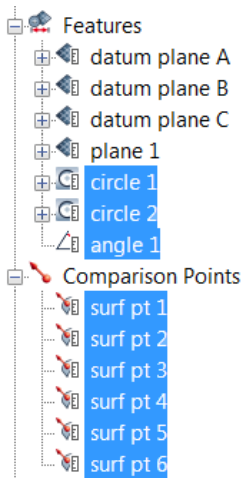


The report table is automatically positioned on a page in the report, and can be viewed by double-clicking the report name in the Tree View.

Feature Table							
Units		Millimeters					
Coordinate System		world					
Data Alignments		best-fit 1					
Name	Control	Nom	Meas	Tol	Dev	Test	Out Tol
circle 2 Diameter		25.400	25.538	±1.000	0.138	Pass	
X		177.540	177.757	±1.000	0.217	Pass	
Y		63.500	63.953	±1.000	0.453	Pass	
Z		16.933	16.747	±1.000	-0.186	Pass	
Organization:		Part name:					
Operator:		Part number:					
E-mail:		Pieces: piece 1					

Notes

If two or more types of object are selected, a report table will be generated for each type. In the example shown below, all selected features are included in one report table, and selected comparison points are included in a second report table.



Feature Table							
Units		Millimeters					
Coordinate System		world					
Data Alignments		best-fit 1					
Name	Control	Nom	Meas	Tol	Dev	Test	Out Tol
circle 1 Diameter		25.400	25.786	±1.000	0.386	Pass	
circle 2 Diameter		25.400	25.538	±1.000	0.138	Pass	
X		177.540	177.757	±1.000	0.217	Pass	
Y		63.500	63.953	±1.000	0.453	Pass	
Z		16.933	16.747	±1.000	-0.186	Pass	
angle 1 3D Angle 1		90.000	91.031	±1.000	1.031	Fail	0.031

Comparison Point Table							
Units		Millimeters					
Coordinate System		world					
Data Alignments		best-fit 1					
Linked Objects		IMP art.igs					
Name	Control	Nom	Meas	Tol	Dev	Test	Out Tol
surface cmp pt 1	3D Distance		-0.682	±0.500	-0.682	Fail	-0.182
surface cmp pt 2	3D Distance		-0.106	±0.500	-0.106	Pass	
surface cmp pt 3	3D Distance		-0.746	±0.500	-0.746	Fail	-0.246
surface cmp pt 4	3D Distance		-0.069	±0.500	-0.069	Pass	
surface cmp pt 5	3D Distance		-0.720	±0.500	-0.720	Fail	-0.220
surface cmp pt 6	3D Distance		0.012	±0.500	0.012	Pass	

Setting report table contents using a template

When a report table is created from a measurement object, its contents are specified by the object's default table template. The report table can then be edited as required, by adding and removing information and changing the formatting. A new user table template can be created from the setup of the current report table, and can then be used to set the contents and formatting for other reports of the same type. It can also be designated as the default table template for the particular type of report table.

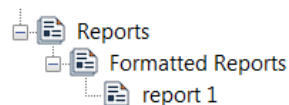
Table templates provided with the PolyWorks Metrology Suite are called standard templates and have the STD prefix in template lists, while table templates created by users have the USR prefix. Standard templates cannot be deleted.

Objective: Set the contents of a report table by applying a table template.

Required: The project contains a report table.

Steps

1. In the Tree View, double-click a report name to open the Report Editor.



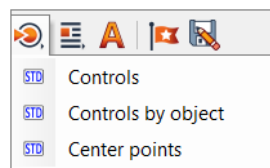
2. In the Report Editor, select a report table.
3. Choose *Edit > Table*.

The table editor opens displaying the report table.

4. On the table editor toolbar, click **Set Contents using Template**.



5. On the menu, select the table template to apply.



Results

The report table has the contents and formatting defined by the table template.

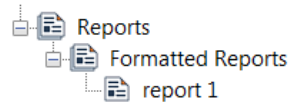
Editing report table contents manually

Objective: Edit the contents of a report table manually.

Required: The project contains a report table.

Steps

1. In the Tree View, double-click a report name to open the Report Editor.



2. In the Report Editor, select the report table to edit.
3. Choose *Edit > Table*.

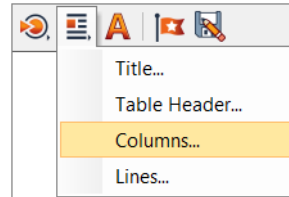
The table editor opens displaying the report table.

4. On the table editor toolbar, click **Edit Contents** to edit the contents.

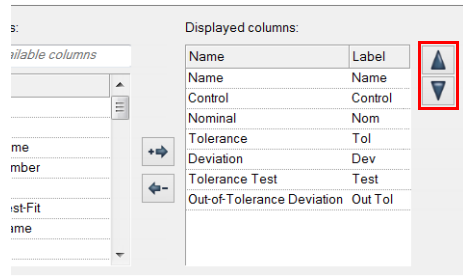
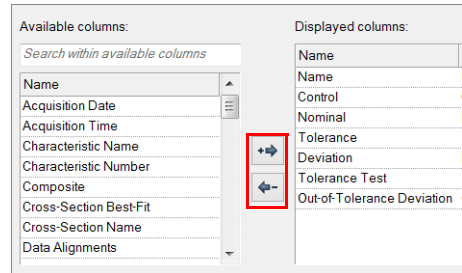


5. On the menu, select the section of the table to modify.

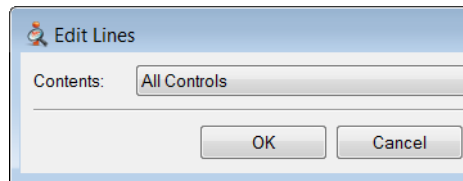
In this example, **Columns** is selected. The Edit Columns dialog box opens.



6. To add columns, select them in the **Available columns** list, and click the Add arrow to transfer them to the **Displayed columns** list.
7. To remove displayed columns, select them in the **Displayed columns** list, and click the Remove arrow to transfer them to the **Available columns** list.
8. Reorder the displayed columns using the Move Up and Move Down arrows located on the right side.
9. Click **Apply** to view the changes in the table.
10. Click **OK** to close the editor.
11. Repeat steps 3 through 9 to edit other sections of the table.



A similar dialog box is available when editing the other sections, such as **Title** and **Table Header**. However, when editing **Lines**, the dialog box features a list box that specifies if a table shows **All Controls** for the measurement objects, or allows selecting **Specific** lines (specific controls and dimensions) – **Specific** must be selected.



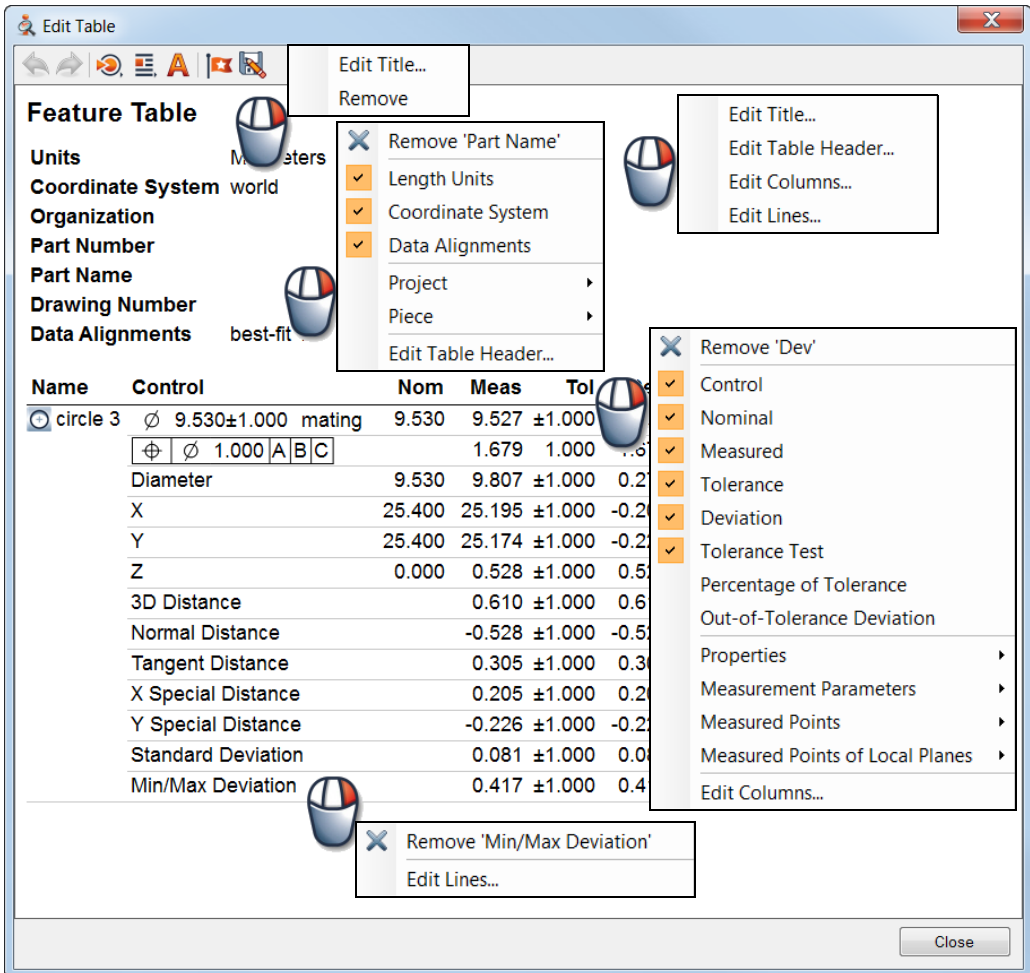
Results

The report table now contains the desired information.

Notes

Edit more quickly using the shortcut menus available by right-clicking in the different sections of the table editor:

- Click **Edit Title**, **Edit Table Header**, **Edit Object Header**, **Edit Columns**, and **Edit Lines** for the dialog boxes like those described previously.
- Add and remove items for each section by clicking them on the section's shortcut menu.



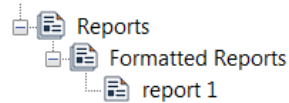
Saving a table template

Objective: Save a table template from a modified report table.

Required: The project contains a report table.

Steps

1. In the Tree View, double-click a report name to open the Report Editor.



2. In the Report Editor, select a report table.
3. Choose *Edit > Table*.

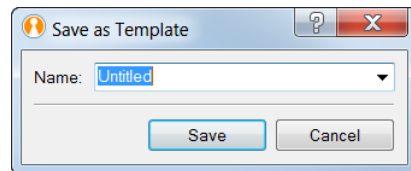
The table editor opens displaying the report table.

4. On the table editor toolbar, click **Save as Template** to save the current report table configuration as a user table template.



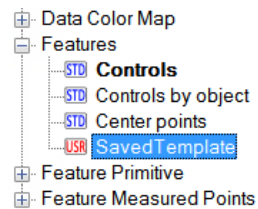
The Save as Template dialog box opens.

5. Specify the name of the new table template and click **Save**.



Results

The new user table template (USR) is saved. It can be applied to other report tables of the same type.



Managing table templates

The template editor allows personalizing table templates, either to create a new table template, to set a table template as the default one, edit the contents of a user table template or delete it from the list of table templates.

Managing existing templates

Objective: For an object type, set a default table template, or delete a user template from the list of table templates.

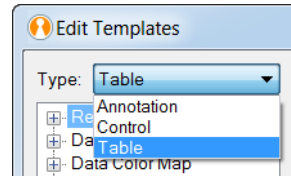
Required: More than two table templates for the object type.

Steps

1. Choose *Tools > Edit templates*.

The Edit Templates dialog box opens.

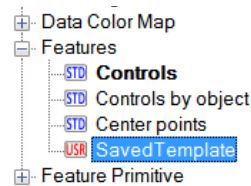
2. In the **Type** list, select **Table**.



3. Browse for the desired table type.

4. Select the table template to define as the default one.

*The feature table template **SavedTemplate** is selected as an example.*



5. To make the selected template the default template, click **Set as Default**.



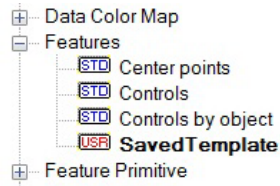
6. To delete an existing user template (USR), select it and click **Delete**.



Results

The new default table template is indicated in bold.

New feature report tables will be created using the new default table template.



Creating and editing a user table template

Objective: Create a new user template for an object type and edit its contents.

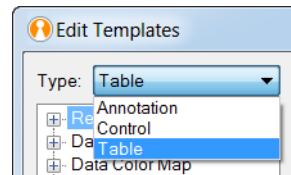
Required: PolyWorks | Inspector is open.

Steps

1. Choose *Tools > Edit Templates*.

The Edit Templates dialog box opens.

2. In the **Type** list, select **Table**.



3. Browse for the desired table type.
 4. To create a new user template, click **Create**.
- The Create User Template dialog box opens.

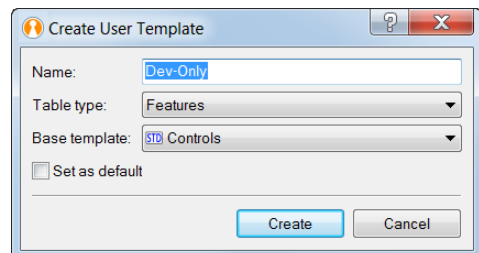


5. Specify the **Name** of the new user template.
6. Specify the **Base template** for the new user template.

If a template is selected upon creation, it is preselected as the base template by default.

7. Click **Create**.

*The newly created user template can be set as default by selecting the **Set as default** option.*



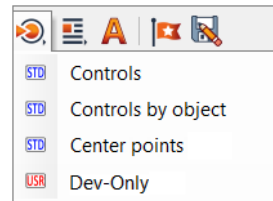
8. Edit the content of the newly created table template directly on the right pane.

For more information, see [Editing report table contents manually](#) on page 573.

9. Click **Apply**.

Results

The new user table template is added to the list of table templates. It can now be assigned to report tables.



Notes

Edit the content of a user table template at any time by selecting it on the template list, making the required modifications on the right pane, then clicking **Apply**.

Editing the report table format

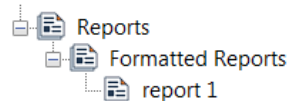
Objective: Edit the font and font size of a report table.

Required: The project contains a report table.

Steps

Edit the format of a single report table

1. In the Tree View, double-click a report name to open the Report Editor.



2. In the Report Editor, select a report table.
3. Choose *Edit > Table* from the Report Editor menu bar.

The table editor opens displaying the report table

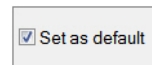
4. On the table editor toolbar, click **Edit Format**

The Edit Table Format dialog box opens.



5. Set the appropriate options.

*To set the table format as the default one for all new tables, select **Set as default** in the Edit Table Format dialog box. The present settings are transferred to the **Objects > Reports > Tables > Formatting** page of the PolyWorks | Inspector options.*



6. Click **Apply**.

The formatting is applied to the report table.

Specify the formatting options of all new report tables

1. Choose *Tools > Options*.

The PolyWorks | Inspector Options dialog box opens.



2. Choose *Objects > Reports > Tables > Formatting*.
3. Set the appropriate options.
4. Click **Apply**.

The formatting is applied to all subsequent new report tables.

Working with report snapshots

Report snapshots are images captured from the 3D scene to report the model and measurement results in full detail.

Since objects are updated through the Automatic Project Update, the report snapshot will automatically be updated. The snapshot contents can also be edited to modify the part orientation or the objects shown at any time. Report snapshots can be incorporated into a formatted report, or exported for use in applications like Microsoft Excel.



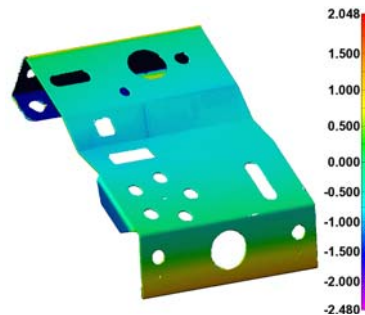
Creating report snapshots

Objective: Create a report snapshot showing the contents of the 3D scene.

Required: The project contains a visible measurement object or model.

Steps

1. Prepare the 3D scene by giving the model a suitable orientation and zoom level, and displaying only the desired measurement objects.



2. The annotations can be repositioned. Choose *Tools > Annotations > Position*.
 - Using the ALT key as a shortcut to drag annotations.
 - When an annotation is repositioned, it is automatically locked. In this case, its position remains unchanged when performing a transformation in the 3D scene (for example, zoom, rotation and translation).

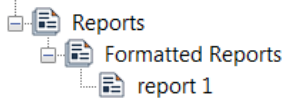


3. Choose *Report > Create Snapshots > Capture 3D Scene*.

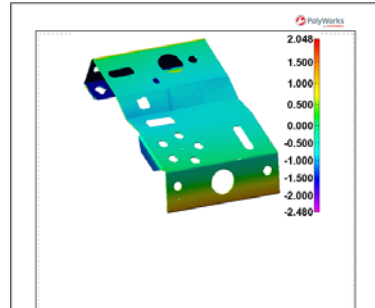


Results

The snapshot is added to the active formatted report, shown at the bottom of the Tree View.



The snapshot is automatically positioned on a page in the report, and can be viewed by double-clicking the report name.



Editing a report snapshot

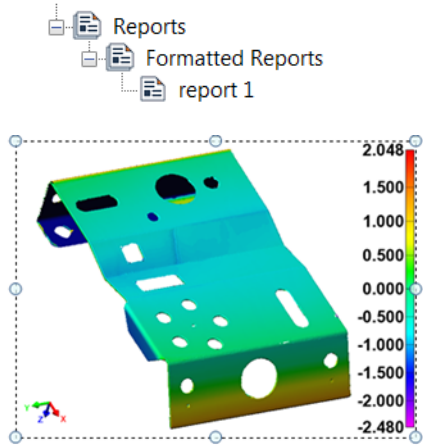
Objective: Edit a report snapshot to change its orientation or contents.

Required: The project contains a report snapshot.

Steps

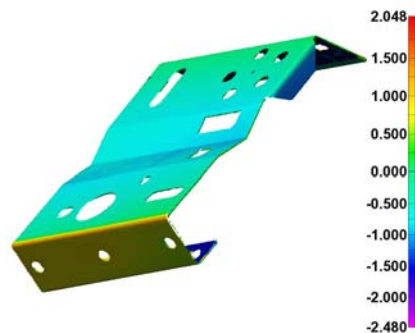
Report snapshots are stored by default in the active formatted report, listed at the bottom of the Tree View.

1. In the Tree View, double-click a report name to open the Report Editor.
2. In the Report Editor, select a report snapshot.
3. Choose *Edit > Snapshot*.



4. In the 3D scene, modify the zoom level, orientation, or objects displayed. The snapshot extent is displayed as a dotted boundary line.

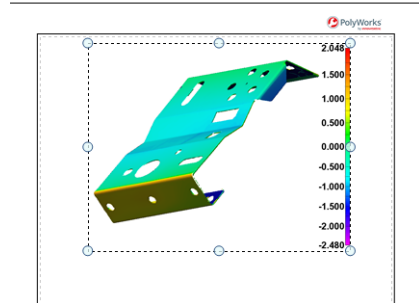
*The extent of the **3D Scene region** and the **Snapshot size** can also be edited using the fields in the Edit Snapshot dialog box.*



5. Click **Recapture**.

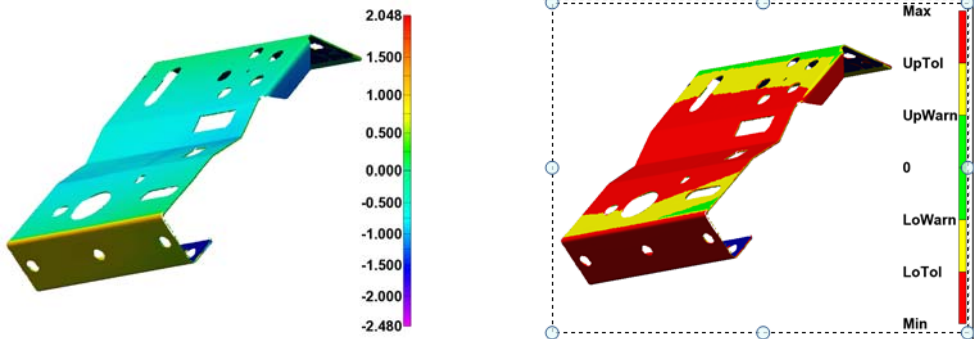
Results

The report snapshot is updated in the formatted report.



Notes

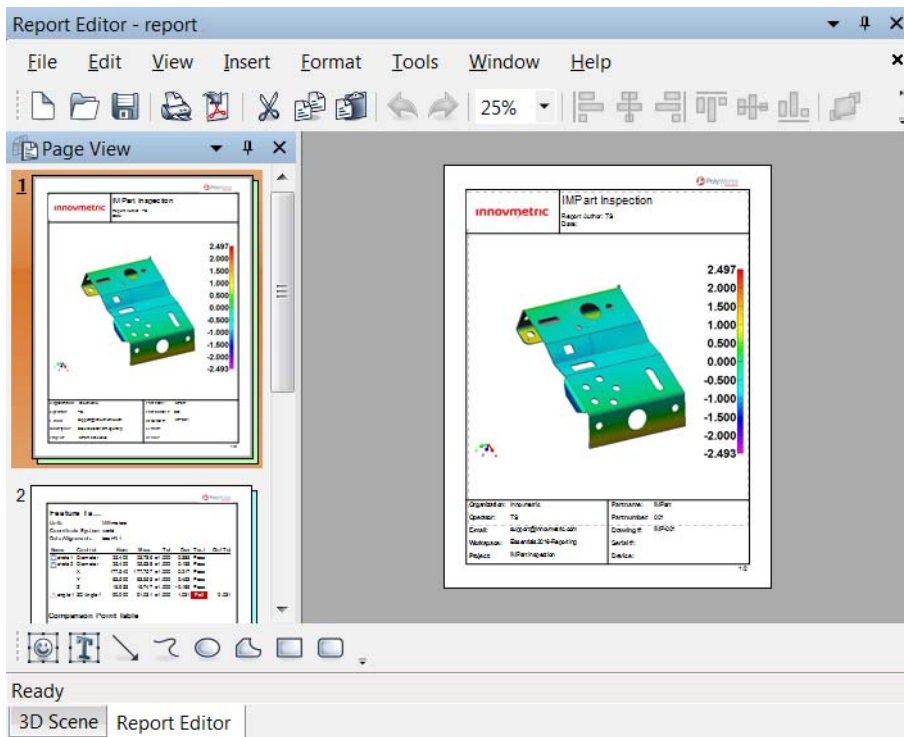
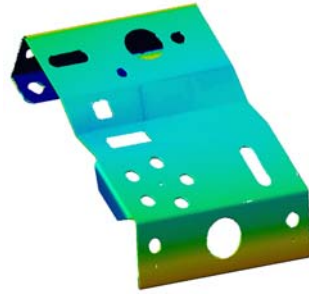
If a color map simply needs to be updated, do not edit the report snapshot. Rather, select the snapshots for which the color map needs to be updated, and choose *Edit > Assign Current Color Scale to Snapshots > Selected*, or right-click and choose *Assign Current Color Scale*.



Working with formatted reports

Reports are a good, organized way to present or share inspection results. Formatted reports normally contain report snapshots, report tables, text, and graphics and can be edited and prepared as needed using the Report Editor tool. While formatted reports are automatically generated when report snapshots or report tables are created, they can also be created manually. After a report is completed, it is generally exported in PDF format for others to review.

Formatted reports are displayed in the Report Editor pane. The Page View interface lists the page thumbnails, and the main interface displays the page currently selected for review and editing. Common tools are offered on toolbars located above and below the page area, respectively.



Creating formatted reports

By default, a formatted report is created as soon as a report snapshot or a report table is created. Subsequent report items are automatically added sequentially on pages of the formatted report as they are generated. A formatted report can also be created manually at any time.

Objective: Create an empty formatted report manually.

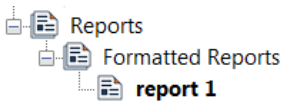
Required: A PolyWorks | Inspector project.

Steps

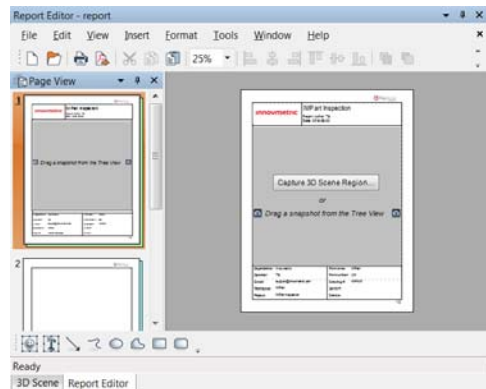
1. Choose *Report > Create Formatted Report*.
2. Specify report properties.
3. Click **Apply** and then click **Close**.

Results

The formatted report object is added at the bottom of the Tree View as the active formatted report (in bold).



The formatted report is initially empty. Newly created report items (snapshots and tables) are automatically added to the active formatted report.



Notes

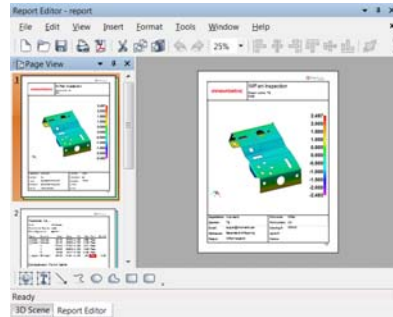
Newly created report items are added to the active formatted report, which is displayed in bold in the Tree View. To make any formatted report the active one, select it in the Tree View and choose *Report > Formatted Report > Activate*.

Editing formatted reports

Objective: Edit an existing formatted report, move report items, and add text and graphics.

Required: The project contains a formatted report.

1. In the Tree View, double-click a report name to open the Report Editor.
2. Perform one of the following edition operations.



Add a snapshot to the title page

For reports using one of the standard layouts, it is possible to quickly add a snapshot to the title page. The title page offers a container for snapshots.

Objective: Add a snapshot to the title page.

Required: A report using one of the standard report layouts.

Steps

1. Click **Capture 3D Scene Region** to launch the capture 3D scene region mode.

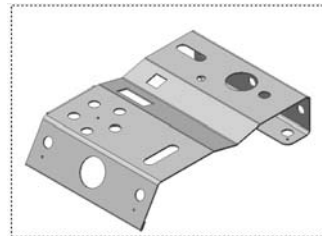
The Capture 3D Scene Region dialog box opens.



2. Show the objects to be captured and orient them in the 3D scene.

Resize the capture region if required.

The Annotation Display Options toolbar is displayed in the lower part of the 3D scene. Move, change options, or show/hide annotations if necessary.



3. In the Capture 3D Scene Region dialog box, it is possible to set the region in pixels, if required.

Set the values for the **Left**, **Top**, **Width**, **Height** parameters.

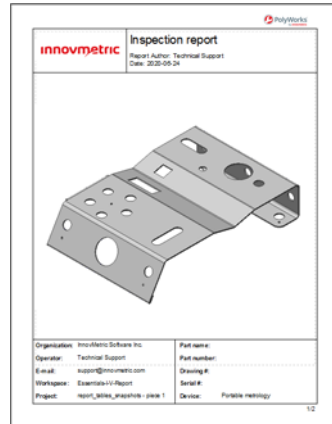
3D Scene region (pixels):

Left:	<input type="text" value="109"/>
Top:	<input type="text" value="70"/>
Width:	<input type="text" value="1090"/>
Height:	<input type="text" value="710"/>

4. Click **Capture**.

Results

The snapshot is automatically added to the title page.



Note

Alternatively, if a snapshot is in the Report > Items branch of the Tree View, it is possible to drag and drop it over the snapshot container to add the snapshot to the title page.

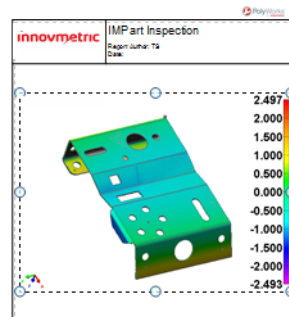
Move or resize report items in the formatted report

Steps

1. In the Report Editor, left-click the report item to select it.
2. Left-click and drag the report item to move it to a different location.

Move the report item to another page by dragging it to a page thumbnail in the Page View.

3. Left-click and drag the handles of the report item to resize it.



Results

The changes are applied to the report items.

Notes

To crop an image, click it to select it, then choose *Format > Crop Image*. Drag the handles to crop, and then click anywhere outside the image to confirm.

Add and edit a text field in the report

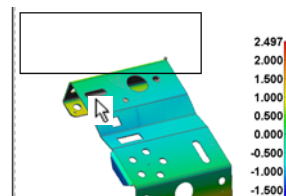
Steps

1. In the Report Editor, choose *Insert > Text Field > Fixed* to add a text field.



2. Position the text field by moving the mouse to the intended location on the page, and left-click to confirm.

*The text field edition mode is automatically activated. To access this mode after creation, select the text field and choose *Edit > Text*.*



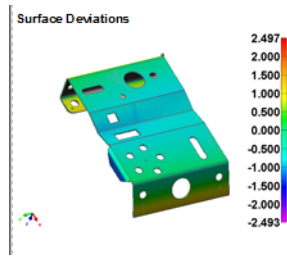
3. Type to add text.

*It is possible to change the format of a text field by clicking it and choosing **Edit > Item Properties**.*



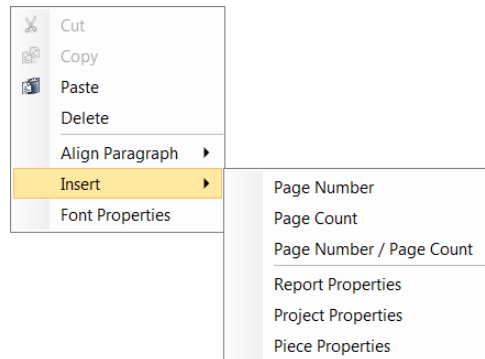
Results

The text field is created and formatted in the report.



Notes

It is also possible to add a variable to the text field. Enter the text field edition mode, then right-click and choose *Insert*. Page numbers, dates, and properties can all be added, and their values update automatically



Add and edit a graphical shape in the formatted report

Steps

1. In the Report Editor, choose *Insert > Shapes > [choose a shape type]* to add graphical shapes.



- Position the shape by moving the mouse to the intended location on the page.
- Left-click and drag to create the shape.



*It is possible to format the created shape by clicking it and choosing **Edit > Item Properties**.*

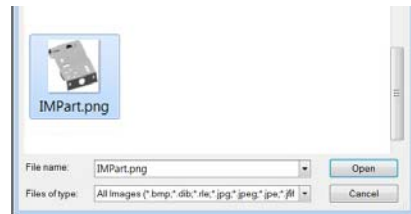
Results

The shape is created and formatted in the formatted report.

Add a fixed image to the formatted report

Steps

- In the Report Editor, add an image by choosing *Insert > Image From File*.
- Browse to the image file.
- Click **Open**.
- Move the image to the desired location on the report page.
- Resize the image, if necessary, by dragging the item handles.



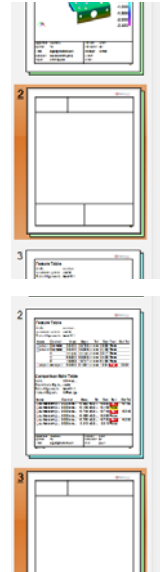
Results

The image is added to the report.

Add, reorder, or remove pages from the formatted report

Steps

1. In the Report Editor, add a report page by choosing *Insert > Page*.
2. Reorder pages by clicking and dragging a page thumbnail.
3. Delete a report page by selecting the thumbnail and choosing *Edit > Delete*.



Results

The pages are added, reordered, or deleted from the formatted report.

Generate a PDF file from a formatted report

Objective: Save a formatted report as a PDF file.

Required: The project contains a formatted report.

Steps

1. In the Tree View, double-click a report name to open the Report Editor.
2. In the Report Editor, choose *File > Export Report > PDF*.
3. Browse to the desired location.
4. Specify a name for the exported formatted report.
5. Click **Save**.



File name:	IMPart InspectionReport.pdf
Save as type:	PDF files (*.PDF)

Results

The formatted report is saved in PDF format.

Exporting report items

Report items, such as tables and snapshots, can be exported from a formatted report. This can be useful when the inspection results are used to fill out an existing external template, or are published online or on a company network. Available export options include Excel, HTML, and text files.

Objective: Export report items using available formats.

Required: The project contains a formatted report with report items.

Steps

1. Open the formatted report that contains the report items to export by double-clicking it in the Tree View.
2. In the Report Editor, select the report items to export.

To select multiple report items, hold the CTRL key while selecting them.
3. Choose **File > Export Items** to export the selected items.

A file browser opens.

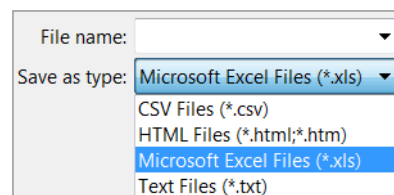
4. Specify a **File name** and location.
5. Select in the **Save as type** list, a file format.

*The **Options** button opens a dialog box that offers export parameters.*

6. Click **Save**.

Name	Control	Nom	Meas	Tol	Dev	Test	Out Tol
circle 1	Diameter	25.400	25.798	±1.000	0.398	Pass	
circle 2	Diameter	25.400	25.528	±1.000	0.128	Pass	
X		177.540	177.757	±1.000	0.217	Pass	
Y		63.500	63.953	±1.000	0.453	Pass	
Z		16.933	16.747	±1.000	-0.186	Pass	
angle 1	3D Angle	90.000	91.031	±1.000	1.031	Fail	0.031

Name	Control	Meas	Tol	Dev	Test	Out Tol
surfacecomp pt 1	3D Distance	-0.882	±0.500	-0.882	Fail	-0.182
surfacecomp pt 2	3D Distance	-0.106	±0.500	-0.106	Pass	
surfacecomp pt 3	3D Distance	-0.746	±0.500	-0.746	Fail	-0.246
surfacecomp pt 4	3D Distance	-0.069	±0.500	-0.069	Pass	
surfacecomp pt 5	3D Distance	-0.720	±0.500	-0.720	Fail	-0.220
surfacecomp pt 6	3D Distance	0.012	±0.500	0.012	Pass	



Results

The selected report items are exported to the specified file format.

Notes

To export items to the Microsoft Excel format, Microsoft Excel must be installed on the computer.

Inspecting multiple pieces



Repeating an inspection on multiple pieces

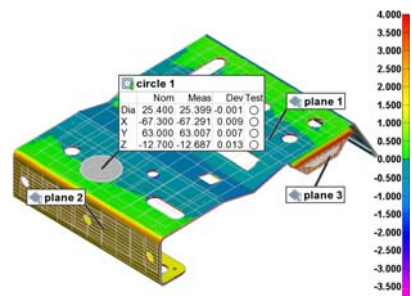
In PolyWorks | Inspector, all measurement object parameters and relations are stored in the project. The stored information facilitates relational, parametric inspection within the PolyWorks | Inspector project, where remeasuring of one object automatically updates any related objects. This functionality also allows for perfect repeatability when inspecting multiple pieces, using the inspection routine of the first inspected piece to inspect subsequent pieces. A single inspection project can contain several piece templates. Then, when measuring a new piece, the operator can choose the appropriate piece template based on the current setup, for example, according to the type of device used.



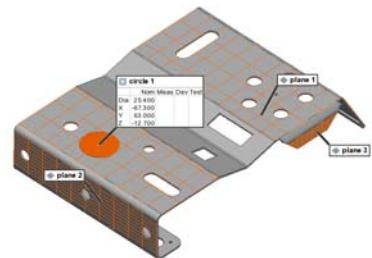
Multipiece inspection consists of four general steps:



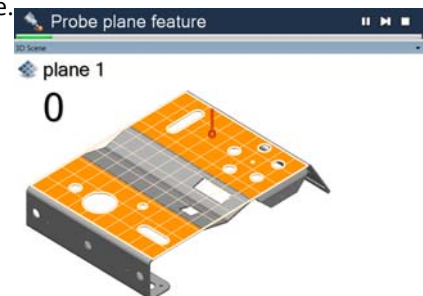
1. Inspect and report the first piece.



2. Create a piece template from the first piece.



3. Create a new piece from the piece template.



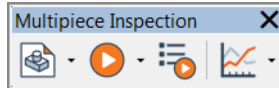
4. Inspect a new piece using Play Inspection.

All pieces are stored within a single PolyWorks|Inspector project and their results can be reviewed individually or summarized using Statistical Process Control (SPC) tools.

Working with multiple pieces

Each physical version of a part is called a **piece**. By default, the first piece is named piece 1. When the inspection is completed for piece 1, a **piece template** can be created from this piece to be used for subsequent pieces.

Operations related to multiple pieces are managed in the **Multipiece Inspection** toolbar.



New Piece

Opens an interface used to add a new piece to the project.



New Piece Template

Opens an interface used to create a new piece template.



Play Inspection

Plays the active measurement sequence.



Sequence Editor

Displays the Sequence Editor pane.



Object Control SPC

Opens an interface to perform statistical process analysis on object controls.

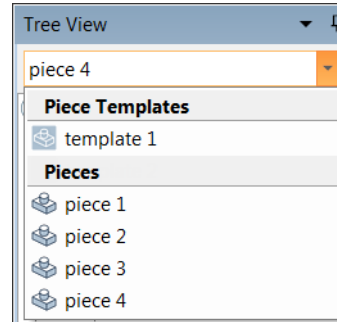


Surface Data SPC

Opens an interface to perform statistical process analysis on surface data objects.

Pieces and piece templates are stored within the project. They are available from the piece list at the top of the Tree View.

Upon opening a piece, it automatically becomes the active piece and therefore the current piece.



Choose *File > Project Properties* to manage pieces and piece templates, and their properties.

Alternatively, use the **Piece Properties** button.



Creating a new piece template

Once a piece inspection is completed, it is recommended to create a piece template using the active piece to ensure uniformity among the new pieces.

Objective: Create a new piece template from the current piece.

Required: A piece completely inspected.

Steps

1. Choose *Tools > Piece > Template > New*.

The New Piece Template dialog box opens.



2. In the **Copy from** list, select **(Current Piece)**.

If the project contains existing templates, these are available in the list to allow creating a new piece template from the selected one.

Propagation links can be established to propagate changes. For more information, see [Propagating piece changes](#) on page 605.

No.	Name	Linked Piece Template
1	piece 1	

Number of pieces: 1 Selected: 0

Select All Select None Clear Filters

Create Cancel

3. Click **Create**.

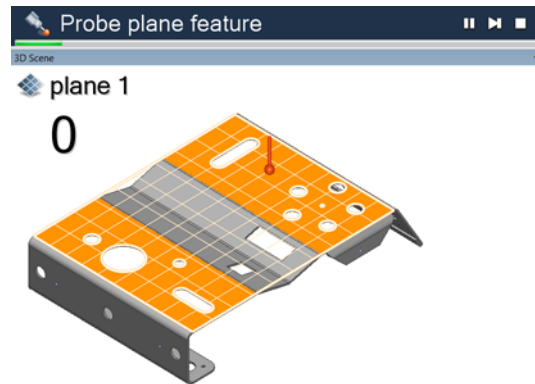
Results

The piece template is created and stored in the project. It contains all the measurements and inspection sequences defined for the first piece.

Inspecting a new piece using Play Inspection

The Play Inspection tool offers a guided, step-by-step sequence to capture 3D datasets of a new piece, using either point cloud digitizing or probing devices.

Combining it with the Control Reviewer pane allows to view control results in real-time as the associated measurement objects are measured. See [Reviewing measurement results using the Control Reviewer](#) on page 533.



The Play Inspection tool:

- Automatically proposes a logical order of measurement that considers device positions, data alignments, object Tree View sequence, and geometric constraints;
- Triggers the right measurement method for each object;
- Manages multiple device positions.

The sequence proposed by the Play Inspection tool is called an autogenerated sequence and it can be customized using the Sequence Editor. For information, see [Introducing the Sequence Editor](#) on page 643.

Objective: Repeat an initial inspection on a new piece.

Required: A first piece completely inspected or a piece template.

Steps

1. Choose *Tools > Piece > New*.



The New Piece properties dialog box opens.

The piece name as well as the creation date and time are automatically specified.

- Choose the **Source piece** used to define the new piece:
 - **Current piece:** A nominal copy of the current piece is used to create the new piece.
 - **Piece template:** The template is used to create the new piece.

By default, if a piece template is used to create the new piece, the new piece is linked to that piece template. This allows for propagating piece changes. For more information, see [Propagating piece changes](#) on page 605.

- If required, specify other standard information for the new piece.
- Click **Create**.

A nominal copy of the current piece or piece template is created.

Source piece:

Current piece

Piece template: template 1

Link to the piece template

Piece template properties

Piece properties

Property	Value
Piece name	piece 2
Date	4/23/2020
Time	2:05:43 PM
Serial number	
Order number	
Operator name	
E-Mail address	
Device	

Fixed alignment setup

Options

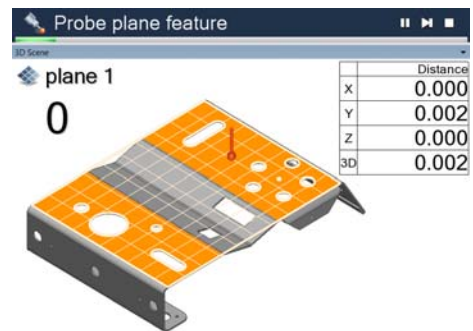
The measurement sequence associated to the current piece or to the piece template is applied to the new piece.

By default, Play Inspection is automatically launched, guiding the user through the steps necessary to calculate the measurement objects for the new piece. The inspection is based on the measurement sequence of the first inspected piece or the piece template.

When required, the measurement sequence is paused and a message is displayed to indicate that a manual action is required.

- If required, use the **Pause**, **Skip Current Operation**, and **Stop** buttons to interrupt or modify the inspection process.

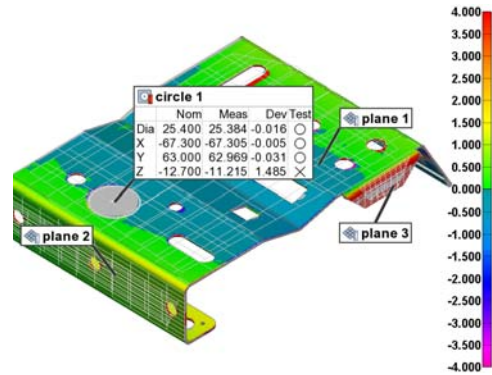
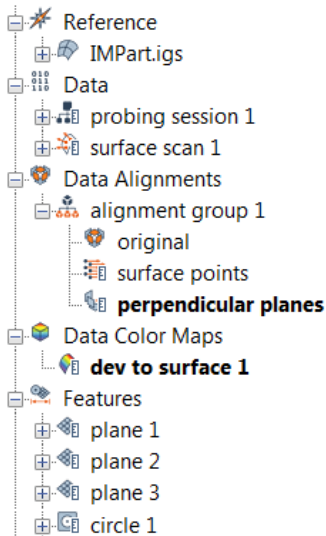
A message is displayed when the new piece inspection is finished.



Results

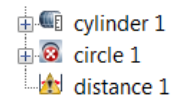
The new inspection is completed and the measurement objects of piece 2 are displayed in the Tree View. A measured status icon indicates the measured status.

The results of the inspection performed in piece 2 are shown in the 3D scene.



Notes

- The inspection sequence can be launched manually at any time by choosing *Tools > Piece > Play Inspection* to measure unmeasured objects.
- If a measurement object cannot be calculated, a warning message is displayed. The message can be used for troubleshooting purposes. The problematic objects are flagged in the Tree View with an X overlay.



Customizing guiding instructions

During an inspection, standard on-screen messages guide the operator through scanning or probing the piece properly. Additional guidance can also be provided through custom instructions or images for any Data objects or measurement objects.



Objective: Repeat an initial inspection on a new piece using custom guidance.

Required: A first piece completely inspected or a piece template.

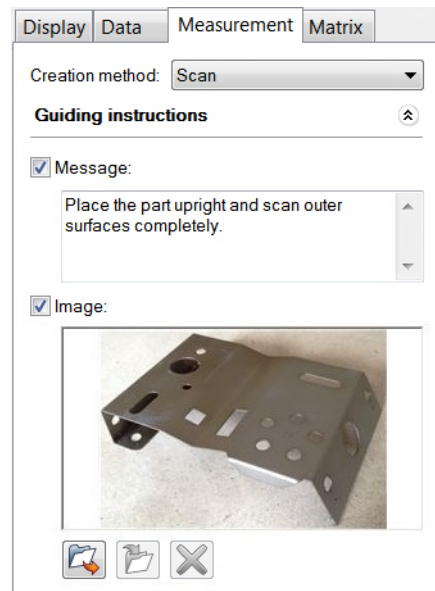
Steps

1. In the Tree View, select a Data object or a measurement object.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. In the **Guiding instructions** section, select **Message** and/or **Image**.

*For measurement objects, these parameters are offered only when the measurement method is **Custom**, **Undefined**, or **Probe**.*

- In the **Message** area, enter custom instructions.
- Under the **Image** area, click the import button and load a drawing or image file that illustrates the instructions. The image is displayed in the viewing area.

5. Click **Apply**.



Results

The custom message and image are displayed during Play Inspection.



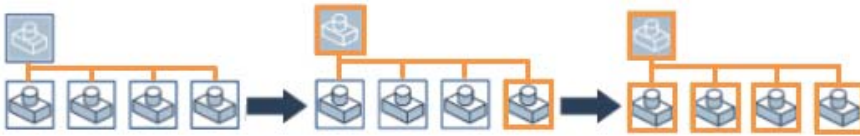
Notes

Custom guiding instructions can also be specified for Data alignment and Data color map objects. These parameters are available in the property sheet of the object, respectively on the **Data alignment** and the **Data color map** tabs.

Propagating piece changes

It is possible to modify or add certain information for multiple measured pieces at once, such as to modify or add data alignments, measurement objects, or inspection reports. This can be useful, for example, to easily correct an error among many pieces that have already been scanned, such as changing tolerances on a geometry control, without having to make that change to each piece manually.

To be able to propagate changes, pieces must first be linked to a piece template. Changes can then be propagated through the linked piece template to the other linked pieces when desired.



Changes can be propagated to previous pieces, subsequent pieces, or a subset of pieces if they are linked to a piece template. More than one template can also be used. An icon indicates whether the current piece or piece template is unlinked or linked.



Linking pieces to a piece template

Objective: Link a new piece to a piece template.

Required: A piece template has already been created.

Steps

1. Choose *Tools > Piece > New* to create a new piece.



The New Piece dialog box opens.

The piece name as well as the creation date and time are automatically specified.

- Choose **Piece template** as the **Source piece** and select the template used to create the new piece.
- The **Link to the piece template** check box is selected by default and links the new piece to the selected piece template.
- If required, specify other standard information for the new piece.
- Click **Create**.

Source piece:

Current piece

Piece template: template 1

Link to the piece template

Piece template properties

Piece properties

Property	Value
Piece name	piece 2
Date	5/29/2020
Time	7:16:45 AM
Serial number	
Order number	
Operator name	
E-Mail address	
Device	

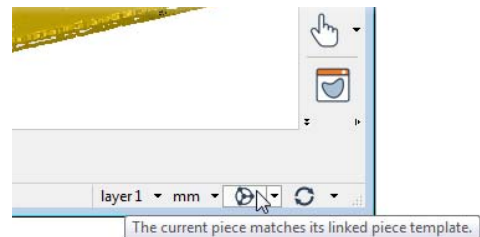
Fixed alignment setup

Options

Create Cancel

Results

The piece that is created is linked to the specified piece template. The linked icon is displayed on the status bar.

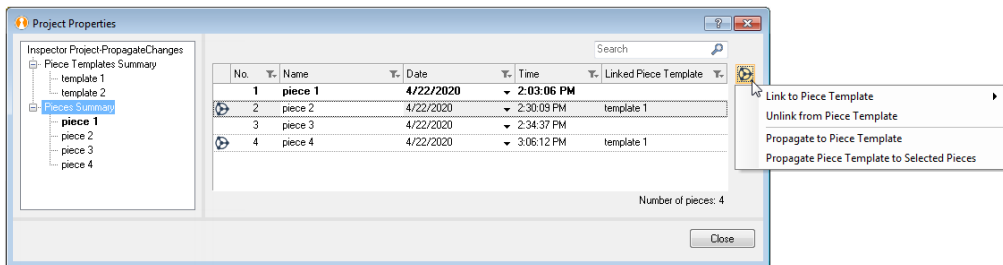


Notes

Propagation links can also be established and managed after a piece or piece template has been created, for example to unlink pieces from a piece template.

This can be done using several tools, including the **Pieces Summary** page of the Project Properties dialog box, shown below, which is available by choosing *File > Project Properties*.

Alternatively, use the **Piece Properties** button.



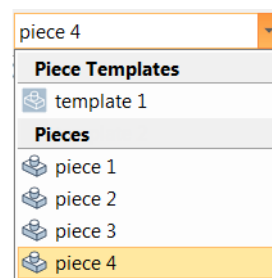
Propagating changes to pieces

Objective: Propagate changes made to the current piece to other linked pieces to correct an error.

Required: Pieces are linked to the same piece template.

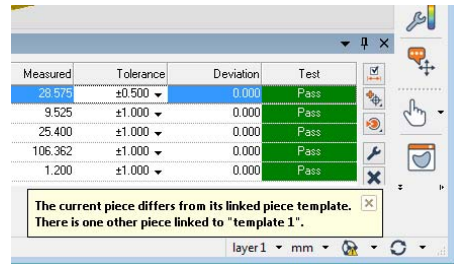
Steps

1. Open a piece.

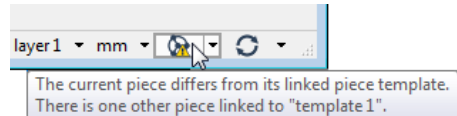


2. Make the desired changes. For example, change the tolerances on any control using the Geometry Controls pane.

A notification is displayed above the status bar when beginning to make changes that can be propagated.



The propagation split button on the status bar displays the propagation status, warning that the current piece now differs from its linked piece template.

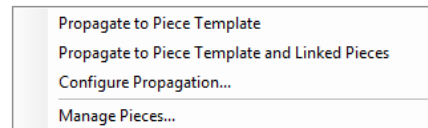


Moving the mouse pointer over the icon displays a tooltip with details.

3. Click the propagation split button and choose *Propagate to Piece Template*. Changes will be made to each linked piece only when they are opened.



*Alternatively, choosing *Propagate to Piece Template and Linked Pieces* makes the changes to the linked pieces immediately. Note that this is a long operation, depending on the amount of linked pieces.*



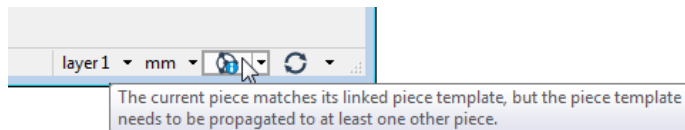
The propagation operation cannot be undone.

Results

The changes made to the current piece are immediately propagated to the linked piece template. These changes are also made to the other linked pieces when they are opened.

The propagation split button on the status bar displays the propagation status. The current piece now matches its linked piece template.

Moving the mouse pointer over the icon displays a tooltip with details.

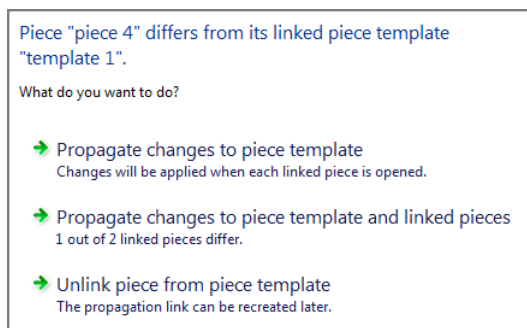


Notes

- The following changes can be propagated:
 - Nominal information that has been added or modified, such as information on the CAD model, nominal components of measurement objects, and tolerances.
 - Some changes that have an impact on a project's measured results and the way those results are reported, such as modifications made to extraction parameters or objects, control views, or formatted reports that have been added or deleted.

Note that if any of these changes are made, they are all propagated if the propagation operation is launched.

- If changes that can be propagated are made but not propagated before closing the current linked piece, the Piece Propagation window is displayed and offers different operations depending on the context.



Guided feature probing using measurement points

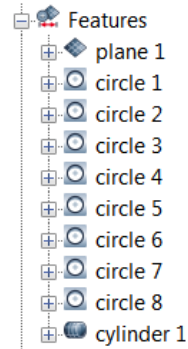
In metrology, it is important to reduce the number of factors that could affect measurement results. For example, when performing a part inspection by probing, controlling the location of the points that are to be probed is sometimes desired.

This is especially useful when setting up a project for a multipiece inspection process, where multiple instances of a part are inspected. In order to obtain meaningful results, each measurement of the part must be performed in the exact same way.

When probing features, the repeatability of each measurement depends largely on the number and the location of the probed data points of each feature. Adding measurement points to features provides guidance as to where the points should be probed, instead of randomly probing points on the part for every new inspection.

There are three main steps when using measurement points for guidance:

1. Features are created.



2. The measurement method is set to **Probe**, the **Guided** point acquisition mode is configured, and measurement points are defined for each feature that will be probed.



3. When measuring the piece, the features are probed using measurement points as guidance.



The Measurement Points pane

When using measurement points for guidance, the Measurement Points pane can be used to customize the measurement points used for the probing operation. To open the Measurement Points pane:

1. In the Tree View, select the desired feature.
2. Choose *Edit > Object Properties*.
The property sheet is opened.
3. On the **Measurement** tab, make sure that:
 - The measurement method is set to **Probe**.
 - The point acquisition mode is set to **Guided**.
4. Under **Point acquisition**, press **Measurement Points**.



The Measurement Points pane is opened: columns provide information on measurement points, and a vertical toolbar offers operations used to create, edit, and delete measurement points:

Columns

Vertical toolbar

No.	Type	X	Y	Z	I	J	K
Local plane							
1		121.825	25.400	97.290	0.000	1.000	0.000
2		121.825	25.400	82.827	0.000	1.000	0.000
3		134.350	25.400	90.059	0.000	1.000	0.000
Circle							
4		132.350	23.400	90.059	-1.000	0.000	0.000
5		122.825	23.400	84.559	0.500	0.000	0.866
6		122.825	23.400	95.558	0.500	0.000	-0.866

The vertical toolbar offers the following buttons:



Define Measurement Points from Probed Data Points

Defines measurement points from previously probed data points.



Anchor Measurement Points

Launches an interactive mode that allows creating measurement points by anchoring them on features.

**Pick Go to Positions**

Can only be used when the measurement method is set to **CNC Probe** (in other words, when automatically probing using a CNC CMM).

Launches an interactive mode that allows adding intermediate positions, or go to position points, between measurement points along the measurement path to avoid collisions with the part or its fixture.

**Edit Points**

Opens a dialog box used to edit the position and normal of selected measurement points.

**Drag Points**

Launches an interactive mode that allows repositioning measurement points by dragging them in the 3D scene.

**Delete Points**

Deletes selected measurement points.

**Delete Points Interactively**

Launches an interactive mode that allows deleting measurement points by clicking them in the 3D scene.

Configuring the guided point acquisition mode

It is possible to configure whether the acquisition of a feature's probed data points is guided or not. The guided point acquisition mode is used to define measurement points at the locations and the order in which data points should be probed.

Once the guided acquisition mode is set, parameters are made available to configure the acquisition.

Parametric information that defines the nominal components of features can be used to define the measurement points.

Objective: Configure the guided acquisition mode for features.

Required: The project contains features with at least the nominal primitives.

Steps

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.

In the **Point acquisition** section:

5. In the **Mode** list, select **Guided**.
6. Set the point acquisition **Order**.
 - Select **Sequential** to specify that the measurement points will be used to guide point acquisition in the order of the measurement points list.
 - Select **Auto-Detect** to specify that the measurement points will be used to guide point acquisition in any order.
7. Set the **Guiding sphere radius**.

Sets the size of the spheres used to guide the user to the location of the measurement points while probing.

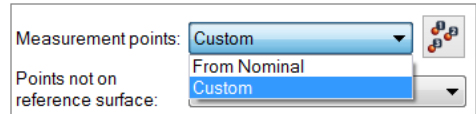
8. If **Limit acquisition to radius** is selected, points can only be acquired when the probe is within the specified radius.

The screenshot shows the 'Point acquisition' dialog box with the following settings:

- Mode:** Guided
- Order:** Auto-Detect
- Guiding sphere radius:** 2.000
- Limit acquisition to radius**
- Measurement points:** From Nominal

9. Set the **Measurement Points** parameter.

- Select **From Nominal** to use the nominal component of the feature to define the measurement points.



Measurement points: Custom

Points not on reference surface: Custom

See the following sections for the parameters related to each feature type.

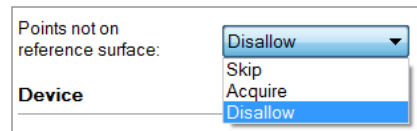
- Select **Custom** to use probed points of the measured component of the feature or to anchor measurement points directly in the 3D scene.



Then, click **Measurement Points** to the right of the list box to open the Measurement Points pane to define and edit measurement points. See [Defining measurement points from the nominal component](#) on page 615.

10. In the **Points not on reference surface** list box, specify the behavior when one or more defined measurement points are not on a reference surface:

- Select **Disallow** to prevent the CNC probe mode from launching when some points are not on a reference surface.



Points not on reference surface: Disallow

Device: Disallow

To correct this situation, edit the point acquisition parameters or edit the measurement points through the Measurement Points pane.

- Select **Skip** to skip the points in error.

A minimum number of points must be probed to form a valid feature.

- Select **Acquire** to attempt to probe the points in error.

This may result in acquisition failure. If there is no reference surface at all, this behavior is used.

11. Click **Apply**.

Measurement points are added to the feature based on the set parameters.

Defining measurement points from the nominal component

When defining measurement points using **From Nominal**, the **Point acquisition** section offers both general and feature-specific parameters that are used to configure the measurement points defined from the nominal component of a feature.

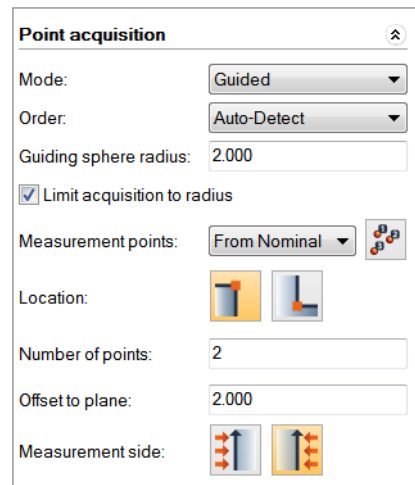
Setting the parameters for line features

Objective: Define measurement points from the nominal component of a feature.

Required: The project contains a feature with a nominal component.

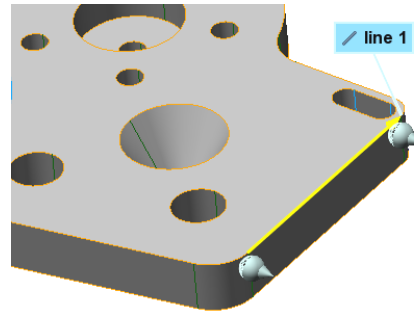
Steps

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Set the **Location**, which specifies the location of the feature on the part:
 - Click **Convex Edge** when the line is located on an edge with an angle pointing outwards.
 - Click **Concave Edge** when the line is on an edge with an angle pointing inwards.
8. Enter the **Number of points** that are evenly distributed along the line.
9. Specify the **Offset to plane** distance between the feature plane and the measurement points. The offset direction is based on the **Location**.
10. Set the **Measurement side**:
 - Click **Left** when the measurement points are located on the left side of the line and the material is on the right side of the line.
 - Click **Right** when the measurement points are located on the right side of the line and the material is on the left side of the line.
11. Click **Apply** and close the Properties dialog box.



Results

Measurement points are defined using the nominal primitive of the selected line.



Setting the parameters for plane features

Objective: Define measurement points from the nominal component of a feature.

Required: The project contains a feature with a nominal component.

Steps

Along Boundary

This pattern is used to distribute measurement points along the external boundary of the plane.

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Set the pattern to **Along Boundary**.
8. Specify the **Number of points** distributed along the boundary.

Point acquisition	
Mode:	Guided
Order:	Auto-Detect
Guiding sphere radius:	2.000
<input checked="" type="checkbox"/> Limit acquisition to radius	
Measurement points:	From Nominal
Pattern:	Along Boundary
Number of points:	3

Results

Measurement points are defined using the nominal primitive of the selected plane.



Grid

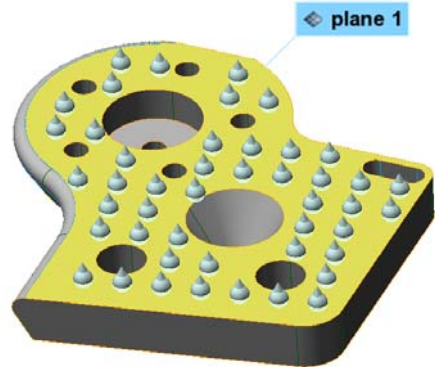
This pattern defines uniformly-spaced measurement points by sampling a grid that covers the plane.

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Set the pattern to **Grid**.
8. Set the **Sampling step** value of the grid.

Point acquisition	
Mode:	Guided
Order:	Auto-Detect
Guiding sphere radius:	4.000
<input checked="" type="checkbox"/> Limit acquisition to radius	
Measurement points:	From Nominal
Pattern:	Grid
Sampling step:	10.000

Results

Measurement points are defined using the nominal primitive of the selected plane.



Setting the parameters for circle features

Objective: Define measurement points from the nominal component of a feature.



Required: The project contains a feature with a nominal component.

Steps

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.

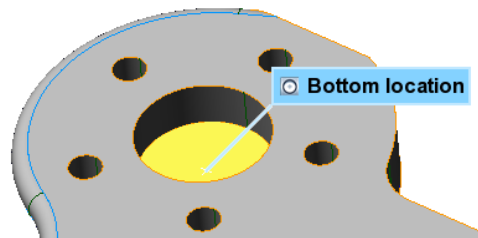
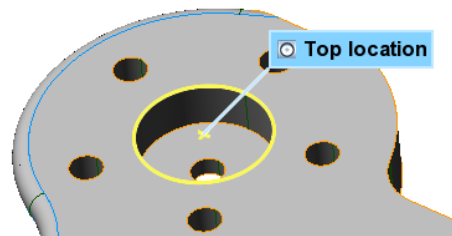
7. Set the **Location**, which specifies the location of the feature on the part:
 - Click **Top** when the circle is located at the top of the part geometry.
 - Click **Bottom** when the circle is located at the bottom of the part geometry.

The in-space orientation considered for the circle feature comes from its nominal component.
8. Enter the **Number of points** that are evenly distributed along the circle.
9. Specify the **Offset to plane** distance between the plane of the circle and the measurement points. The offset direction is based on the **Location**.
10. Click **Apply** and close the Properties dialog box.

Point acquisition	
Mode:	Guided
Order:	Auto-Detect
Guiding sphere radius:	2.000
<input checked="" type="checkbox"/> Limit acquisition to radius	
Measurement points:	From Nominal
Location:	 
Number of points:	3
Offset to plane:	2.000

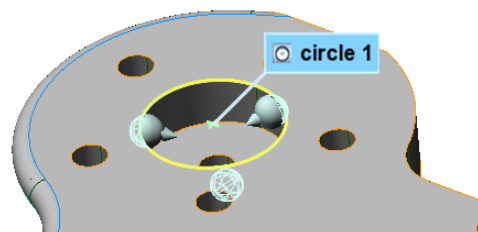
Tip

- Top location of an internal circle.
- Bottom location of an internal circle.



Results

Measurement points are defined using the nominal primitive of the selected circle.



Notes

The icons displayed for the **Location** are based on the in-space orientation of the nominal component of the feature.

For an external circle, the **Top** and **Bottom** location icons are shown to the right.



Setting the parameters for slot features

Objective: Define measurement points from the nominal component of a feature.

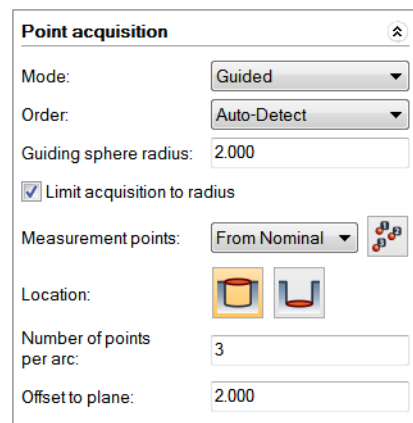
Required: The project contains a feature with a nominal component.

Steps

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Set the **Location**, which specifies the location of the feature on the part:
 - Click **Top** when the slot is located at the top of the part geometry.
 - Click **Bottom** when the slot is located at the bottom of the part geometry.

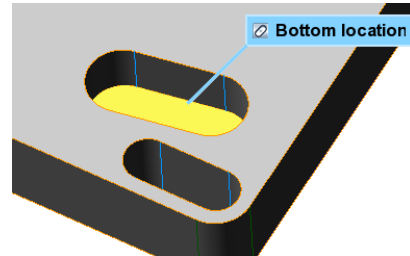
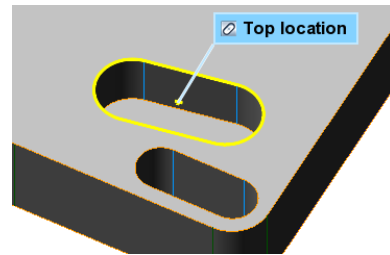
The in-space orientation considered for the slot feature comes from its nominal component.

8. Enter the **Number of points per arc** that are evenly distributed along each arc.
9. Specify the **Offset to plane** distance between the plane of the slot and the measurement points. The offset direction is based on the **Location**.
10. Click **Apply** and close the Properties dialog box.



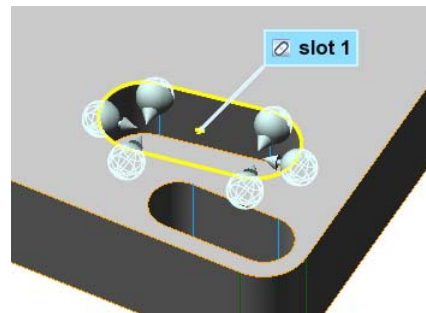
Tip

- Top location of an internal slot.
- Bottom location of an internal slot.



Results

Measurement points are defined using the nominal primitive of the selected slot.



Notes

The icons displayed for the **Location** are based on the in-space orientation of the nominal component of the feature.

For an external slot, the **Top** and **Bottom** location icons are shown to the right.

Location:



Setting the parameters for rectangle features

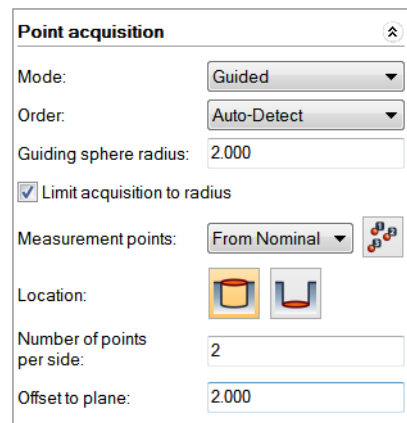
Objective: Define measurement points from the nominal component of a feature.

Required: The project contains a feature with a nominal component.

Steps

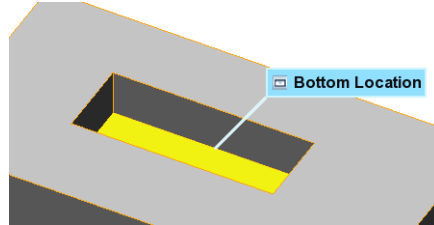
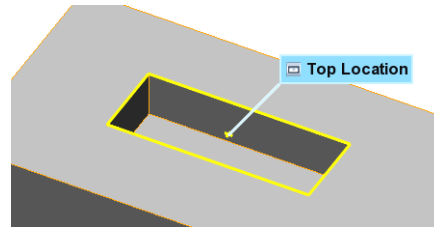
1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Set the **Location**, which specifies the location of the feature on the part:
 - Click **Top** when the rectangle is located at the top of the part geometry.
 - Click **Bottom** when the rectangle is located at the bottom of the part geometry.

The in-space orientation considered for the rectangle feature comes from its nominal component.
8. Enter the **Number of points per side** that are evenly distributed along each side.
9. Specify the **Offset to plane** distance between the plane of the rectangle and the measurement points. The offset direction is based on the **Location**.
10. Click **Apply** and close the Properties dialog box.



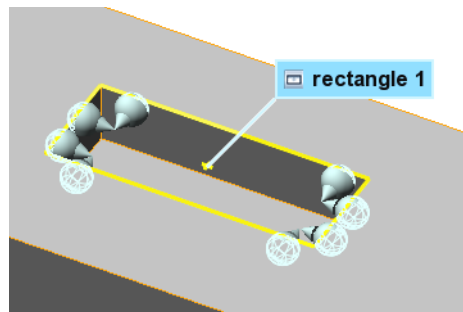
Tip

- Top location of an internal rectangle.
- Bottom location of an internal rectangle.



Results

Measurement points are defined using the nominal primitive of the selected rectangle.



Notes

The icons displayed for the **Location** are based on the in-space orientation of the nominal component of the feature.

For an external rectangle, the **Top** and **Bottom** location icons are shown to the right.



Setting the parameters for polygon features



Objective: Define measurement points from the nominal component of a feature.

Required: The project contains a feature with a nominal component.

Steps

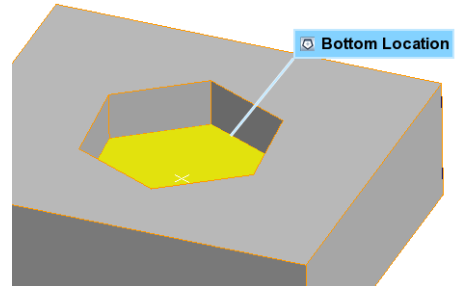
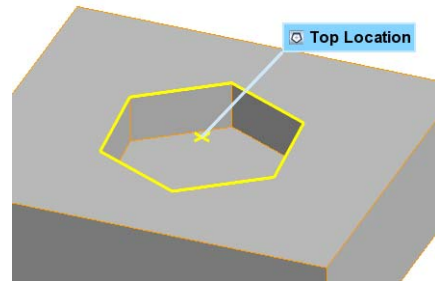
1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Set the **Location**, which specifies the location of the feature on the part:
 - Click **Top** when the polygon is located at the top of the part geometry.
 - Click **Bottom** when the polygon is located at the bottom of the part geometry.

The in-space orientation considered for the polygon feature comes from its nominal component.
8. Enter the **Number of points per side** that are evenly distributed along each side.
9. Specify the **Offset to plane** distance between the plane of the polygon and the measurement points. The offset direction is based on the **Location**.
10. Click **Apply** and close the Properties dialog box.

Point acquisition	
Mode:	Guided
Order:	Auto-Detect
Guiding sphere radius:	2.000
<input checked="" type="checkbox"/> Limit acquisition to radius	
Measurement points:	From Nominal
Location:	 
Number of points per side:	2
Offset to plane:	2.000

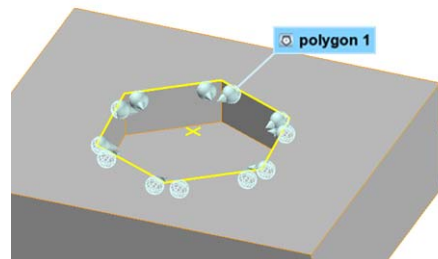
Tip

- Top location of an internal polygon.
- Bottom location of an internal polygon.



Results

Measurement points are defined using the nominal primitive of the selected polygon.



Notes

The icons displayed for the **Location** are based on the in-space orientation of the nominal component of the feature.

For an external polygon, the **Top** and **Bottom** location icons are shown to the right.



Setting the parameters for ellipse features

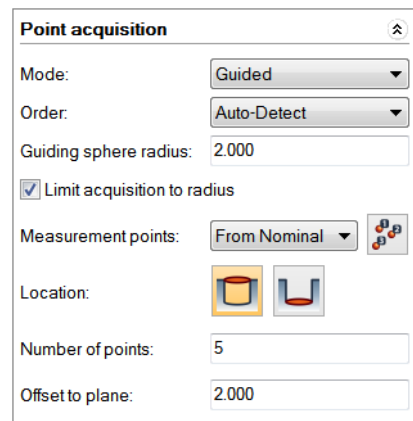
Objective: Define measurement points from the nominal component of a feature.

Required: The project contains a feature with a nominal component.

Steps

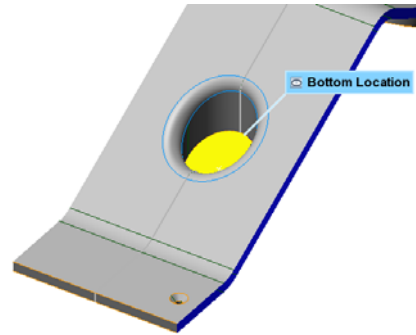
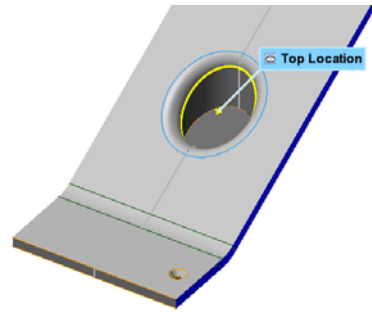
1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Set the **Location**, which specifies the location of the feature on the part:
 - Click **Top** when the ellipse is located at the top of the part geometry.
 - Click **Bottom** when the ellipse is located at the bottom of the part geometry.

The in-space orientation considered for the ellipse feature comes from its nominal component.
8. Enter the **Number of points** that are evenly distributed along the ellipse.
9. Specify the **Offset to plane** distance between the plane of the ellipse and the measurement points. The offset direction is based on the **Location**.
10. Click **Apply** and close the Properties dialog box.



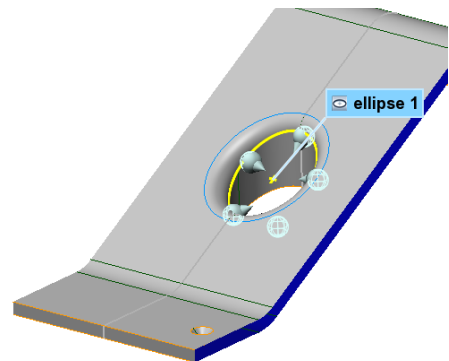
Tip

- Top location of an internal ellipse.
- Bottom location of an internal ellipse.



Results

Measurement points are defined using the nominal primitive of the selected ellipse.



Notes

The icons displayed for the **Location** are based on the in-space orientation of the nominal component of the feature.

For an external ellipse, the **Top** and **Bottom** location icons are shown to the right.



Setting the parameters for cylinder features

Objective: Define measurement points from the nominal component of a feature.

Required: The project contains a feature with a nominal component.

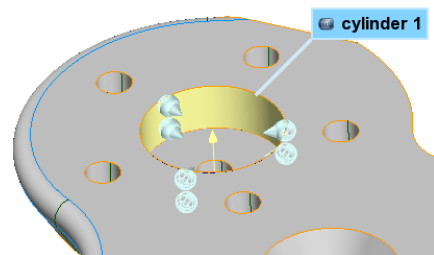
Steps

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Enter the **Number of circles**, which specifies the number of equidistant circles on which measurement points are defined.
8. Enter the **Number of points per circle**, which specifies the number of measurement points that are evenly distributed along each defined circle.
9. Enable the **Max Depth** option, if needed. It allows users to define the maximum distance of the deepest circle. The depth considers the axis orientation of the cylinder.
10. Click **Apply** and close the Properties dialog box.

Point acquisition	
Mode:	Guided
Order:	Auto-Detect
Guiding sphere radius:	2.000
<input checked="" type="checkbox"/> Limit acquisition to radius	
Measurement points:	From Nominal
Number of circles:	2
Number of points per circle:	3
<input type="checkbox"/> Max depth:	10.000

Results

Measurement points are defined using the nominal primitive of the selected cylinder.



Setting the parameters for cone features

Objective: Define measurement points from the nominal component of a feature.

Required: The project contains a feature with a nominal component.

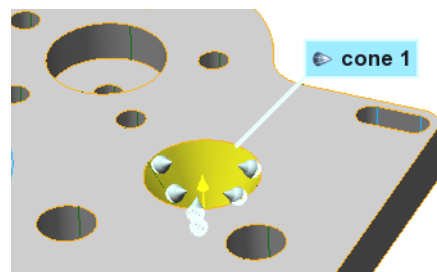
Steps

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Enter the **Number of circles**, which specifies the number of equidistant circles on which measurement points are defined.
8. Enter the **Number of points per circle**, which specifies the number of measurement points that are evenly distributed along each defined circle.
9. Enable the **Max Depth** option, if needed. It allows users to define the maximum distance of the deepest circle. The depth considers the axis orientation of the cone.
10. Click **Apply** and close the Properties dialog box.

Point acquisition	
Mode:	Guided
Order:	Auto-Detect
Guiding sphere radius:	2.000
<input checked="" type="checkbox"/> Limit acquisition to radius	
Measurement points:	From Nominal
Number of circles:	2
Number of points per circle:	3
<input type="checkbox"/> Max depth:	10.000

Results

Measurement points are defined using the nominal primitive of the selected cone.








Setting the parameters for sphere features

Objective: Define measurement points from the nominal component of a feature.

Required: The project contains a feature with a nominal component.

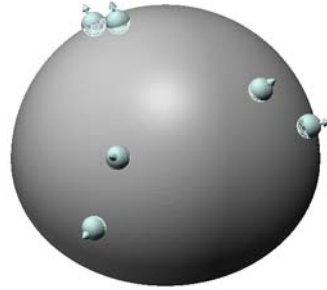
Steps

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Enable **Point on top of the sphere**, if needed.
This creates a measurement point at the top of the sphere.
8. Enter the **Number of circles**, which specifies the number of equidistant circles on which measurement points are defined.
9. Enter the **Number of points per circle**, which specifies the number of measurement points that are evenly distributed along each defined circle.
10. For the **First circle**:
Set the **Angle from top**, at which the first circle is positioned.
11. For the **Last circle**:
Set the **Angle from top**, at which the last circle is positioned.
12. Click **Apply** and close the Properties dialog box.

Point acquisition 	
Mode:	Guided 
Order:	Auto-Detect 
Guiding sphere radius:	2.000
<input checked="" type="checkbox"/> Limit acquisition to radius	
Measurement points:	From Nominal  
<input type="checkbox"/> Point on top of the sphere	
Number of circles:	2
Number of points per circle:	3
First circle:	
Angle from top:	25.000
Offset from equator:	13.595
Last circle:	
Angle from top:	45.000
Offset from equator:	10.607

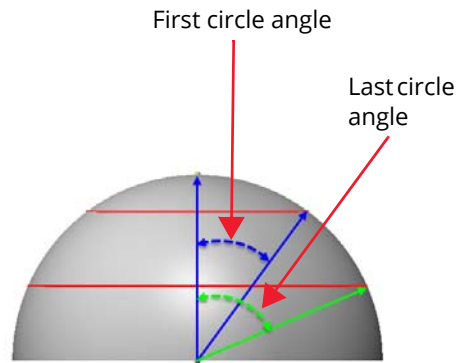
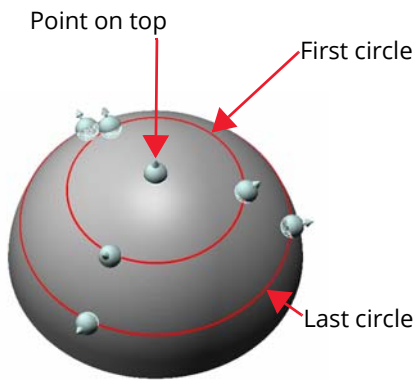
Results

Measurement points are defined using the nominal primitive of the selected sphere.



Notes

The parameters for the sphere are illustrated below.



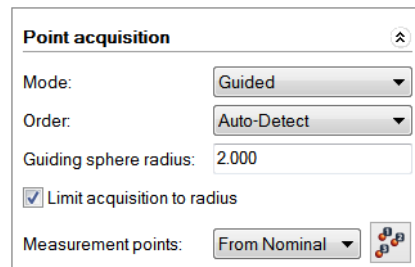
Setting the parameters for polyline features

Objective: Define measurement points from the nominal component of a feature.

Required: The project contains a feature with a nominal component.

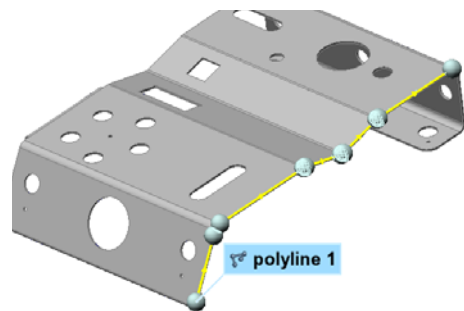
Steps

1. In the Tree View, select the feature to which measurement points will be added.
2. Choose *Edit > Object Properties*.
The Properties dialog box opens.
3. Click the **Measurement** tab.
4. Set the **Method** to **Probe**.
5. Set the point acquisition **Mode** to **Guided**.
6. Set the **Measurement points** to **From Nominal**.
7. Click **Apply** and close the Properties dialog box.



Results

Measurement points are defined at each vertex of the nominal primitive of the selected polyline.



Notes

Since polylines do not have surface information, each measurement point only contains a location (x, y, and z coordinates) but no orientation.

No.	X	Y	Z	I	J	K
1	186.779	0.038	32.938			
2	170.192	0.224	4.212			
3	165.339	0.000	0.000			
4	107.293	0.000	0.000			
5	106.274	0.000	0.125			

Defining custom measurement points

The Measurement Points pane can be used to define, visualize, and edit measurement points on features. Two creation methods are offered:

- **Define Measurement Points from Probed Data Points:** Measurement points are defined automatically from probed data points.
- **Anchor Measurement Points:** Measurement points are anchored directly on the feature or on a Reference object in the 3D scene.

Measurement points are only available for features that support the **Probe** or the **CNC Probe** measurement method, and they are specific to each feature.

Defining measurement points from probed data points

Objective: Define measurement points from probed data points for selected features.

Required: The selected feature must have probed data points.

Steps

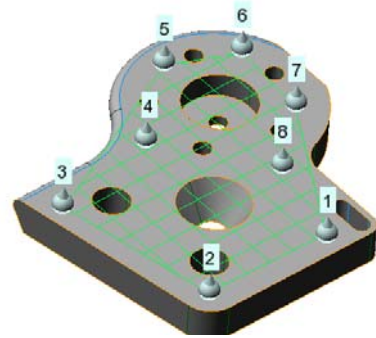
1. Open the Measurement Points pane:
 - by clicking **Measurement Points** in the **Properties** of the feature (see [Configuring the guided point acquisition mode](#) on page 613).
 - by right-clicking the feature and choosing *Edit > Measurement Points*.
2. Click **Define Measurement Points from Probed Data Points**.



Results

Measurement points are defined using probed data points obtained from the selected feature.

No.	Type	X	Y	Z	I	J	K
1	●	137.513	25.400	51.320	0.000	1.000	0.000
2	●	135.988	25.400	98.534	0.000	1.000	0.000
3	●	77.732	25.400	110.5...	0.000	1.000	0.000
4	●	69.789	25.400	69.635	0.000	1.000	0.000
5	●	45.687	25.400	42.148	0.000	1.000	0.000
6	●	55.400	25.400	18.026	0.000	1.000	0.000
7	●	85.039	25.400	20.395	0.000	1.000	0.000
8	●	104.449	25.400	42.209	0.000	1.000	0.000



Defining measurement points by anchoring

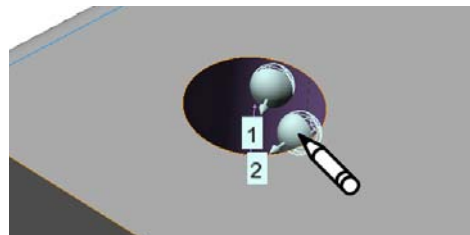
Objective: Define measurement points by anchoring points on features.

Required: The PolyWorks | Inspector project must have one or more features that support the **Probe** measurement method.

Steps

- Open the Measurement Points pane:
 - by clicking **Measurement Points** in the **Properties** of the feature (see [Configuring the guided point acquisition mode](#) on page 613)
 - by right-clicking the feature and choosing *Edit > Measurement Points*
- Click **Anchor Measurement Points** to launch an interactive anchoring mode.
- In the 3D scene, click directly on the feature to create the measurement points.

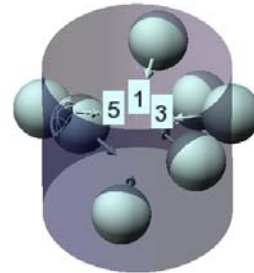
The number of measurement points must be equal to or larger than the minimum number of points for that feature type.
- Right-click to exit the anchoring mode.



Results

Measurement points are defined on the feature.

No.	Type	X	Y	Z	I	J	K
1	●	82.438	22.292	91.856	0.583	0.000	0.813
2	●	85.329	18.167	90.799	0.130	0.000	0.992
3	●	89.584	20.354	91.872	-0.583	0.000	0.813
4	●	89.825	17.793	102.179	-0.583	0.000	-0.813
5	●	81.046	20.905	101.082	0.773	0.001	-0.635
6	●	79.688	15.888	96.430	0.995	-0.001	0.098
7	●	92.324	21.372	97.652	-0.995	0.000	-0.098



Anchoring points on features that have arcs or sides

Slot, rectangle, and polygon features are probed using individual arcs or sides. To anchor measurement points for these features, the arc or side associated with each measurement point must be specified in order to provide the appropriate guidance. The Anchor Measurement Points dialog box is used to specify which arc or side the measurement point will be anchored on.

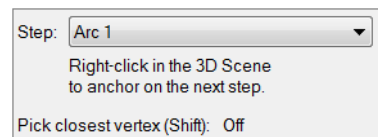
Steps

1. In the Tree View, select the slot, rectangle, or polygon feature to which measurement points will be added.
2. In the Measurement Points pane, click **Anchor Measurement Points** to launch an interactive anchoring mode.



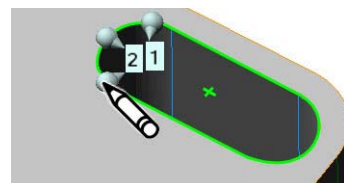
The Anchor Measurement Points dialog box opens.

3. From the **Step** list, select the corresponding arc or side for which measurement points will be anchored.

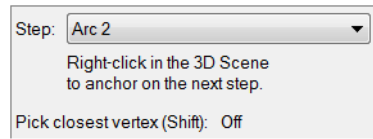


4. In the 3D scene, click directly on the feature or on the Reference object to anchor the measurement points at the desired locations.

The number of measurement points must be equal to or larger than the minimum number of points.



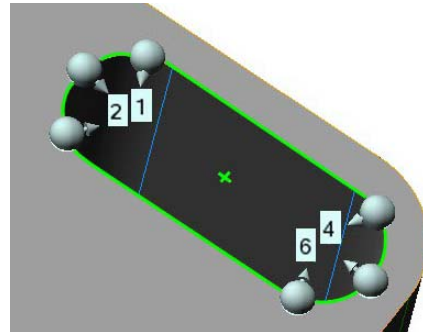
- From the list, select the next arc or side and continue anchoring the points on the feature.
Repeat the procedure for the other arcs or sides.
- Right-click to exit the anchoring mode.



Results

Measurement points are defined on the feature for each arc or side.

No.	Type	X	Y	Z	I	J	K
Arc 1							
1	●	130.723	25.400	36.837	0.087	0.000	0.996
2	●	127.825	25.400	40.000	1.000	0.000	0.000
3	●	130.723	25.400	43.163	0.087	0.000	-0.996
Arc 2							
4	●	141.277	25.400	43.163	-0.087	0.000	-0.996
5	●	144.175	25.400	40.000	-1.000	0.000	0.000
6	●	141.277	25.400	36.837	-0.087	0.000	0.996



Editing and deleting measurement points

The Measurement Points pane offers various tools to edit and delete measurement points.

Editing measurement points numerically

Measurement points can be edited numerically by editing their position and normal.

Steps

- In the Measurement Points pane, select the measurement point that will be edited.
Use a multiple selection to edit more than one measurement point at a time.
- Click **Edit Points**.



The Edit Points dialog box opens.

- Specify the **Position** and **Normal** for the measurement point.

The measurement point x, y, or z coordinates can also be edited directly in the Measurement Points pane. Click a coordinate to edit its value.

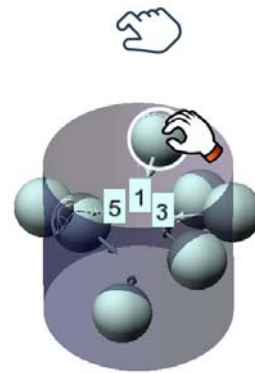
Position	
X:	127.902
Y:	25.400
Z:	39.305
Normal	
I:	0.976
J:	-0.000
K:	0.219
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Editing measurement points interactively

Measurement points can be edited interactively by dragging them in the 3D scene.

Steps

- Click **Drag Points** to launch an interactive dragging mode.
- Reposition measurement points by dragging them directly in the 3D scene.
- Right-click to exit the dragging mode.



Editing arcs and sides of measurements points

In cases where a measurement point was anchored to the wrong side or arc, it is possible to correct this and change the arc or the side the measurement points belong to.

Steps

1. In the Measurement Points pane, click the measurement point that is on the wrong arc or side and drag it to the correct arc or side.

Measurement Points - slot 1						
No.	X	Y	Z	I	J	K
Arc 1						
130.293	25.400	36.905	0.223	-0.000	0.975	
127.902	25.400	39.305	0.976	-0.000	0.219	
129.575	25.400	42.837	0.449	-0.000	-0.894	
Arc 2						
141.411	25.400	36.852	-0.129	0.000	0.992	
144.125	25.400	39.441	-0.984	0.000	0.176	

Deleting selected measurement points

Measurement points can be deleted by selecting them and launching the deletion operation.

Steps

1. In the Measurement Points pane, select the measurement points that will be deleted.
Use a multiple selection to delete more than one measurement point at a time.
2. Click **Delete Points**.

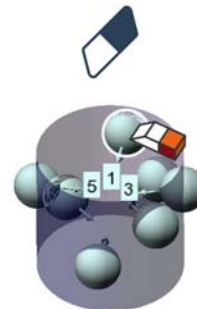


Deleting measurement points interactively

Measurement points can be deleted interactively.

Steps

1. Click **Delete Points Interactively** to launch an interactive deleting mode.
2. Click the measurement points in the 3D scene to delete them.
3. Right-click to exit the mode.



Probing features using measurement points

Once the guided acquisition mode has been configured (*Configuring the guided point acquisition mode* on page 613) and the measurement points have been defined (*Defining measurement points from the nominal component* on page 615), via the properties of the feature, the probing in a guided mode can be launched.

The order in which the points are probed is controlled by the specified acquisition **Order**:

- **Sequential:** The next measurement point is the first unmeasured point in the list of measurement points.
- **Auto-Detect:** The next measurement point is the one closest to the probing device.

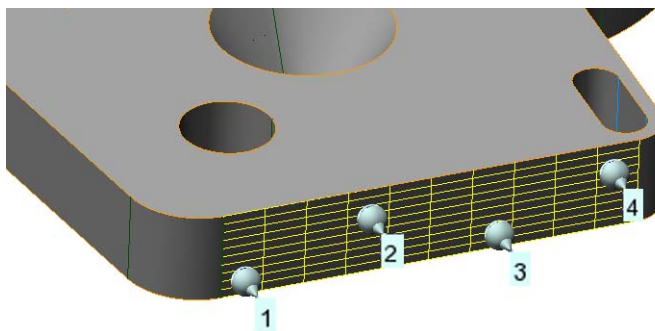
The measurement point guiding spheres are highlighted in the 3D scene. The **Limit acquisition to radius** option allows restricting the point acquisition for each measurement point to a precise zone delimited by cylinders or by spheres (for point with no normal) centered at the measurement points. The next measurement point guiding sphere is always highlighted as follows in the 3D scene:

- If the **Limit acquisition to radius** option is set, it is highlighted in red if the probing device is out of the zone, and in green if it is in the zone. A sound also indicates if the probing device is within the zone or not.
- If the **Limit acquisition to radius** option is not set, the guiding zone of the measurement point closest to the probing device is highlighted in green.

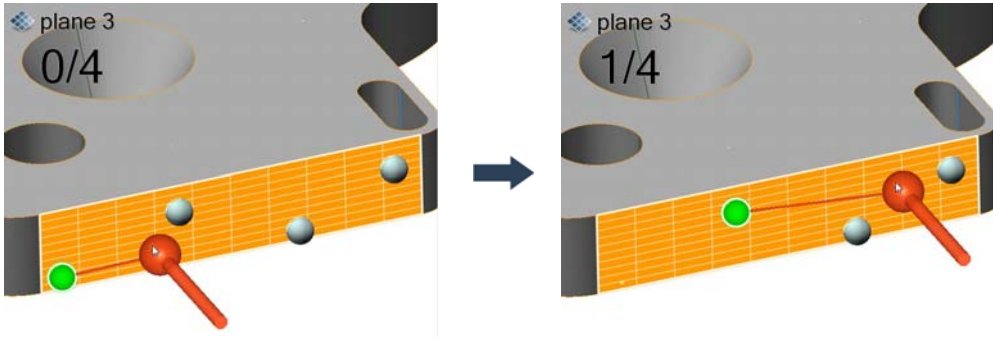
Once a measurement point has been used for point acquisition, its guiding zone is no longer displayed in the 3D scene.

Example

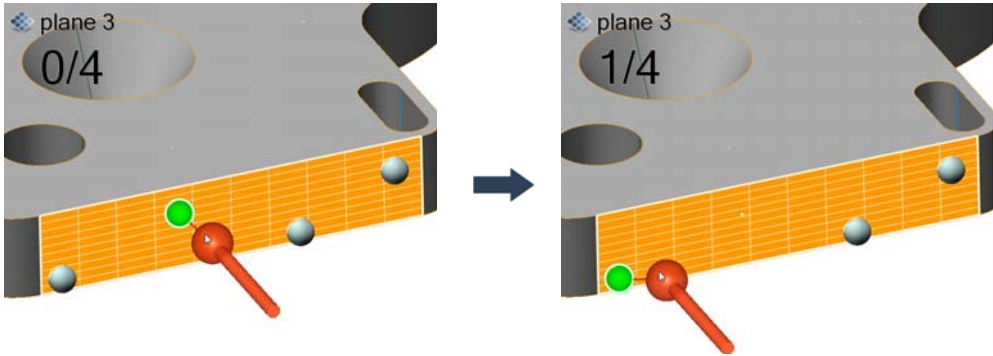
Defined measurement points on a plane feature.



Probing a plane feature in the **Sequential** order. The points are probed in numerical order.



Probing a plane feature in the **Auto-Detect** order. The points are probed in any order.



Using measurement points with features probed in multiple device positions

Sometimes, features need to be probed from multiple device positions. This can be due to the size of the part or the physical constraints that make it impossible to access a feature from a certain position.

Defining measurement points using the **From Probed Data Points** method will automatically group the measurement points, taking into consideration the device positions in which they were gathered.

No.	Type	X	Y	Z	I	J	K
Device position point group 1							
1	●	48.318	25.400	47.311	0.000	1.000	-0.002
2	●	61.147	25.345	20.469	0.000	1.000	-0.002
3	●	109.818	25.403	45.394	0.000	1.000	-0.002
Device position point group 2							
4	●	136.486	25.416	50.336	0.000	1.000	-0.002
5	●	134.161	25.524	101.960	0.000	1.000	-0.002
6	●	81.259	25.538	111.750	0.000	1.000	-0.002

Objective: Probe a feature from multiple device positions using measurement points as guidance.

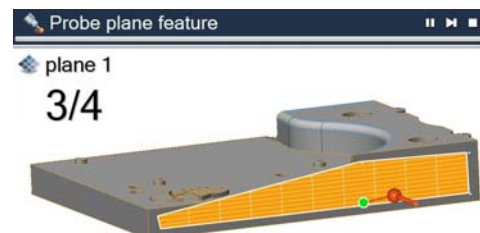
Required: A PolyWorks | Inspector project that contains at least one feature with defined measurement points grouped in different device positions.

Steps

1. Open the project and click **Play Inspection**.

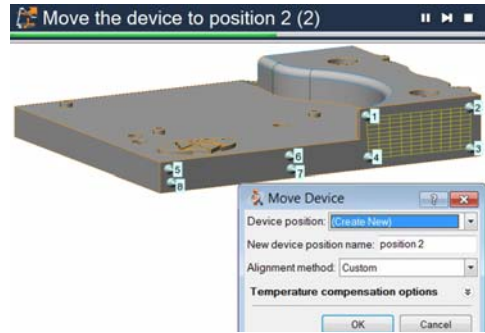


2. Probe the points in the first device position.

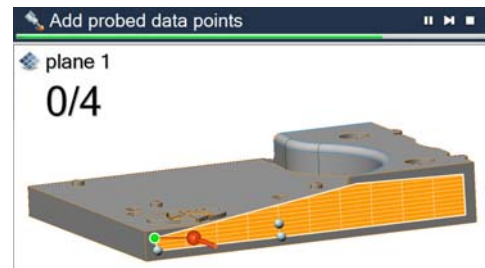


3. Move the device.

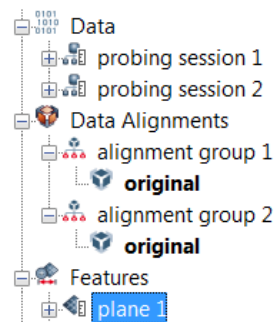
For more information on multiple device positions, see [Working with multiple device positions](#) on page 666.



4. Continue the Play Inspection sequence.
5. Probe points in the second device position.



6. If more than two device positions are required, repeat steps 3 to 5 until the feature is completely probed.



Results

When playing the inspection of a project that contains a feature originally created by probing, using multiple device positions and the **From Probed Data Points** method measurement points, PolyWorks prompts the user to probe the feature displaying only the measurement points that belong to the first position. Then the sequence is paused and the user is prompted to move the device. Once the new device position is created, the user is prompted to probe the feature displaying corresponding measurement points.

Introducing the Sequence Editor

The measurement objects needed for the inspection are listed in the Tree View pane. These objects contain all the information related to how the measurements must take place and the links that exist between the objects. In multipiece inspection, the Play Inspection tool plays an autogenerated measurement sequence, by default, that is based on the objects in the Tree View. It can also play a measurement sequence that has been customized through the Sequence Editor.

The Sequence Editor can be used to customize the autogenerated sequence or to create a custom sequence where steps can be added one by one. Instructions for the operator can be added, the order of measurements can be changed, and additional steps for measuring can be included. Also, by inserting macro scripts, many functionalities can be added, such as those related to reporting.

The Sequence Editor pane

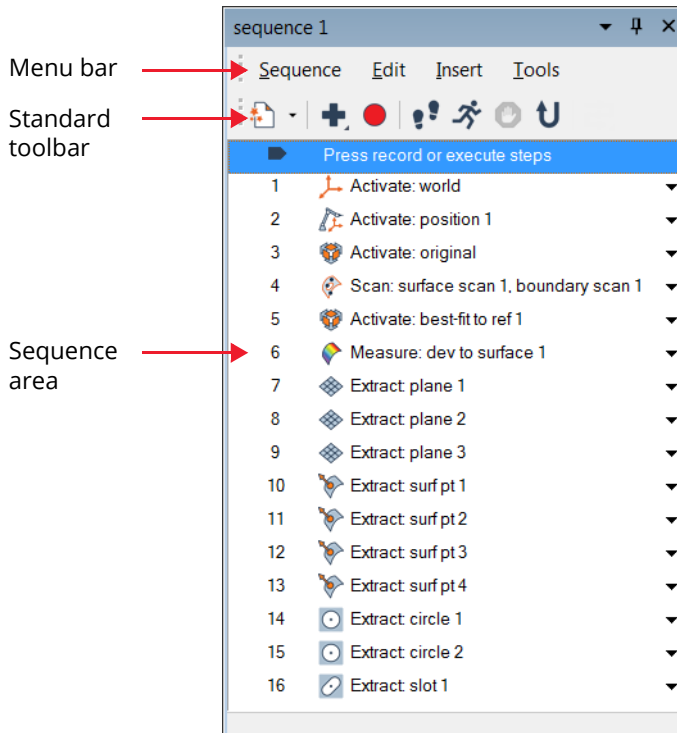
To open the Sequence Editor pane:

1. Choose *Tools > Sequence Editor*.

*Alternatively, click the **Sequence Editor** button from the Multipiece Inspection toolbar.*



The Sequence Editor pane is displayed. It features a menu bar, a standard toolbar, and a sequence area.



Menu bar

Provides access to all Sequence Editor functionalities.

Standard toolbar

Provides quick access to the main functionalities of the Sequence Editor.



Create From Autogenerated Sequence / New Sequence

Split button used to create a new sequence, either from the autogenerated sequence or from scratch.



Insert Step

Displays the Insert toolbar (see [Insert toolbar](#) on page 645) that offers quick access to the steps that can be added to the sequence.



Start Recording

Starts the recording mode.



Execute One by One From Marker

Executes the next step in the sequence based on the marker position.



Execute From Marker

Executes steps starting at the marker position.



Stop Execution

Stops the execution of the sequence.



Reset Execution Status

Removes all information concerning step execution. The marker is positioned at the beginning of the sequence.

Insert toolbar

To open the Insert toolbar, click the **Insert** button on the Standard toolbar. It offers quick access to the steps that can be added to the sequence.



Guiding Instructions

Inserts a step used to add text and/or an image that will be displayed when running the sequence.



Question

Inserts a question step composed of a question and an answer variable.



Conditional Statement for Selected Steps

Inserts a conditional statement based on the content of an answer variable.



Comment

Inserts text to provide useful information about the sequence.



Scanning Options

Inserts a step to change the scanning profile or customize scanning options.



Coordinate System

Inserts a step to activate a different coordinate system in the sequence.



Alignment

Inserts a step to create or activate a data alignment.



Device Position

Inserts a step to activate a device position or to perform a move device operation.



Data Object

Inserts a step to acquire a Data object.



Measurement Object

Inserts a step to measure an object.



Add Probed Data Points

Inserts a step to add probed data points to a specified object.



Macro Script

Inserts a macro script by creating a new one or by importing a copy of a script.

Sequence area

This area lists all the steps that make up a sequence.

1	Activate: world	▼
2	Activate: position 1	▼
3	Activate: original	▼
4	Scan: surface scan 1, boundary scan 1	▼
5	Activate: best-fit to ref 1	▼
Press record or execute steps		
6	Measure: dev to surface 1	▼
7	Extract plane 1	▼
8	Extract plane 2	▼
9	Extract plane 3	▼
10	Extract surf pt 1	▼
11	Extract surf pt 2	▼
12	Extract surf pt 3	▼
13	Extract surf pt 4	▼
14	Extract circle 1	▼
15	Extract circle 2	▼
16	Extract slot 1	▼

Marker

The marker sets the position from which the execution of the sequence starts or from which sequence steps are added with the recording mode. Also, the real-time sequence validation considers the steps starting at the marker position. Choose *Sequence > Set Marker Position* to set the marker position. Alternatively, the marker can be dragged to the desired position.

Step number

Incremental number for each sequence step.

Execution status color

The execution status of steps is indicated by their background color.

- Light green: step has been executed.
- White: step has not been executed.
- Gray: step was interactively skipped during the execution of the sequence.

8	Extract: plane 2	▼
9	Extract: plane 3	▼
10	Probe: surf pt 1	▼
11	Probe: surf pt 2	▼
	▶ Press record or execute steps	
12	Probe: surf pt 3	▼
13	Probe: surf pt 4	▼

Icons

When creating a measurement sequence, a real-time sequence validation is performed in the background. If a step may cause a problem, an icon is displayed to inform the user.

- Warning icons are displayed to inform the user of a potential problem.
- Error icons are displayed when there is an error in the step.
- Information icons are displayed when a step is unusual or unnecessary.

Moving the mouse pointer over an icon displays a tooltip with the details.

11	Probe: trim pt 2	▼
12	Probe: trim pt 3	▼

At least one object must be measured before this step:
surf pt - trim pt 3

Possible actions
→ [Add missing steps](#)

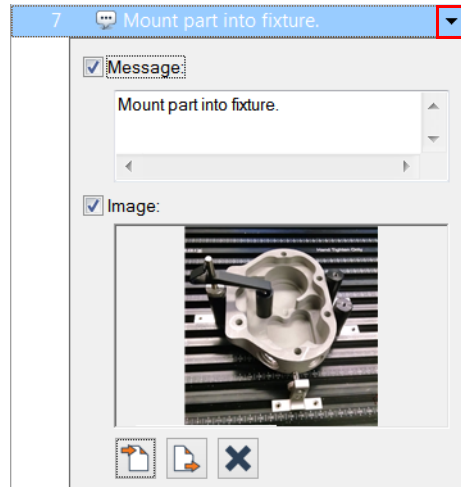
The status bar at the bottom of the Sequence Editor pane provides a summary of the problems detected in the measurement sequence.



Edit step shortcut

Opens a dialog box to define or edit a sequence step. The content of the dialog box depends on the step that is currently defined or edited.

For example, when editing guiding instructions, edit the step to add instructions (message and/or image) for the operator to illustrate how to mount a part into its fixture.



Using the Sequence Editor

Creating a measurement sequence from recorded actions

A measurement sequence is an ordered list of operations, called steps, created to measure objects.

From the autogenerated sequence

When the nominal inspection project is ready, it is possible to create a measurement sequence from the autogenerated one.

Steps

1. Choose *Tools > Sequence Editor*.
*Alternatively, click the **Sequence Editor** button on the Multipiece Inspection toolbar.*
2. In the Sequence Editor pane, choose *Sequence > Create from Autogenerated Sequence*.
3. Type the **Name** and **Description** of the sequence.
4. Click **Create**.

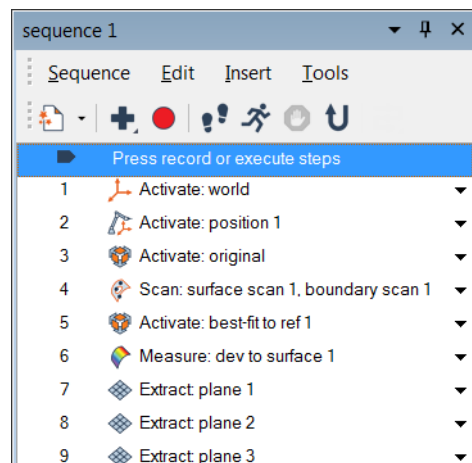


Name:	<input type="text" value="sequence 1"/>
Description:	<input type="text" value="Sequence for part"/>
<input type="button" value="Create"/> <input type="button" value="Cancel"/>	

Results

The sequence is created. It contains the objects from the Tree View along with initial conditions and device positions.

The sequence is ready to be customized. See [Customizing a measurement sequence](#) on page 652.



From recorded actions

Once a measurement device is properly configured and connected, it is possible to start the inspection of a part and create the sequence at the same time. Once recorded, subsequent pieces can be inspected just by playing the inspection sequence.

Steps

1. Choose *Tools > Sequence Editor*.

*Alternatively, click the **Sequence Editor** button on the Multipiece Inspection toolbar.*



2. In the Sequence Editor pane, choose *Sequence > New*.



3. Type the **Name** and **Description** of the sequence.

4. Click **Create**.

Name:	<input type="text" value="sequence 1"/>
Description:	<input type="text" value="Sequence for part"/>
<input type="button" value="Create"/> <input type="button" value="Cancel"/>	

5. Start the recording mode. Choose *Edit > Start Recording*.

The actions performed subsequently are added to the sequence (for example, alignments, activation of a coordinate system, and measurement operations).

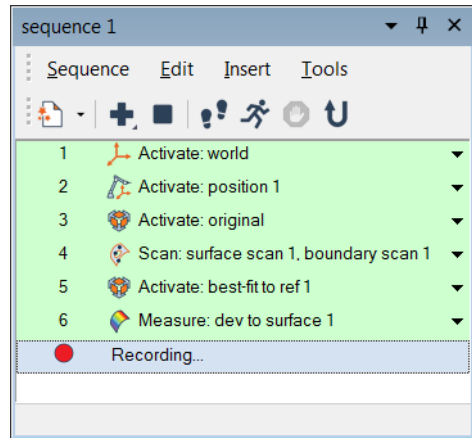


To stop recording actions, choose *Edit > Stop Recording*.



Results

A sequence is created. The initial conditions are automatically added to the sequence area.



Customizing a measurement sequence

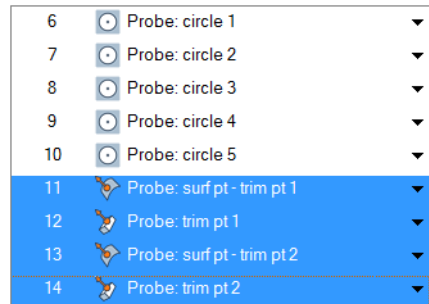
Once the measurement sequence is created, changes can be made. For example, it is possible to change the order of sequence steps to optimize an inspection, add guiding messages and images to facilitate the inspection of the part, or include macro scripts to increase efficiency.

Modifying the order of sequence steps

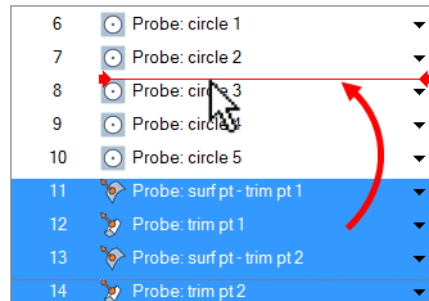
Steps in the sequence can be reordered, if required.

Steps

1. In the Sequence Editor pane, select the sequence steps that will be moved in the sequence area.

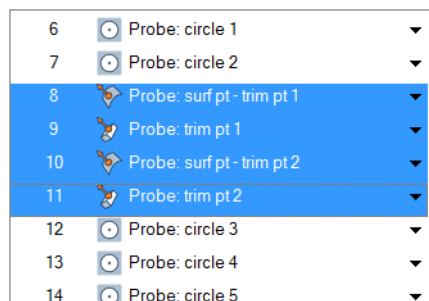


2. Drag the selected sequence steps to the desired position in the sequence.



Results

In the Sequence Editor, the steps are displayed at the new position.



Adding sequence steps

Steps can be added to the sequence. They can be broadly categorized as follows:

- General steps
- Context steps
- Measurement steps
- Macro script steps

When a step is added, a contextual dialog box is opened to specify a Tree View object or required information. If an object is preselected in the Tree View, then the inserted step is automatically set to the selected object.

Choosing steps from the Insert menu

Steps

1. In the Sequence Editor pane, choose *Insert > [select which step to add]*.

A step is added to the sequence and a contextual dialog box is displayed.



2. Specify the required information in the contextual dialog box.

3. Click outside the dialog box to confirm.

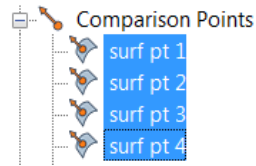
Notes

- Steps added using the **Insert** menu can be used as placeholders if relevant objects have not yet been created in the project. A warning icon appears. These steps must be defined before executing the sequence.
- Moving the mouse pointer over the object displays a tooltip with recommended actions to correct the issue.
- When a step is inserted, it appears at the end of the sequence, except if there is a selection in the sequence; in this case, the step is inserted above the selection.

Selecting objects in the Tree View

Steps

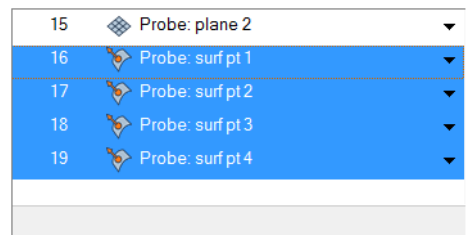
1. In the Tree View, select the objects associated with the steps that will be added to the sequence.



2. In the main menu, choose *Measure > Add to Sequence*.

Results

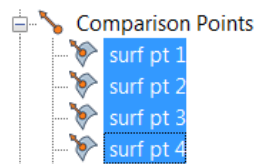
The steps are added to the sequence.



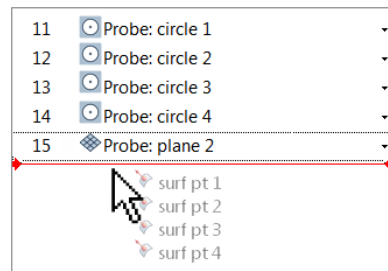
Dragging objects from the Tree View

Steps

1. In the Tree View, select the objects associated with the steps that will be added to the sequence.



2. Drag the objects to the desired position in the active sequence.



Notes

A convenient way to add steps for existing objects is to drag the objects from the Tree View.

Inserting macro script steps

Steps

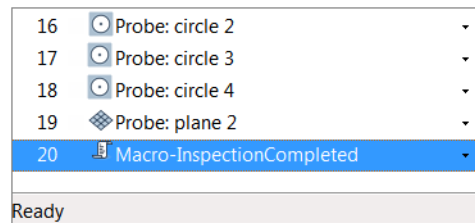
1. In the Sequence Editor pane, choose *Insert > Macro Script*. Two items are offered.

- **New:** The Macro Script Editor opens, allowing users to create a macro script.
- **Import:** Allows users to browse for an existing macro script.



Results

The step is added to the sequence. The macro script or, when imported, a copy of the macro script is saved in the current piece.



Editing sequence steps

Once added, sequence steps can be modified by accessing the step's contextual dialog box.

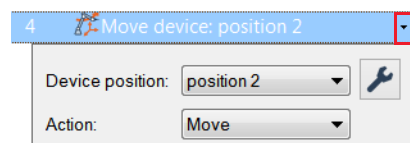
Steps

1. In the Sequence Editor pane, select the sequence step that will be modified and choose *Edit > Step*.

Alternatively, click the Edit Step shortcut arrow that corresponds to the step that will be modified.

The sequence step contextual dialog box opens.

2. Set the required parameters.



Defining conditional blocks

Depending on the inspection requirements, some measurement objects can be excluded from the inspection. It is possible to define these requirements, in a measurement sequence, as conditional statement steps. These steps allow users to construct a sequence that is adaptable to certain variations in an inspection, like the presence of an optional part.

Steps

1. Choose *Insert > Question*.



2. Type in a Yes or No question.
3. From the **Answer variable** list, select the variable where the answer will be stored.
If a new variable must be created, choose **(Create New)** and type the **Variable name**.
4. Set the initial value of the variable using the **Sequence validation answer** list. Refer to the Notes section for more information.
5. Select the steps in the sequence that are affected by the conditional statement.
6. Choose *Insert > Conditional Statement*.

7. Select the **variable** that will be used in the conditional statement.
8. Set the conditional value of the variable. If the variable is equal to the answer provided by the operator, the steps that are part of the conditional statement are executed. Otherwise, they are shown in gray and will not be executed.



Results

The steps are added to the sequence.

The steps that were selected when the conditional statement was inserted are grouped under the conditional statement.

9	?	Are comparison points required? variable: variable 1	▼
10	▶	If variable 1 equals "Yes"	▼
11	┆	Probe: surf pt 1	▼
12	┆	Probe: surf pt 2	▼

Notes

The **Sequence validation answer** allows users to validate the corresponding conditional statement steps for a given scenario. This value is not used when the measurement sequence is executed in the Sequence Editor or processed by the Play Inspection tool.

Evaluating pieces using Statistical Process Control

Statistical Process Control (SPC) is a tool that analyzes common measurements between multiple pieces for repeatability. Reviewing SPC results tells the user whether a part is being manufactured predictably or not.

PolyWorks | Inspector offers two SPC tools:

Object Control SPC:

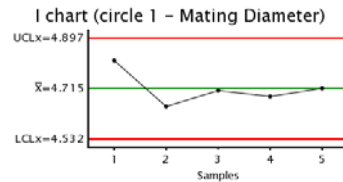
Analyzes measurement object dimensions, such as hole diameter, flatness, and length.

Surface Data SPC:

Analyzes the range of part shape using color maps.

Analyze part dimensions for repeatability using Object Control SPC

The **Object Control SPC** tool analyzes the values of any measurement common to multiple pieces in the inspection project. Results are displayed in graphic and tabular format in the Object Control SPC Results pane.



Objective: Display statistics of measured object values for all pieces.

Required: A completed inspection project containing more than one inspected piece.

Steps

1. In the Tree View of a piece, select the objects to consider in the SPC analysis.
2. Choose *Tools > Statistical Process Control > Object Controls*.



3. The Object Control SPC dialog box opens, with a list of the inspected pieces.

Pieces can be excluded from analysis by deselecting them in the list.

Pieces				
	No.	Name	Date	Time
<input checked="" type="checkbox"/>	1	Bracket 1	2015-01-...	11:00:20 AM
<input checked="" type="checkbox"/>	2	Bracket 2	2015-01-05	11:07:55 AM
<input checked="" type="checkbox"/>	3	Bracket 3	2015-01-05	11:14:31 AM
<input checked="" type="checkbox"/>	4	Bracket 4	2015-01-05	11:21:37 AM
<input checked="" type="checkbox"/>	5	Bracket 5	2015-01-05	11:28:52 AM

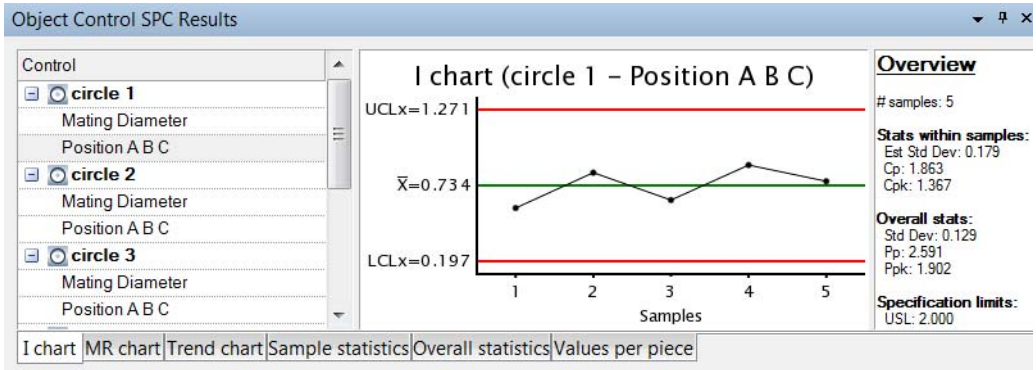
4. In the **Options** section, specify the **Sample size**.

See the following section for more information.

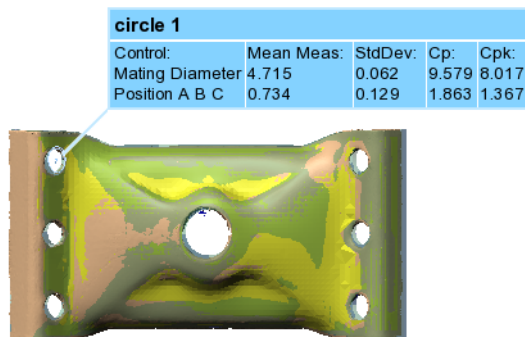
Options	
Contents:	Specific
Sample size:	1

Results

The value for each of the five pieces is displayed for the selected measurement object in the Object Control SPC Results pane.



Statistics of the piece values is shown in the object annotation.



Sample size

A sample consists of a collection of pieces. The sample size determines how many pieces represent a sample and is used to calculate mean values. The default sample size is **1**.

- Sample size equal to 1: I-MR charts are generated, consisting of the Individuals (I) chart and the Moving Range (MR) chart.
- Sample size greater than 1: Xbar-R charts are generated, consisting of the Mean (Xbar) chart and the Range (R) chart.

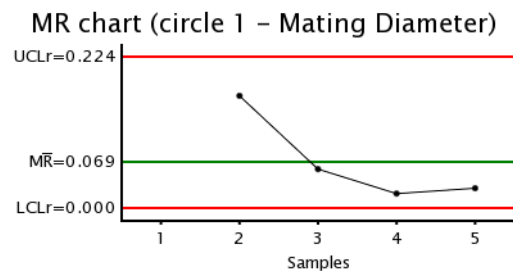
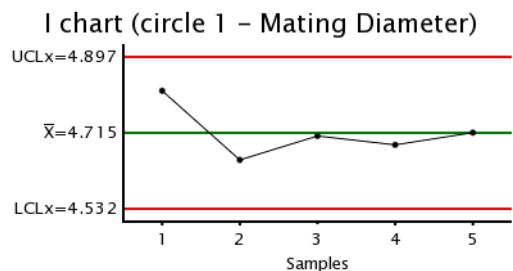
Understanding SPC results

The SPC tool offers different charts to analyze results. Depending on the sample size (1, or more than 1), different charts are generated.

I-MR charts

Generated when the sample size is 1, the I-MR charts display information about the performance of the process. They consist of:

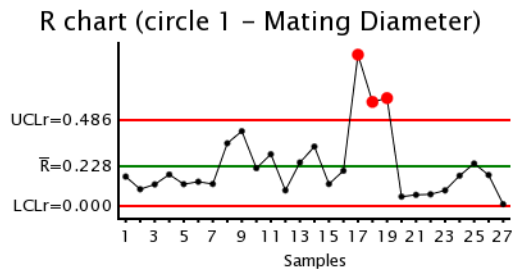
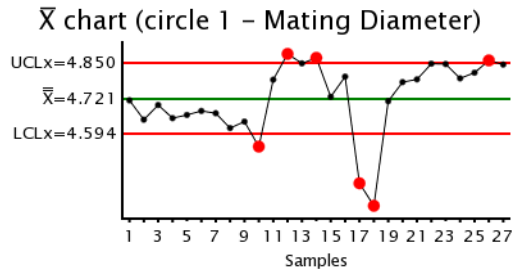
- Individuals (I) chart: It displays the measured values per piece for the selected control of the selected measurement object, and the mean of these values.
- Moving Range (MR) chart: For the selected control of the selected measurement object, it displays the variation from the previous piece, and the mean of these variations.



Xbar-R charts

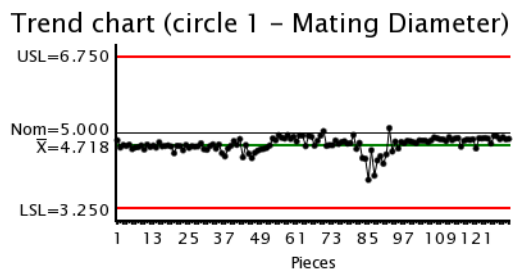
Generated when the sample size is greater than 1, the Xbar-R charts display information about the behavior of the process and outcome over time. They consist of:

- Mean (Xbar) chart: It displays the average measured value per sample for the selected control of the selected measurement object, and the mean of these values.
- Range (R) chart: For the selected control of the selected measurement object, it displays the range of data in each sample, and the mean of these ranges.



Trend charts

- Trend charts display the measured value per piece for the selected control of the selected measurement object and the mean of these values.



Sample statistics table

The Sample statistics table displays, per piece within each sample, the measured value for the selected control of the selected measurement object, the mean and the range of these values, and their average values.

Sample	Mean Meas	Range	#1	#2	#3	#4	#5
sample 1	4.715	0.166	4.816	4.649	4.707	4.686	4.715
sample 2	4.644	0.095	4.614	4.650	4.658	4.698	4.603
sample 3	4.698	0.122	4.716	4.660	4.693	4.648	4.771
sample 4	4.650	0.178	4.677	4.685	4.696	4.674	4.518
sample 5	4.662	0.124	4.689	4.685	4.585	4.708	4.642
sample 6	4.676	0.137	4.681	4.668	4.677	4.746	4.609
sample 7	4.668	0.125	4.596	4.681	4.721	4.622	4.718
sample 8	4.614	0.355	4.517	4.444	4.623	4.688	4.799
Average	4.721	0.228					

Overall statistics table

The Overall statistics table displays statistics values such as the Mean, Std Dev, Min, Max, Cp, Cpk, %Within Tol (percentage of pieces within tolerance), and more for the selected measurement object.

Control	Nominal	Tolerance	# Pieces	Mean Meas	Mean Dev	Std Dev	Min	Max	% Within Tol	Cp	Cpk
circle 1											
Mating Diameter	5.000	±1.750	132	4.718	-0.282	0.171	3.901	5.099	100.000	5.900	4.962
Position A B C		2.000	132	0.886	0.886	0.213	0.398	1.637	100.000	1.750	1.547

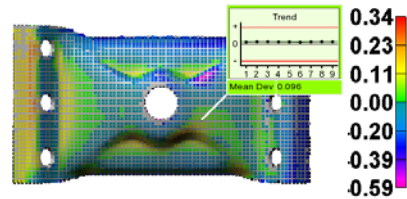
Values per piece table

The Values per piece table displays the measured values per piece for the selected controls of the selected measurement object.

Control	Nominal	Tolerance	1 - Bracket 1	2 - Bracket 2	3 - Bracket 3	4 - Bracket 4	5 - Bracket 5
circle 1							
Mating Diameter	5.000	±1.750	4.816	4.649	4.707	4.686	4.715
Position A B C		2.000	0.574	0.824	0.630	0.878	0.763

Analyze part shape for repeatability using Surface Data SPC

The **Surface Data SPC** tool analyzes the range in shape of multiple pieces in the inspection project by comparing them to the Reference object. Various results can be displayed using a color map. They include the range of variation, mean deviation, highest deviating areas, and more.



Objective: Display statistical analyses of Data object shape variation for all pieces.

Required: An inspection project containing a Reference object and more than one piece.

Steps

1. Choose *Tools > Statistical Process Control > Surface Data Objects*.

The Initialize Surface Data SPC dialog box and the Surface Data SPC pane open.

2. In the **Pieces** list, select **From Project**.

3. In the **Reference objects** list, select an object.

4. Specify the **Sampling step** defining the density of deviation measurements made to the Data objects.

The value should be at least as large as the point spacing of point cloud Data objects.

For all Data object types, the sampling step must be small enough to measure meaningful details.

5. Specify the **Max distance** to use to measure the deviations of the Data objects from the Reference object.

6. Click **Initialize**.

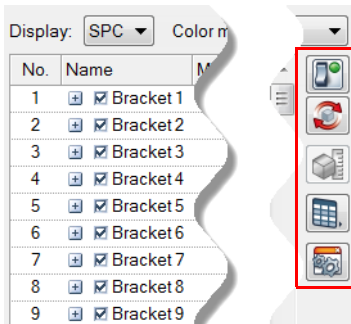


Method	
Pieces:	From Project ▼
Parameters	
Reference objects:	Corner Bracke... ▼
Sampling step:	1.000
Max distance:	4.000

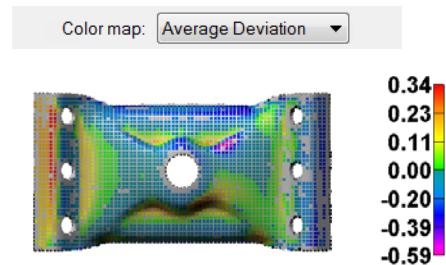
Results

The Surface Data SPC pane opens, listing the pieces included in the analysis. Deselect pieces to be removed from the analysis.

Tools used to initialize, reset, remeasure, report, and change options are located on the vertical toolbar of the Surface Data SPC pane.



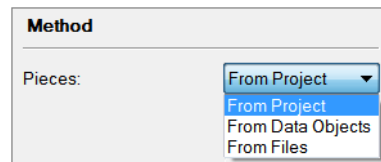
A color map in the 3D scene, generated on the Reference object, is shown based on the **Color map** type specified in the Surface Data SPC pane.



Notes

Other piece locations are available in the **Pieces** list of the Initialize Surface Data SPC dialog box.

When using **From Data Objects**, individual Data objects that are aligned to the Reference object can also be used for Surface Data SPC. They can be added to the Surface Data SPC tool, using pane buttons, and are immediately measured.



Working with multiple device positions



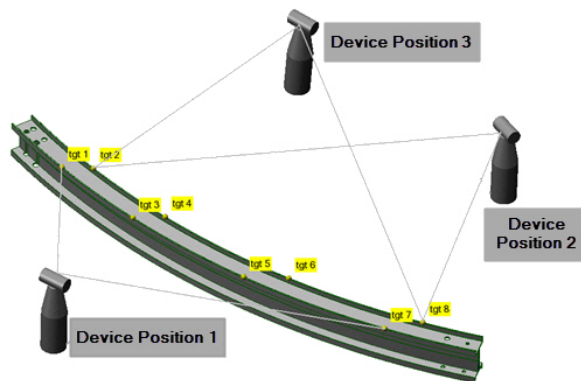
Working with multiple device positions

There are many challenges to acquiring data points on large objects. Some objects are larger than the measurement volume of a device. Sometimes, data must be acquired on a portion of a part that is hidden from the measuring device. These situations demand that the device be moved around the part to acquire the data. In PolyWorks | Inspector, device positions address this specific need.

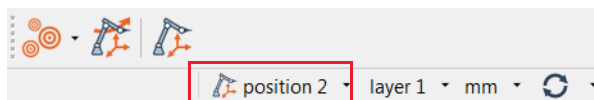
The different device positions must be aligned to each other. Two methods are detailed in this manual:

- Using targets
 - From points obtained by probing targets
- Using a custom alignment such as the best-fit of Data to Data objects

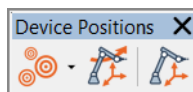
In the example illustrated below, a part is measured using a laser tracker in three different device positions.



The active device position is displayed on the status bar, in the lower right corner of the interface.



The Device Positions toolbar is located at the bottom of the PolyWorks | Inspector user interface.



It offers the following tools:



Define Measured Targets

Opens the Define Measured Device Position Targets dialog box from which to measure the targets.



Move Device

Allows the creation of a new device position.



Manage Device Positions and Targets

Opens the Device Positions and Targets pane which allows visualizing the different device positions and targets and editing them.

The basic acquisition workflow with multiple device positions using targets

For this typical workflow, the part is inspected by probing and/or scanning, and probed targets are used for the alignment. The targets can be defined from physical targets strategically placed on the part and/or around the part, or they can be features (points, circles, spheres) on the part. Note that a combination of physical targets and features on the part can be used.

Position 1



Define measured targets

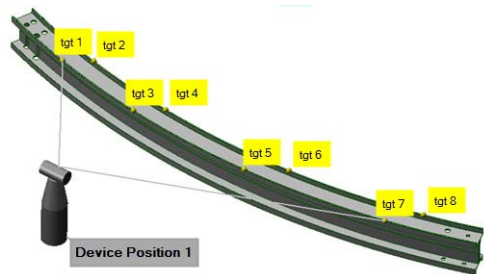
Define the accessible targets of *position 1*.

Scan the part & measure objects

Scan the accessible surfaces of the part and, if required, measure objects using scanned or probed data.



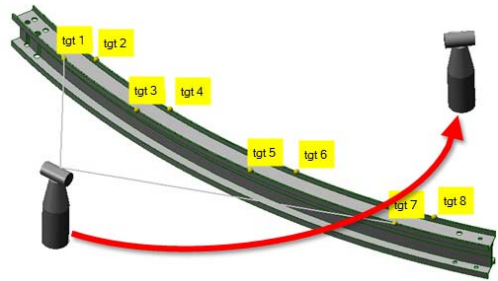
Note that the Data object needs to be aligned to the Reference object to measure certain types of objects.





Move device

Define a new device position and physically move the device or the part.



Position 2

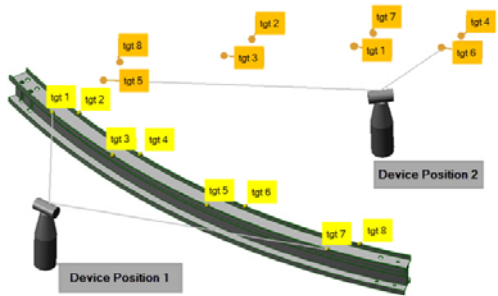
Define measured targets

The acquisition of targets is automatically launched.



Define all the accessible targets of *position 2*. There must be a minimum of three common targets, with the previous position, for alignment.

There can be more targets than in the previous position. Typically, these will be used with targets in future positions.



Scan the part & measure objects



Scan the accessible surfaces of the part and, if required, measure objects using scanned or probed data.

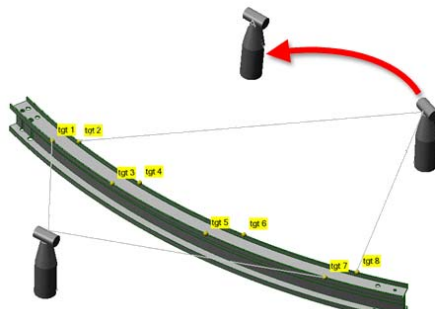


Note that the Data objects need to be aligned to the Reference object to measure certain types of objects.



Move device

Define a new device position and physically move the device or the part.



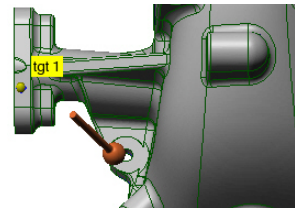
Position n

Repeat the process

Repeat this process for as many positions needed to acquire the required data.

Moving the device or the part by probing targets

A robust and repeatable method for defining targets is using target locations. The targets can be on the part itself or placed around it. The probe is used to define each target every time the device or the part is moved, aligning the different setups together.



Objective: Move the device or the part and align it to previous data by probing device position targets.

Required: A probing device is available and ready to use. The required plug-ins are loaded.

Steps

1. In *position 1*, define the targets by choosing *Tools > Device Positions > Define Measured Device Position Targets*.



The Define Measured Device Position Targets dialog box opens.

2. In the **Method** list, select **Probe**.
3. Set the **Type** to **Point**.
4. In the **Submethod** list, select **Uncompensated**.
The target will be acquired at the probe tip center.

5. Click **Probe**.
6. Probe all accessible stable target locations.

Three targets are a minimum to perform the target alignment, but more are recommended.

The targets are listed in the Device Positions and Targets pane.

Target	RMS Dev	Max Dev	position 1 (active)
tgt 1			<input checked="" type="checkbox"/> 0.000
tgt 2			<input checked="" type="checkbox"/> 0.000
tgt 3			<input checked="" type="checkbox"/> 0.000
tgt 4			<input checked="" type="checkbox"/> 0.000
tgt 5			<input checked="" type="checkbox"/> 0.000
tgt 6			<input checked="" type="checkbox"/> 0.000

- Scan the accessible surfaces of the part and, if required, measure objects.

Note that the Data object needs to be aligned to the Reference object to measure certain types of objects.

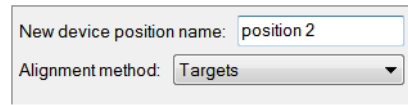


- Physically move the device or part, and define a new device position – *position 2* – by choosing *Tools > Device Positions > Move Device*.



The Move Device dialog box opens.

- In the **Alignment method** list, select **Targets**.
- Click **OK**.



The Define Measured Device Position Targets dialog box opens.

- Probe all accessible and stable target locations to align *position 2*.

Target	RMS Dev	Max Dev	position 1	position 2 (ac
tgt 1	1.069	1.069	<input checked="" type="checkbox"/> 0.000	<input checked="" type="checkbox"/> 1.069
tgt 2	0.467	0.467	<input checked="" type="checkbox"/> 0.000	<input checked="" type="checkbox"/> 0.467
tgt 3	0.833	0.833	<input checked="" type="checkbox"/> 0.000	<input checked="" type="checkbox"/> 0.833
tgt 4			<input checked="" type="checkbox"/> 0.000	
tgt 5			<input checked="" type="checkbox"/> 0.000	
tgt 6	1.007	1.007	<input checked="" type="checkbox"/> 0.000	<input checked="" type="checkbox"/> 1.007

At least three targets must be the same as in position 1. Although a minimum of three targets is required to perform the target alignment, it is recommended to use more targets.

The targets can be probed in any order.

The targets are listed in the Device Positions and Targets pane, with alignment errors.

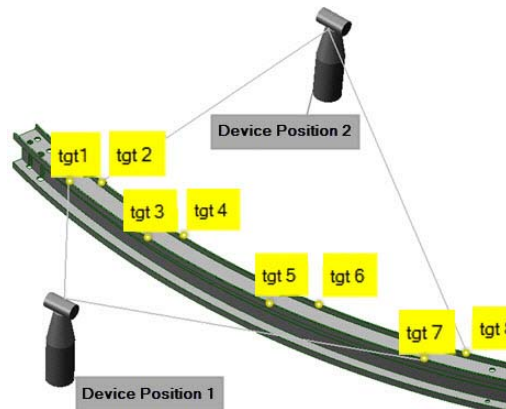
- Continue scanning and probing the part in the new position.



Results

- The new device position is aligned to the previous device position.

New scanning and probing Data objects are created in the Tree View to keep the device positions independent of each other.



- The probed targets and the alignment accuracy are displayed in the Device Positions and Targets pane.

Target	RMS Dev	Max Dev	position 1	position 2 (active)
tgt 1	1.069	1.069	<input checked="" type="checkbox"/> 0.000	<input checked="" type="checkbox"/> 1.069
tgt 2	0.467	0.467	<input checked="" type="checkbox"/> 0.000	<input checked="" type="checkbox"/> 0.467
tgt 3	0.833	0.833	<input checked="" type="checkbox"/> 0.000	<input checked="" type="checkbox"/> 0.833
tgt 4			<input checked="" type="checkbox"/> 0.000	
tgt 5			<input checked="" type="checkbox"/> 0.000	
tgt 6	1.007	1.007	<input checked="" type="checkbox"/> 0.000	<input checked="" type="checkbox"/> 1.007
Max Value	1.069	1.069		1.069

Notes

All accessible targets should be measured from each device position. This ensures that plenty of targets are available for future device position alignments. As soon as there are three common targets between the device positions, the current device position is aligned. Adding additional common targets strengthens the alignment.

To avoid ambiguous alignment solutions, physical targets should be placed in a nonsymmetrical array. If necessary, target matches between device positions can be edited manually. The **Edit Target Match** tool, found on the **Target alignment results** tab of the Device Positions and Targets pane, allows defining specific target match pairs.



When connected to a laser tracker probing device with the capability to orient to gravity, it is possible to orient to gravity while creating the device position.

New device position name: position 2

Alignment method: Targets

Orient to gravity

Temperature compensation options

OK Cancel

Bundle adjustment

Bundle adjustment optimizes the auto-match alignment of device positions by finding the most probable effective location for each target through averaging, and aligns all instances of the target to that location. The effective location is updated every time a new device position is added. This option gives best results when the entire part volume is measured such that the last device position is adjacent to the first device position.

Without bundle adjustment, any target measured in more than one device position is always aligned to the first measured instance of that target.

Objective: Activate bundle adjustment for target alignment.

Required: A project with multiple device positions aligned using targets.

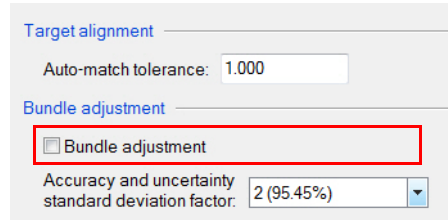
Steps

1. Choose *Tools > Options* to access the PolyWorks | Inspector options.



The PolyWorks | Inspector Options dialog box opens.

2. Open the **Device Positions** options page.
3. Select **Bundle adjustment**.
4. Click **Apply**.



5. Close the PolyWorks | Inspector Options dialog box.

Results

The alignment is updated, and the error for all targets is recalculated based on the new effective target location.

Target /	RMS Dev	Max Dev	position 1	Position 2 (active)
⊕ tgt 1	0.006	0.006	☑ 0.006	☑ 0.006
⊕ tgt 2	0.009	0.009	☑ 0.009	☑ 0.009
⊕ tgt 3	0.000	0.000	☑ 0.000	
⊕ tgt 4	0.010	0.010	☑ 0.010	☑ 0.010
⊕ tgt 5	0.000	0.000	☑ 0.000	
⊕ tgt 6	0.005	0.005	☑ 0.005	☑ 0.005
Max Value	0.010	0.010	0.010	0.010

Device positions | Target alignment | Target positions

Notes

Accuracy models are used to determine the most probable location for each target. It is therefore important to set the proper accuracy model for each device position according to the device used. The default accuracy model can be set from the **Device Positions** page in the PolyWorks | Inspector options. Alternatively, a specific accuracy model can be set for each device position in the **Device positions** tab of the Device Positions and Targets pane. Two bundle adjustment accuracy models are available:

Uniform accuracy model

Digitizing arms have small measurement volumes and, as such, have fairly consistent accuracy throughout that volume. For digitizing arms, the **Uniform accuracy model** is used.

Range and angle accuracy model

Laser trackers have large measurement volumes, and the point positional certainty varies throughout that volume based on the distance and angle from the tracker. Therefore, for laser trackers, the **Range and angle accuracy model** is used.

Default accuracy models

Uniform accuracy model:

Accuracy: μm

Range and angle accuracy model (laser tracker):

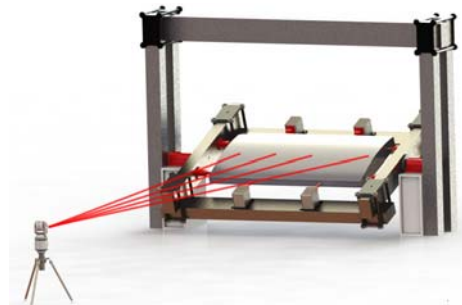
Range accuracy: μm + $\mu\text{m}/\text{m}$

Angle accuracy: μm + $\mu\text{m}/\text{m}$

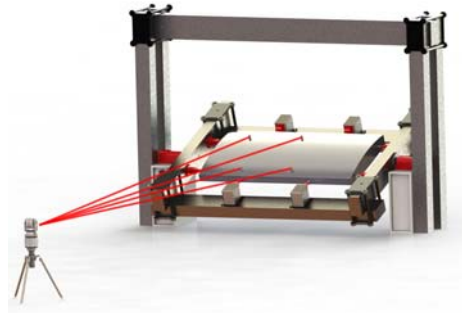
Device position uncertainty

When physically predetermining a set of target locations that can be measured from several device positions, users can, by inadvertence or inexperience, set up their alignment targets in ways that systematically produce unstable alignment solutions. It is possible to detect these unreliable target setups by evaluating device position uncertainties. Here are some of the most common problematic target setups:

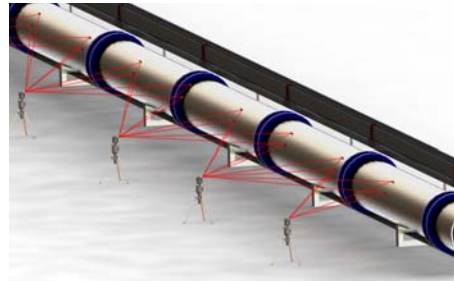
- The alignment targets of a device position are nearly collinear.



- The horizontal angular span of the alignment targets of a device position is small.



- The measurement device is moved along one direction, and a new device position is aligned by only using targets measured by the closest previous device position, producing daisy-chained device positions.



Objective: Measure the uncertainty of a device position.

Required: A project with two or more device positions. Bundle adjustment must be enabled.

Steps

1. Choose *Tools > Device Positions > Manage Positions and Targets*. The Device Positions and Targets pane opens.
2. Select the **Device positions** tab at the bottom of the pane.



Name	Alignment	Bundle Constraint	Accuracy Model
position 1	Target Bundle	Lock	▾ Range and Angle
position 2	Target Bundle	None	▾ Range and Angle

3. Click **Evaluate Position Uncertainty**.

*The **Device Positions** options page offers parameters related to uncertainty computation.*



Results

The uncertainty of the unlocked positions is calculated and shown in the Device Positions and Targets pane along with a pass/fail test.

Name	Alignment	Bundle Constraint	Accuracy Model	Uncertainty	Test
Tracker position 1	Target Bundle	Lock	Range and Angle	-----	-----
Tracker position 2	Target Bundle	None	Range and Angle	0.017	Pass

Notes

- When targets are added or removed, the uncertainty values are considered out-of-date and are removed. The uncertainty tool must be relaunched to update the uncertainty values.
- It is possible to modify the tolerance for the device position uncertainty on the **Device Position** page of the PolyWorks|Inspector options.

The basic acquisition workflow with multiple device positions using surface information

For this workflow, the part is digitized and the scanned surfaces are used for alignment, which involves relying on the overlap between the objects to align.

Position 1



Scan the part

In *position 1*, scan the area of the part that is accessible.



Move device

Define a new device position and physically move the device or the part.

New device position name:	<input type="text" value="position 2"/>
Alignment method:	<input type="text" value="Custom"/>

Position 2



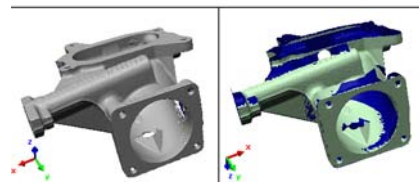
Scan the part

In *position 2*, scan the area of the part that is accessible to capture surface data that overlaps with previous scans.



Align scan with previous scans

Based on the surface information, use a Best-Fit Data to Data objects alignment to align the scanned data; overlapping data is required.



Move device

Define a new device position and physically move the device or the part.

New device position name:	<input type="text" value="position 3"/>
Alignment method:	<input type="text" value="Custom"/>

Position n

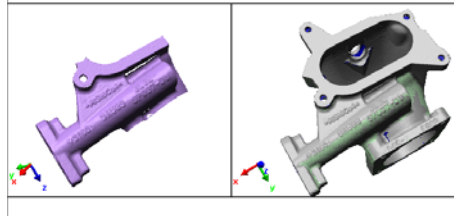
Repeat the process

Repeat this process for as many positions as needed to acquire the required data.

Last position

Scan the part and align with previous scans

In the *last position*, scan the part capturing surface data that overlaps with previous scans. Based on the surface information, use a Best-Fit Data to Data objects alignment to align the scanned data where overlapping data is available.

**Globally optimize alignments**

Use this alignment when three or more Data objects, from different device positions, were scanned and are closely aligned.



Moving the device or the part using a custom alignment

When scanning a part, it is sometimes impossible to capture all of the surfaces without moving the scanning device, or changing the part position relative to the scanning device. This generates one surface Data object for each different setup used. If the device positions are not already aligned, it is possible to align these surface Data objects to one another. The alignment uses the areas that overlap with neighboring polygonal Data objects. First a pre-alignment is done by the user by picking matching points, and then a best-fit alignment is performed using the surface shape in the overlying regions.



Objective: Scan the part into different device positions.

Required: A scanning device is available and ready to use. The required plug-ins are loaded.

Steps

1. In *position 1*, scan the areas of the part that are accessible.

Position 1 can be aligned to a Reference object.



2. Move the device or part, and define a new device position – *position 2* – by choosing *Tools > Device Positions > Move Device*.



The Move Device dialog box opens.

3. In the **Alignment method** list, select **Custom**.

New device position name:	<input type="text" value="position 2"/>
Alignment method:	<input type="text" value="Custom"/>
Temperature compensation options	<input type="text" value=""/>
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

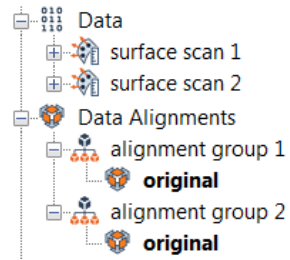
4. Click **OK**.

5. Continue scanning the part in the new position.



Results

New scanned Data objects are created in the Tree View for each device positions.



Next step

Once all the part has been digitized, align the polygonal Data objects using a Best-fit Data to Data objects.

Unifying polygonal Data objects

When data is acquired using multiple device positions, one polygonal Data object per device position is typically created. Before inspecting the part, it is recommended to unify the polygonal Data objects into only one Data object.

Objective: Unify polygonal Data objects into one object.

Required: Polygonal Data objects acquired using different device positions that are already aligned to each other.

Steps

1. Select the polygonal Data objects in the Tree View.



2. Choose *Tools > Data Objects > Create Polygonal Models*.

The Create Polygonal Data Objects dialog box opens.

3. Specify a **Name**.
4. In the **Method** list, select **From Data Objects**.
5. Under **Source data objects**, select the Data objects that will be unified.

When the objects are selected in the Tree View, their check box is automatically selected.

6. In the **Meshing method** list, select **Quality Meshing**.



Creation method

Method: From Data Objects

Parameters

Source data objects:

manifold_setup1
 manifold_setup2

Meshing method: Quality Meshing

Data type: Line Scan

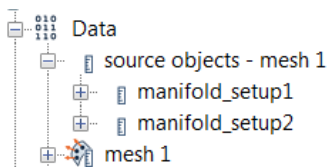
Meshing parameters:

From source objects
 Custom

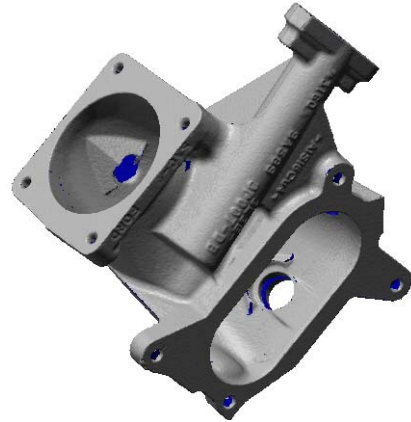
7. Specify how the **Meshing parameters** are to be obtained (**From source objects** is recommended). If **Custom** is selected, the following parameters must be specified:
 - **Sampling step:** Determines the average distance between vertices. The smaller the step, the finer the details are in the resulting polygonal Data object. It is recommended to use the same step as the one used when scanning.
 - **Max edge length:** Sets the maximum size for any triangle edge. A good approximation is ten times the sampling step, but always keep it smaller than the finest details on the polygonal Data object. Otherwise, details will lose definition.
8. Click **Create**.

Results

A new polygonal Data object is created and added to the Data branch of the Tree View. The source objects are moved into a group. The new polygonal Data object is dependent on the source objects it was created from. If any of the source objects are modified, this new Data object is updated.



The new polygonal Data object is displayed in the 3D scene.



Using the Offline Simulation mode



Introducing the Offline Simulation mode

The Offline Simulation mode allows users to perform simulated data acquisition without using a physical device. Simulated data is provided for all scanning and probing operations. As a result, an entire inspection project, including data color maps, device positions, measurement sequences, and formatted reports, can be prepared. Once the inspection project is complete, it can be used with an actual device to play the inspection, acquire the data, and measure the part.

Activating/deactivating the Offline Simulation mode

To create inspection projects offline using simulated data, the Offline Simulation mode must first be activated.

1. On the Devices toolbar, click the **Activate Offline Simulation** button.



The **Simulated Data** watermark is displayed at the bottom of the 3D scene as a reminder that simulated data is being used in the project.

The inspection can be performed using:

- simulated scanned data (see [The workflow when performing an inspection using simulated scanned data](#) on page 684)
- simulated probed data (see [The workflow when performing an inspection using simulated probed data](#) on page 689)

Note that the watermark is displayed as long as the Offline Simulation mode is activated and the project contains simulated data.

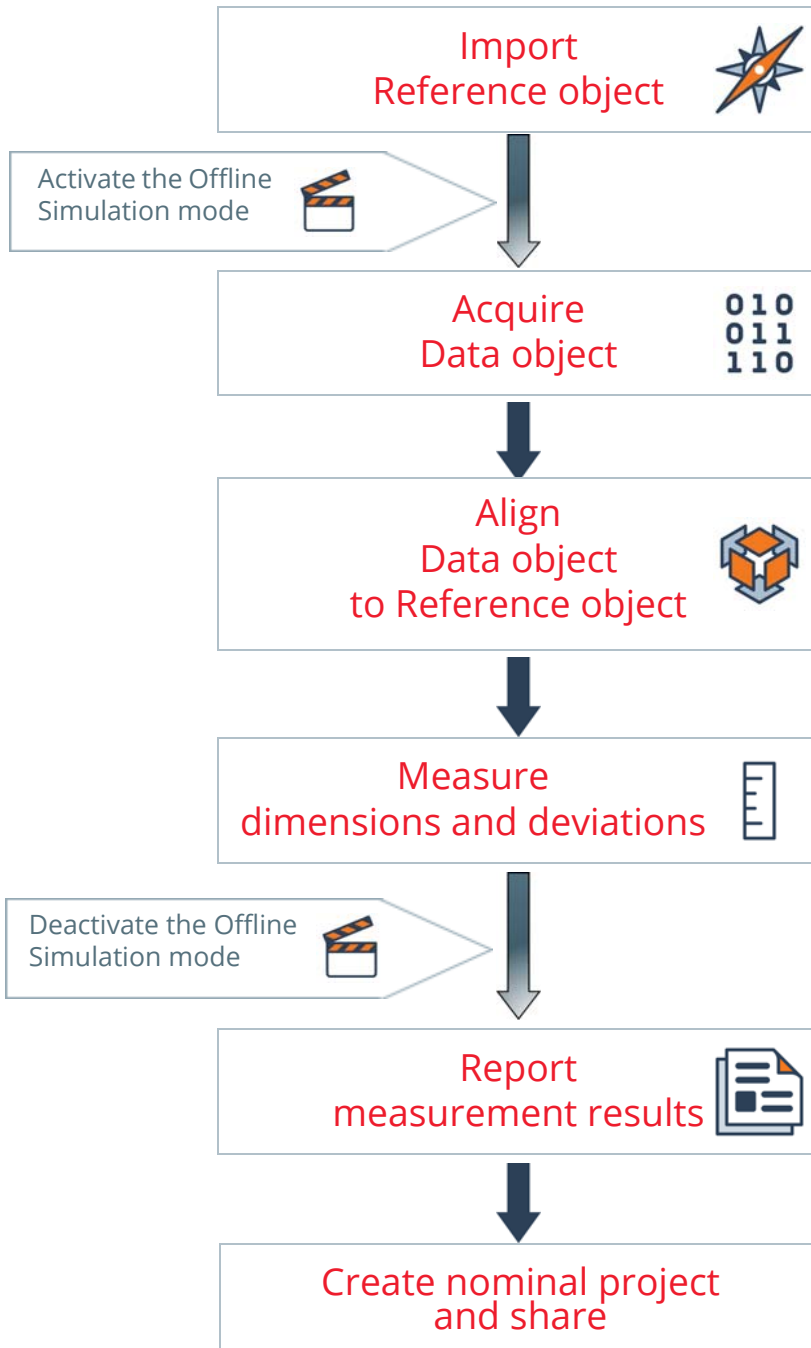
Simulated Data

2. To deactivate the Offline Simulation mode, press the same button on the Devices toolbar, now named **Deactivate Offline Simulation**.



The workflow when performing an inspection using simulated scanned data

The workflow for an inspection using a scanning device and the Offline Simulation mode is illustrated below:

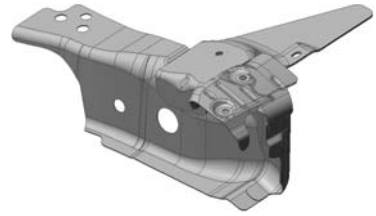


Understanding the workflow



Import Reference object

Import the nominal information of the part (typically a CAD model) into the inspection project.



Activate the Offline Simulation mode



Activate the Offline Simulation mode

Click the button on the Devices toolbar to activate the Offline Simulation mode.



Specify the scanning plug-in

In the scanning plug-in list, select the required plug-in for your scanning device.



Acquire Data object

Acquire simulated scanned data, based on the Reference object.



Align Data object to Reference object



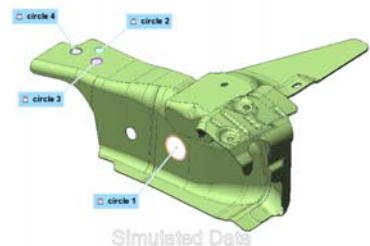
Alignment

Align the Data object to the Reference object using a best-fit alignment.



Refine alignment

Refine the alignment of the Data object to the Reference object using one of the many alignment techniques once the required objects are available.



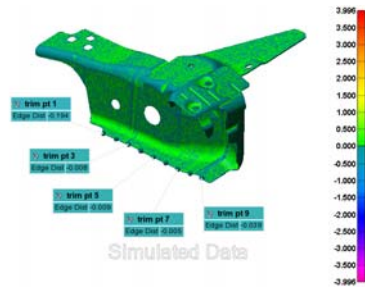


Measure dimensions and deviations

Create the objects required to perform the inspection of the part.

Set the required dimensional and GD&T controls, if applicable.

Measure the different objects using simulated scanned data.



Deactivate the Offline Simulation mode

Click the button on the Devices toolbar to deactivate the Offline Simulation mode.

Report measurement results



Report the inspection measurement results using:

- Control views in the Control Reviewer;
- Tables and snapshots in formatted reports.

Feature Table									
Units	Comments								
mm									
Name	Control	Nom	Meas	Ref	Dev	Stat	Def	Def	Def
Circle 1	Circle	28.000	28.018	#1.000	0.018	Pass			
	A	488.762	488.762	#1.000	0.000	Pass			
	V	864.287	864.284	#1.000	0.003	Pass			
	Z	88.070	88.070	#1.000	0.000	Pass			
Circle 2	Circle	10.000	9.992	#1.000	-0.008	Pass			
	A	488.800	488.800	#1.000	-0.000	Pass			
	V	868.816	868.816	#1.000	0.000	Pass			
	Z	180.824	180.824	#1.000	0.000	Pass			
Circle 3	Circle	10.000	10.004	#1.000	0.004	Pass			
	A	470.001	470.000	#1.000	-0.001	Pass			
	V	438.102	438.102	#1.000	0.000	Pass			
	Z	180.819	180.819	#1.000	0.000	Pass			
Circle 4	Circle	10.000	10.000	#1.000	0.000	Pass			
	A	470.000	470.000	#1.000	0.000	Pass			
	V	437.837	437.837	#1.000	0.000	Pass			
	Z	210.329	210.317	#1.000	-0.012	Pass			

Create nominal project and share



Save the inspection project

Create nominal project

The creation of the nominal project removes all measurements and saves the project under a new name.

The project is now ready to be used to play the inspection.

Particularities when acquiring simulated scanned data

Before using a scanning device to acquire simulated scanned data, make sure that the following requirements have been met:

- The project contains a Reference object.
- The Offline Simulation mode is enabled.
- The scanning plug-in has been selected and the connection to the device has been established.
- An alignment to the Reference object has been previously simulated.

Scanning options

If required, set options for the acquisition of simulated scanned data. The goal is to have data dense enough so that the extraction of measurement objects is successful. Note that these options are not applicable when scanned data is acquired with an actual scanning device.

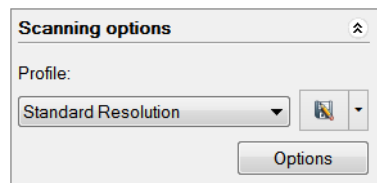
1. Click the **Scan** button.

The plug-in dialog box opens.



For information on the parameters offered for scanning, see [Acquiring surface and boundary Data objects](#) on page 91.

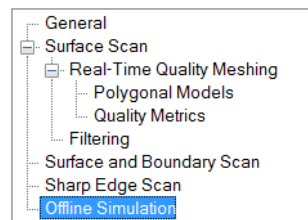
2. Under **Scanning options**, select the appropriate **Profile** based on the size of the smallest details on the part.
3. If required, click **Options** to set the options specific to the device.



The Line Scan Options dialog box opens.

4. On the **Offline Simulation** page, set the required parameters.

This page contains the default parameters used to configure the acquired simulated scanned data.

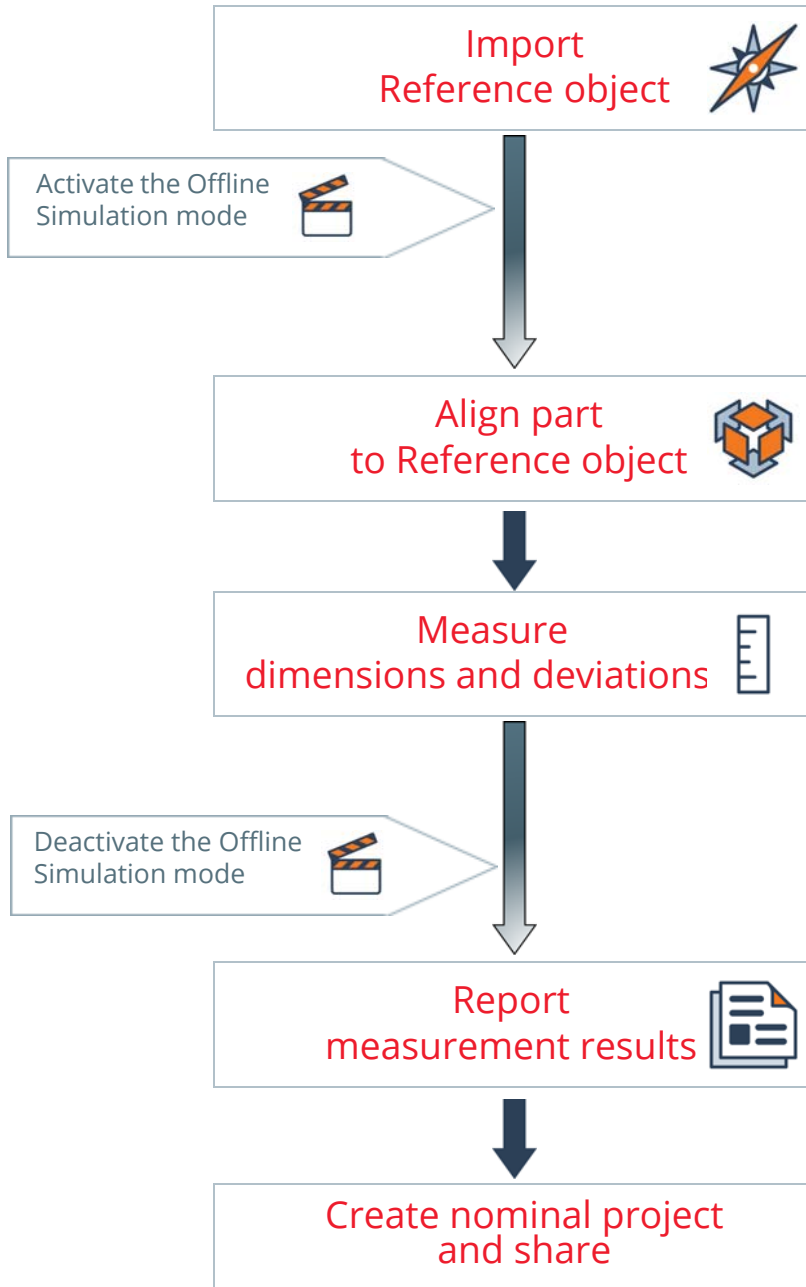


- **Line-to-line distance:** Controls the distance between consecutive scan lines.
- **Point-to-point distance:** Controls the distance between two neighboring points on the same scan line.
- **Noise:** Introduces noise into the simulated scanned data based on a tolerance.
- **Scan nominal primitives for measured primitive extraction:** Additional data is simulated for features in the Tree View for which the measurement method is set to **Extract** or **Undefined**. This ensures that feature extraction is done successfully.

Scan lines
Line-to-line distance
Point-to-point distance
Noise
Additional scan passes
Scan nominal primitives for measured primitive extraction

The workflow when performing an inspection using simulated probed data

The workflow for an inspection using a probing device and the Offline Simulation mode is illustrated below:



Understanding the workflow



Import Reference object

Import the nominal information of the part (CAD model) into the inspection project.



Activate the Offline Simulation mode



Activate the Offline Simulation mode

Click the button on the Devices toolbar to activate the Offline Simulation mode.



Specify the probing plug-in

In the probing plug-in list, select the required plug-in for your probing device.



Connect to the probing device

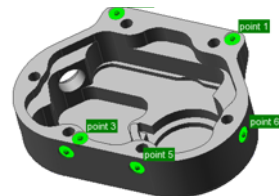
Connect to the probing device to simulate the connection.

Align part to Reference object

Alignment



Simulate the alignment of the part to the Reference object using a Surface Points alignment.

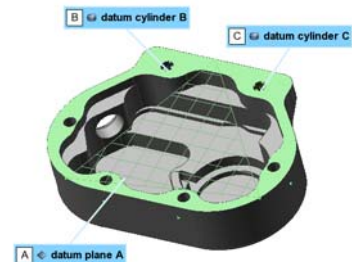


Simulated Data

Refine the alignment



Refine the alignment of the part to the CAD model using one of the many alignment techniques once the required objects are available.



Simulated Data

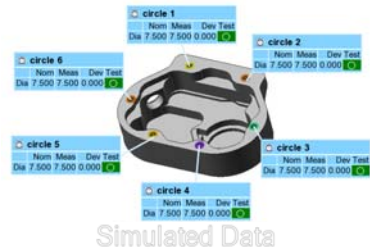
Measure dimensions and deviations

Create the necessary objects in order to perform the inspection of the part.



Set the required dimensional and GD&T controls, if applicable.

Then, specify the Probe method, set the related probing parameters, and measure the different objects using simulated probed data.



Deactivate the Offline Simulation mode



Disconnect from the probing plug-in

Click the button on the Devices toolbar to disconnect from the probing plug-in.



Deactivate the Offline Simulation mode

Click the button on the Devices toolbar to deactivate the Offline Simulation mode.

Report measurement results



Report the measurement results based on the probed data through:

- Control views in the Control Reviewer;
- Tables and snapshots in formatted reports.

Feature Table							
Units	Millimeters						
Coordinate System	world						
Data Alignments	off - A B/M C/M						
Name	Control	Nom	Meas	Tol	Dev	Test	Out Tol
circle 1	Diameter	7.500	7.500	±1.000	0.000	Pass	
circle 2	Diameter	7.500	7.500	±1.000	0.000	Pass	
circle 3	Diameter	7.500	7.500	±1.000	0.000	Pass	
circle 4	Diameter	7.500	7.500	±1.000	0.000	Pass	
circle 5	Diameter	7.500	7.500	±1.000	0.000	Pass	
circle 6	Diameter	7.500	7.500	±1.000	0.000	Pass	

Create nominal project and share



Save the inspection project





Create nominal project

The creation of the nominal project removes all measurements and saves the project under a new name.

The project is now ready to be used to play the inspection.

Simulating multiple device positions

The Offline Simulation mode can be used to simulate probing targets in different device positions. Four targets are automatically generated and probed, and they are listed in the Device Positions and Targets pane.

Target	RMS Error	Max Error	position 1 (active)
 tgt 1			<input checked="" type="checkbox"/> 0.000
 tgt 2			<input checked="" type="checkbox"/> 0.000
 tgt 3			<input checked="" type="checkbox"/> 0.000
 tgt 4			<input checked="" type="checkbox"/> 0.000

However, when measuring the actual part, any number of targets can be created for each device position as long as there is a minimum of three targets per position that are measurable in the next device position.

For more information on multiple device positions, see [Working with multiple device positions](#) on page 666.

Creating a nominal project

Once the inspection project containing simulated data is completed, the simulated data must be removed before sharing it with operators. This can be done by creating a nominal project. The project can then be used to start measuring the different pieces of a part.

Objective: Create a nominal project ready for use with a measurement device.

Required: A completed inspection project made with simulated data.

Steps

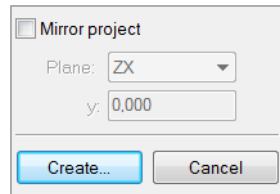
1. Make sure that the Offline Simulation mode is deactivated.

2. Choose *File > Create Nominal Project*.

The Create Nominal Project dialog box opens.

3. Click **Create**.

A browser opens.



4. Specify a file name and location.

5. Click **Save**.

The project is ready to use. All measurements are deleted.

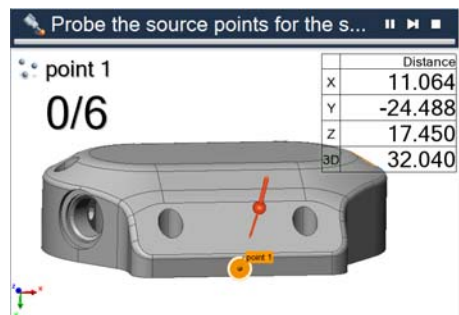
6. Connect to the measurement device.

7. Choose *Tools > Piece > Play Inspection*.



Results

The Play inspection is launched, guiding the user through the steps necessary to measure the measurement objects.



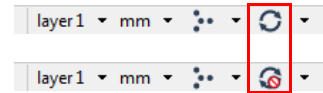
Appendix



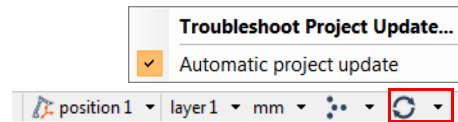
Automatic project update

By default, PolyWorks | Inspector projects are always automatically up to date. This means that measurement objects, alignments, data color maps, report items and formatted reports are automatically recalculated as necessary.

The icon of the Automatic project update status, located at the right end section of the status bar, indicates whether the automatic project update is activated or not.

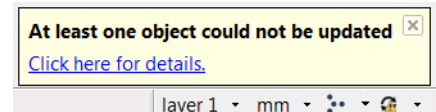


If needed, the Automatic project update can be activated/deactivated by clicking the arrow to the right of the icon and selecting the Automatic project update command. When activated, the command is preceded by a check mark.

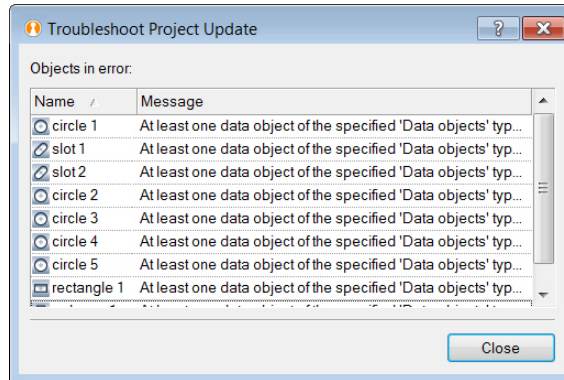


Troubleshooting the automatic project update





The update status of objects in the Tree View is constantly monitored by the Automatic project update function. In some cases, object updates cannot be performed, such as when the measurement parameters of an object do not allow its remeasurement or when an object is linked to an object that no longer exists. In these cases, a yellow warning triangle is superimposed on the Automatic project update icon on the status bar and a notification is shown.



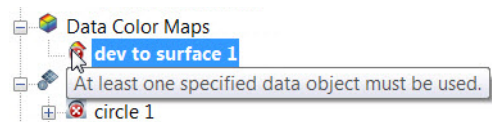
Choose *Tools > Troubleshoot Project Update* to display a dialog box that lists the objects in error and a message for each.



The following update status icons can be displayed on the object icon in the Tree View:

-  A red x mark indicates that the object is in error.
-  A yellow warning triangle indicates that the object could not be updated.
-  An anchor symbol indicates that the object is nonupdatable or rendered as such.
-  A clock icon indicates that the object is out-of-date. An object can have an out-of-date status only when the Automatic project update is deactivated.

Moving the mouse pointer over the object displays a tooltip with the details.



Operations that trigger an automatic project update

Common operations that trigger the automatic project update include:

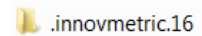
- Modifying a Data or Reference object by replacing the object or by deleting elements.
- Changing the active alignment.
- Opening a formatted report.

Setting and saving the user configuration

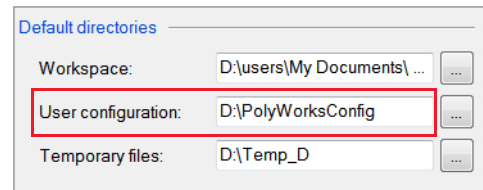
The PolyWorks Metrology Suite has a default **standard configuration** for the Workspace Manager and each module. The standard configuration holds information about window sizes and positions, display modes, selection modes, dialog box parameters, templates (annotations, controls, and tables), and various module options like units and Options panel settings.

A **user configuration** also exists, which augments the standard configuration. As a user works in the software, any changes to these settings and parameters can be saved in the user configuration.

On the computer, the user configuration files are found in a folder named `.innovmetric.[version]`.



Each user can have their own personal user configuration. The location of the user configuration directory, by default within the \User directory, is shown in the **Workspace Manager Options** dialog box.

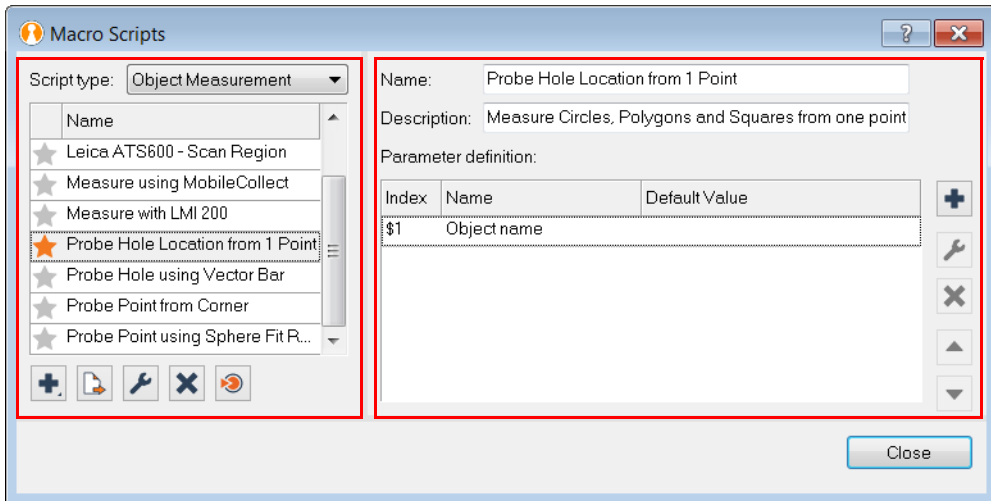


Changes to the parameters and settings in the user configuration can be saved by choosing *Tools > Save User Configuration* in any module. The saved settings become the default the next time the module is launched. In the Workspace Manager, the current user configuration for all modules can be exported by choosing *Tools > User Configuration > Export*. Likewise, an external user configuration can be loaded by choosing *Tools > User Configuration > Load*.

Scripted measurement methods

Scripted measurement methods can be used to measure objects in a custom way. These methods can be accessed through the property sheet of an object (see the note later in this section for the list of objects that support this method).

They can also be managed using the Macro Scripts dialog box, which is opened when choosing *Tools > Macro Scripts > Manage Macro Scripts*.



Once the **Object Measurement** macro script type has been selected, the left side of the Macro Scripts dialog box can be used to:

- Select the macro script type.
- Displays a list of available macro scripts.
- Create, export, edit, and delete macro scripts.
- Assign the selected macro script to the selected measurement objects.
- Flag selected macro scripts as favorites (with a star), so that they are made available in the list of measurement methods.

Once a macro script has been imported into the project, the right side of the Macro Scripts dialog box can be used to:

- Specify or edit the name or the description of the macro script.
- Add, edit, remove, or reorder parameters for the script.

For more information, see the PolyWorks | Inspector Reference Guide.

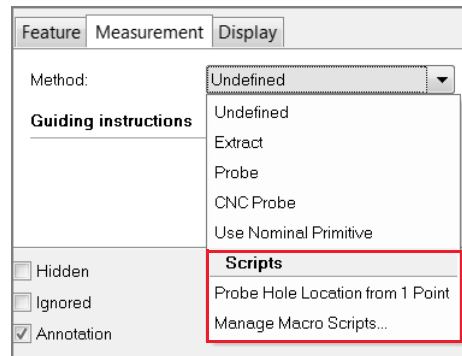
Objective: Measure an object using a scripted measurement method.

Required: The project contains objects as well as macro scripts to measure the objects.

Steps

1. Select the object in the Tree View, then choose *Edit > Object Properties*.
2. On the **Measurement** tab, choose a macro script from the **Method** list. The macro scripts available in the list are those that have been flagged as favorites in the Macro Scripts dialog box.

*Clicking **Manage Macro Scripts** at the bottom of the list opens the Macro Scripts dialog box.*

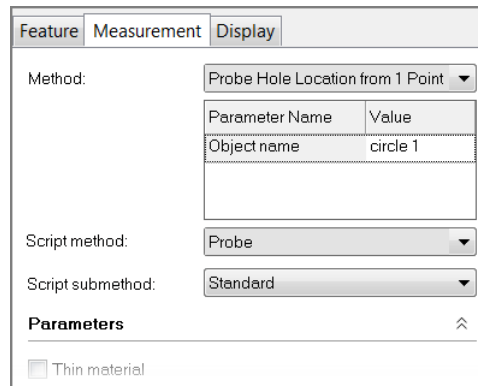


Once the desired script is selected from the **Scripts** list, a list area on the **Measurement** tab displays the parameters defined for the selected script and allows specifying their values.

*The **Object name** parameter and its default value are uneditable.*

3. Choose the **Script method** and, if applicable, the **Script submethod**.

This specifies which measurement method and, when applicable, submethod will be used by the scripted operation to measure the object.



4. Specify other parameters as needed.
The parameters offered vary depending on the selected script method and submethod.
5. Click **Apply** in the property sheet of the object. The object is ready to be measured.

Notes

- The following objects can be measured using a scripted measurement method:

Object Types	Details
Data objects	All types
Features	All types, except angles, distances, patterns, and slabs
Cross-sections	All types
Comparison points	All types
Gauges	All types
Custom measurements	All types
Reference targets	All types, except feature reference target points and cross-section reference target points.

- Several object measurement scripts are included in PolyWorks | Inspector.
 - These scripts are listed in the object measurement script list area of the Macro Scripts dialog box.
 - Parameters have already been defined for these scripts.
 - Once the scripts have been imported into the project, parameter definitions can be edited (except for the **Object name**) and new parameters can be added.
 - These scripts can also be edited and modified.
 - Some of the scripts offered are specifically designed to be used with certain plug-ins.

Saving a compressed copy of a project

When only one specific inspection project needs to be shared or exported, it is possible to create a compressed copy of this project directly from an open PolyWorks | Inspector project.

When opening this compressed file in the PolyWorks Metrology Suite, or in the free PolyWorks | Reviewer™, a new workspace is automatically created.

Objective: Share inspection results by saving a compressed copy of an inspection project.

Required: A completed, open inspection project.

Steps

1. To create a compressed copy of the project, choose *File > Save Compressed Copy*.
2. In the message window that is displayed, click **OK**.

The PolyWorks | Inspector project is saved and closed.

A file browser opens.
3. Browse for the location to save the file and click **Save**.

Results

A copy of the project is saved to a new workspace that is compressed and then saved at the specified location as a .pzip file. It can be opened with the PolyWorks Metrology Suite or the PolyWorks | Reviewer.



Notes

From the Workspace Manager, it is also possible to create a compressed copy of one or more PolyWorks | Inspector projects. Simply open the workspace, select the projects, and choose *File > Export > Objects to Compressed Workspace*.

PolyWorks keyboard shortcuts

General interfaces

Select objects mode	CTRL+SHIFT+SPACEBAR
Switch to local split view mode	CTRL+SPACEBAR
Exit interactive modes	ESC
Cancel macro script	ESC

File >

Open project	CTRL+O
Save project	CTRL+S

Edit >

Undo	CTRL+Z
Redo	CTRL+Y
Undo transformation	CTRL+SHIFT+Z
Duplicate objects (E, I)	CTRL+U
Ignore objects	CTRL+SHIFT+E
Use objects	CTRL+SHIFT+U

Delete >

Objects	SHIFT+DEL
Elements	DEL

Select >

Elements >

All	CTRL+A
Invert	CTRL+I
None	CTRL+N
Interactively	SPACEBAR

Objects >

All	CTRL+SHIFT+A
Invert	CTRL+SHIFT+I
None	CTRL+SHIFT+N

View >		<i>Hide ></i>	
Normal View (E,I)	SHIFT+F9	Objects	CTRL+SHIFT+D
<i>Center></i>		Elements	CTRL+D
All Objects	CTRL+C	<i>Restore ></i>	
Objects	CTRL+SHIFT+C	Objects	CTRL+SHIFT+R
<i>Standard Views ></i>		Elements	CTRL+R
+X	SHIFT+F1	<i>Keep></i>	
-X	SHIFT+F2	Objects	CTRL+SHIFT+K
+Y	SHIFT+F3	Elements	CTRL+K
-Y	SHIFT+F4		
+Z	SHIFT+F5		
-Z	SHIFT+F6		
<i>Rotate 90° around View</i>	SHIFT+right-click		
To display the color map	CTRL+T		

Object Display Options

Static display >

Flat	CTRL+F
Flat + Wireframe (E, I)	CTRL+L
Point	CTRL+P
Smooth	CTRL+G
Wireframe	CTRL+W

Dynamic display >

Flat	CTRL+SHIFT+F
Flat + Wireframe (E,I)	CTRL+SHIFT+L
Point	CTRL+SHIFT+P
Smooth	CTRL+SHIFT+G
Wireframe	CTRL+SHIFT+W
Bounding box	CTRL+SHIFT+B

Specific interfaces

IMAlign

Edit >

Lock Images	CTRL+SHIFT+L
Unlock Images	CTRL+SHIFT+O

Align >

Start Best-Fit alignment	CTRL+F2
Stop Best-fit alignment	CTRL+F3
Point Pairs	CTRL+F5

PolyWorks|Inspector

Display menu on objects and annotations	Right-click
Position annotations	ALT + left-click
Select > Data Points > Measured Data Points	CTRL+M
Select > Data Points > Measured Data Points of Local Planes	CTRL+B
Tools > Reference Objects > Correct Inverted CAD Surfaces	CTRL+Q

Getting help

Users can access help resources both within the software interface and externally. Resources include reference guides, illustrated tips, video tutorials, and webinars.

Getting help through the software interface



Reference Guides

PDF documents located on the *Help* menu of the Workspace Manager and of each module. The reference guides define all of the functionalities of the PolyWorks Metrology Suite.



Question mark

Found on the title bar of displayed dialog boxes. Clicking the question mark icon opens the appropriate reference guide at the section documenting the active dialog box.



F1

Pressing F1 opens the reference guide to the section documenting the active dialog box. If the 3D scene has the focus during an interactive mode, pressing F1 opens the How To window.

Release Notes

A PolyWorks Metrology Suite intermediate release is produced approximately once a month. These releases contain new tools along with bug fixes. The "Release Notes" document gives the details of the intermediate release and is available through *Help > Release Notes*.

Getting help outside of the software interface

The website

The InnovMetric website (www.innovmetric.com) offers technical webinars ([ABOUT US > Webinars](#) page), which introduce new major functionalities and demonstrate the use and benefits of existing PolyWorks tools.

The PolyWorks Technical Support Zone (TSZ)

The secure Technical Support Zone offers PolyWorks users 24/7 online access to:

Software Releases:

This is where you will find the downloadable versions of the PolyWorks Metrology Suite and the PolyWorks Collaborative Suite, as well as access to the Release Notes.

Knowledge Base:

Provides articles, illustrated tips, and video tutorials to help you get the most out of PolyWorks, including practical tips and tricks, workflow-oriented videos to help you learn the basic capabilities, videos on how to solve PolyWorks training session exercises, and technical webinars designed to boost your PolyWorks skills.

Support:

Get direct assistance from a PolyWorks expert by creating and submitting a Technical Support Zone Case where you can confidentially share files for troubleshooting purposes. You can also access articles related to your questions as well as your other open or closed cases.

Macro Zone:

The PolyWorks Metrology Suite offers the possibility of creating macro scripts to automate repetitive tasks or generate a personal set of tools to save time and perform tasks in a predefined, controlled manner. In the Macro Zone, you will find a variety of useful macro scripts developed by our Technical Support team.

Forum:

A platform where you can ask questions and share ideas and comments with the PolyWorks user community.

Global Search Tool:

A powerful tool that searches the entire site content to find the information you are looking for.