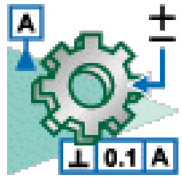
3D Functional Tolerancing & Annotation



Overview

Welcome to the Functional Tolerancing & Annotation User's Guide!
This guide is intended for users who need to become quickly familiar with the product.

This overview provides the following information:

- Functional Tolerancing & Annotation in a Nutshell
- Before Reading this Guide
- Getting the Most Out of this Guide
- Accessing Sample Documents
- Conventions Used in this Guide

Functional Tolerancing & Annotation in a Nutshell

Functional Tolerancing & Annotation lets you easily define and manage 3D tolerance specifications and annotations directly on 3D parts or products.

The intuitive interface of the product provides an ideal solution for new application customers in small and medium-size industries, looking to reduce reliance on 2D drawings, and increase the use of 3D as the master representation for driving from design to manufacturing engineering process.

Annotations in Functional Tolerancing & Annotation can be extracted, using the annotation plane concept in the Generative Drafting product.

The product elements can be reviewed using specific functionalities, which constitute comprehensive tools for the interpretation of tolerancing annotations.

This manual is intended for users who need to specify tolerancing annotations on 3D parts or on 3D products. It assists designers in assigning the correct tolerances on the selected surfaces by:

- Selecting the surfaces to be toleranced.
- Choosing among the available options, the tolerance types, the modifiers, etc. The system offers a choice of options which are consistent with the selected surfaces.
- Entering the tolerance value. The tolerance annotation is then created and displayed around the 3D geometry. It is also located and oriented in an annotation plane, using a standardized model (usual standards: ISO, ASME / ANSI).

As a consequence, designers do not need to wonder whether the tolerancing syntax is correct, because this syntax is directly elaborated with regard to the chosen tolerancing standards (ISO, ASME / ANSI).

Designers are ensured that their tolerancing schema is consistent with the part geometry. They do not need to be tolerancing experts, having in mind all the complex standardized tolerancing rules. Moreover, the tolerancing specifications will remain consistent whatever the geometrical modifications are.

See Reference Information for further detail.

Functional Tolerancing & Annotation allows you to work with the cache system, in other words in Visualization Mode. Annotations can be recorded or not with the related cgr document using this mode. From the cgr document, annotations are displayed and they can be queried and filtered. Of course, when you are editing an annotation the related document is automatically switched to Design Mode.

Note that 3 workbenches are available depending on whether you are working on:

- a part (Functional Tolerancing and Annotation workbench)
- a product (Product Functional Tolerancing and Annotation workbench)
- a process (Process Tolerancing and Annotation workbench).

This guide is intended for users of all 3 workbenches, as the functionalities available are exactly the same from one workbench to another. However, note that the scenarios provided in this guide use parts (CATPart documents) as examples.

Before Reading this Guide

Before reading this guide, you should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars. Therefore, we recommend that you read the *Infrastructure User's Guide* that describes generic capabilities common to all Version 5 products. It also describes the general layout of V5 and the interoperability between workbenches.

You may also like to read the following complementary product guides, for which the appropriate license is required:

- *Product Structure*
- *Part Design*
- *Generative Drafting*

Getting the Most Out of this Guide

To get the most out of this guide, we suggest that you start reading and performing the step-by-step Getting Started tutorial.

Once you have finished, you should move on to the User Tasks section, which deals with handling all the product functions.

The Workbench Description section, which describes the Functional Tolerancing & Annotation workbench, and the Customizing section, which explains how to set up the options, will also certainly prove useful.

Navigating in the Split View mode is recommended. This mode offers a framed layout allowing direct access from the table of contents to the information.

Accessing Sample Documents

To perform the scenarios, sample documents are provided all along this documentation. For more information about this, refer to Accessing Sample Documents in the Infrastructure User's Guide.

Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications.

Graphic Conventions

The three categories of graphic conventions used are as follows:

- Graphic conventions structuring the tasks
- Graphic conventions indicating the configuration required
- Graphic conventions used in the table of contents

Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted as follows:

This icon...	**Identifies...**
	estimated time to accomplish a task
	a target of a task
	the prerequisites
	the start of the scenario
	a tip
	a warning
	information
	basic concepts
	methodology
	reference information
	information regarding settings, customization, etc.
	the end of a task

	functionalities that are new or enhanced with this release
	allows you to switch back to the full-window viewing mode

Graphic Conventions Indicating the Configuration Required

Graphic conventions indicating the configuration required are denoted as follows:

This icon...	Indicates functions that are...
P1	specific to the P1 configuration
P2	specific to the P2 configuration
P3	specific to the P3 configuration

Graphic Conventions Used in the Table of Contents

Graphic conventions used in the table of contents are denoted as follows:

This icon...	Gives access to...
	Site Map
	Split View mode
	What's New?
	Overview
	Getting Started
	Basic Tasks
	User Tasks or the Advanced Tasks
	Workbench Description
	Customizing
	Reference
	Methodology
	Glossary
	Index

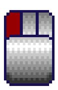
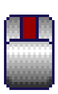
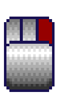
Text Conventions

The following text conventions are used:

- The titles of CATIA, ENOVIA and DELMIA documents *appear in this manner* throughout the text.
- **File** -> **New** identifies the commands to be used.
- Enhancements are identified by a blue-colored background on the text.

How to Use the Mouse

The use of the mouse differs according to the type of action you need to perform.

Use this mouse button...	Whenever you read...
	• Select (menus, commands, geometry in graphics area, ...) • Click (icons, dialog box buttons, tabs, selection of a location in the document window, ...) • Double-click • Shift-click • Ctrl-click • Check (check boxes) • Drag • Drag and drop (icons onto objects, objects onto objects)
	• Drag • Move
	• Right-click (to select contextual menu)

What's New?

New Functionalities

Annotations and Cache System

This reference chapter described how and what it is possible to do with 3D annotations in Visualization mode.
All the annotation contained in a product or process document are displayed and checked in Visualization mode.
These annotations can be edited, related component in Visualization mode can be switched to Design mode if needed.

Technological Package for Functional Tolerancing & Annotation

This functionality allows you to store Functional Tolerancing & Annotations data in a technological package to be saved in ENOVIA VPM context.
Functional Tolerancing & Annotations data as 3D annotations, 3D views/Annotation planes, captures, construction geometry, and partial surfaces.

Enhanced Functionalities

Filtering Annotations

You are now able to display, query and filter 3D annotations in visualization mode.

Creating an Automatic Constructed Geometry

This command allows to create common axis of coaxial cylinders, cylindrical surface defined by its axis and passing through one or two cylinders.

Dimensioning and Tolerancing Threads

You can associate general tolerances with semantic toleranced dimension through a new option.

Creating Dimensions

You can now edit the dimension orientation after the creation, see Setting Dimension Representations.

You can now edit the angular dimension angle sector after the creation, see Creating Angular Dimensions.

Generating Dimensions

You can generate dimension from user feature specifications.

Customizing Settings

Always try to create semantic general tolerances on dimensions

Tolerancing option to set as semantic a general tolerances dimension during its creation.

Transparency

Display option to define the surface color transparency of a partial surface.

Transparency

Constructed Geometry option to define the surface color transparency of a constructed geometry.

General tolerance class

Tolerances option to define the general tolerance class for angular size.

General tolerance class

Tolerances option to define the general tolerance class for linear size.

Cgr Management for 3D Annotation

A new option allows you to take into account 3D Annotations in cgr documents. 3D Annotations are now displayed in Visualization mode.

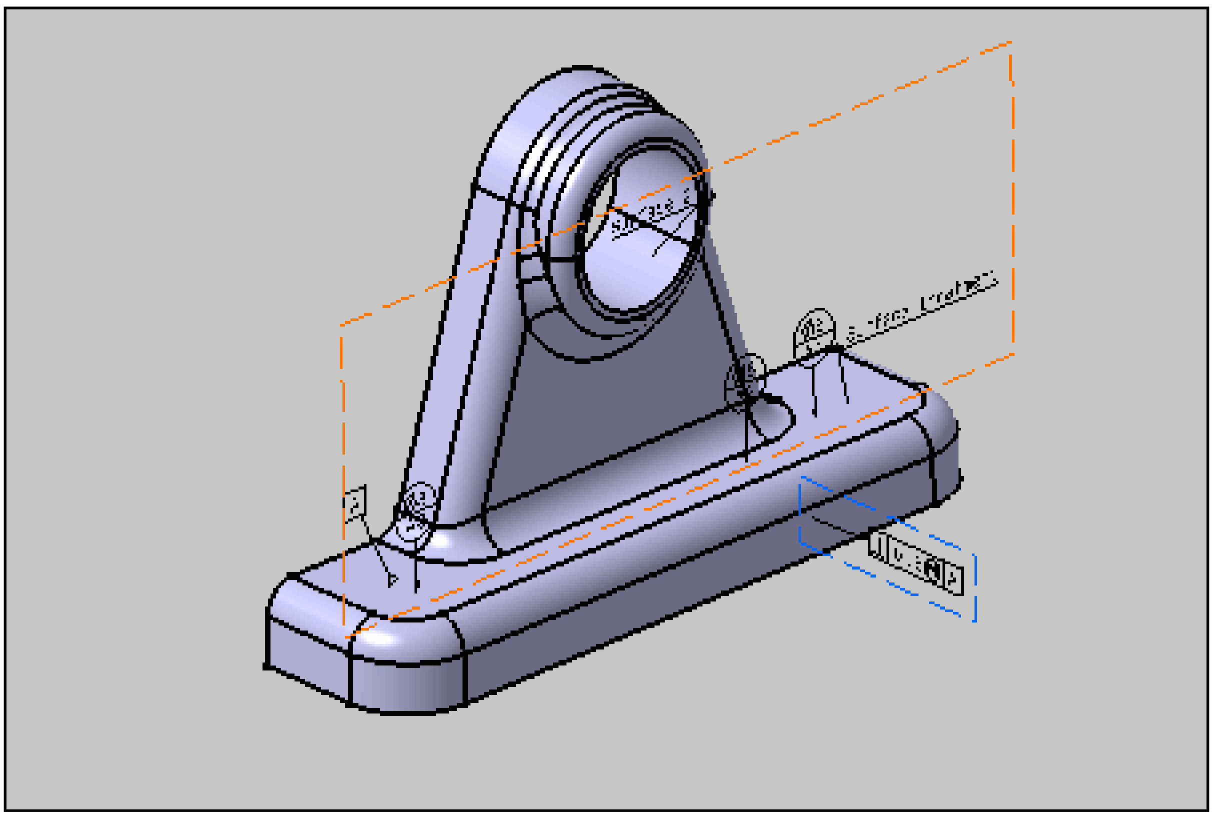
Getting Started

Before we discuss the detailed instructions for using the Functional Tolerancing & Annotations or the Product Functional Tolerancing & Annotations workbench, the following scenario aims at giving you a feel for what you can do.
You just need to follow the instructions as you progress.

The Getting Started section is composed of the following tasks:

This scenario should take about 15 minutes to complete.

Eventually, the toleranced part will look like this:



Entering the 3D Functional Tolerancing & Annotation Workbench

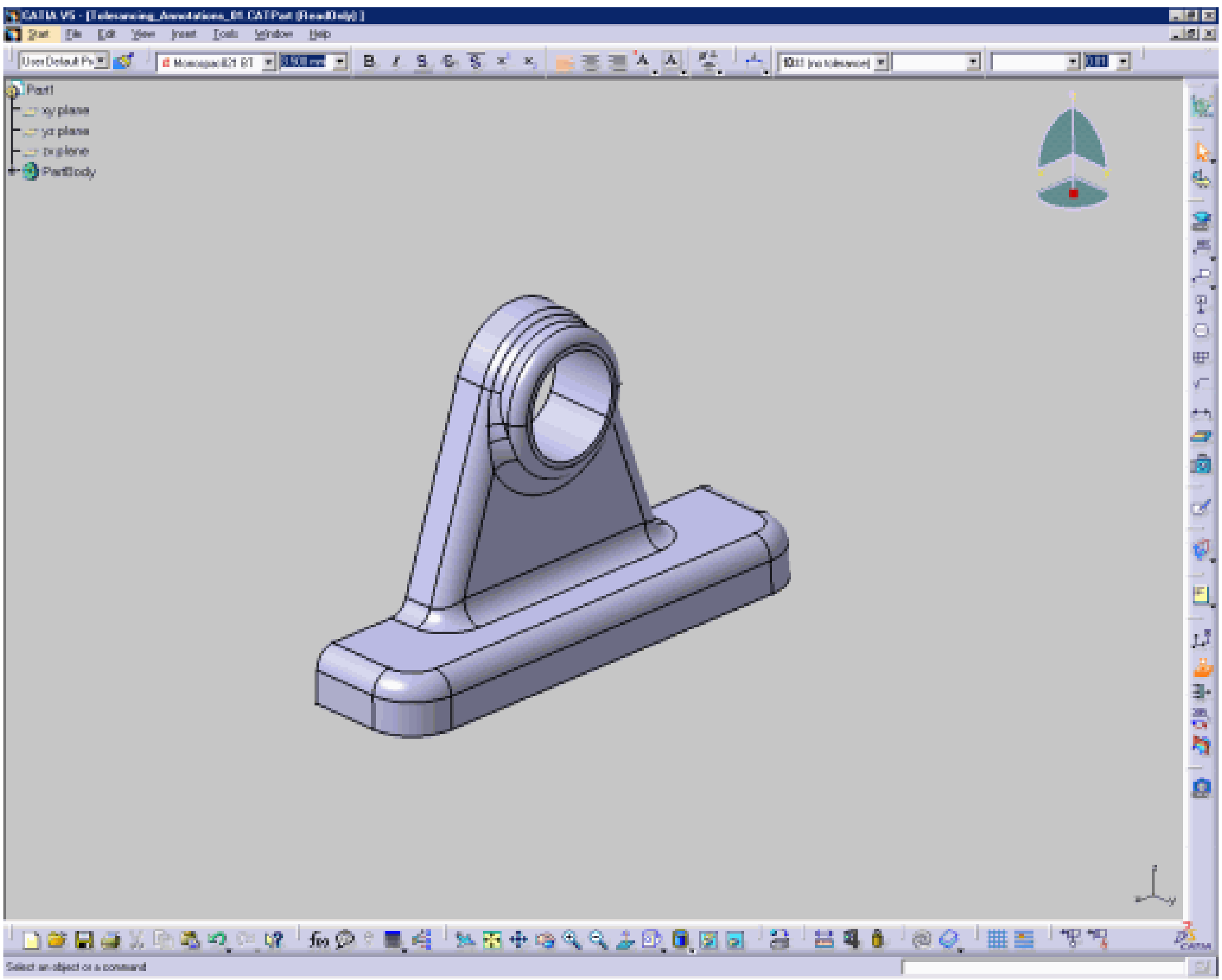
This task shows you how to enter the workbench and open the document you need for performing this tutorial.

1. Select the **Start -> Mechanical Design -> Functional Tolerancing & Annotation** command to launch the workbench.

   The 3D Functional Tolerancing & Annotation workbench is opened.
   The commands are available in the toolbar to the right of the application window.

2. Open the Tolerancing_Annotations_01 CATPart document.

   This is what you get:



To know how to use the commands available in the Standard and View toolbars located in the application window border, please refer to *Infrastructure User's Guide Version 5*.

Choosing the Standard

This task shows you how to set the standard you need for tolerancing your part.

You must choose a standard before creating the first annotation in a document. See also Standards.

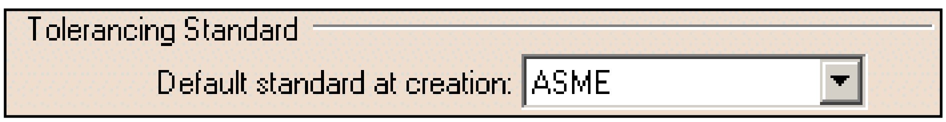
1. Select the **Tools -> Options** command.

   The **Options** dialog box is displayed

2. Click **Mechanical Design** then **Functional Tolerancing** in the left-hand column.

   See Tolerancing setting for further detail.

3. If not still done, set **ASME** as the standard to be used in the tutorial.



4. Click **OK** to validate and close the dialog box.

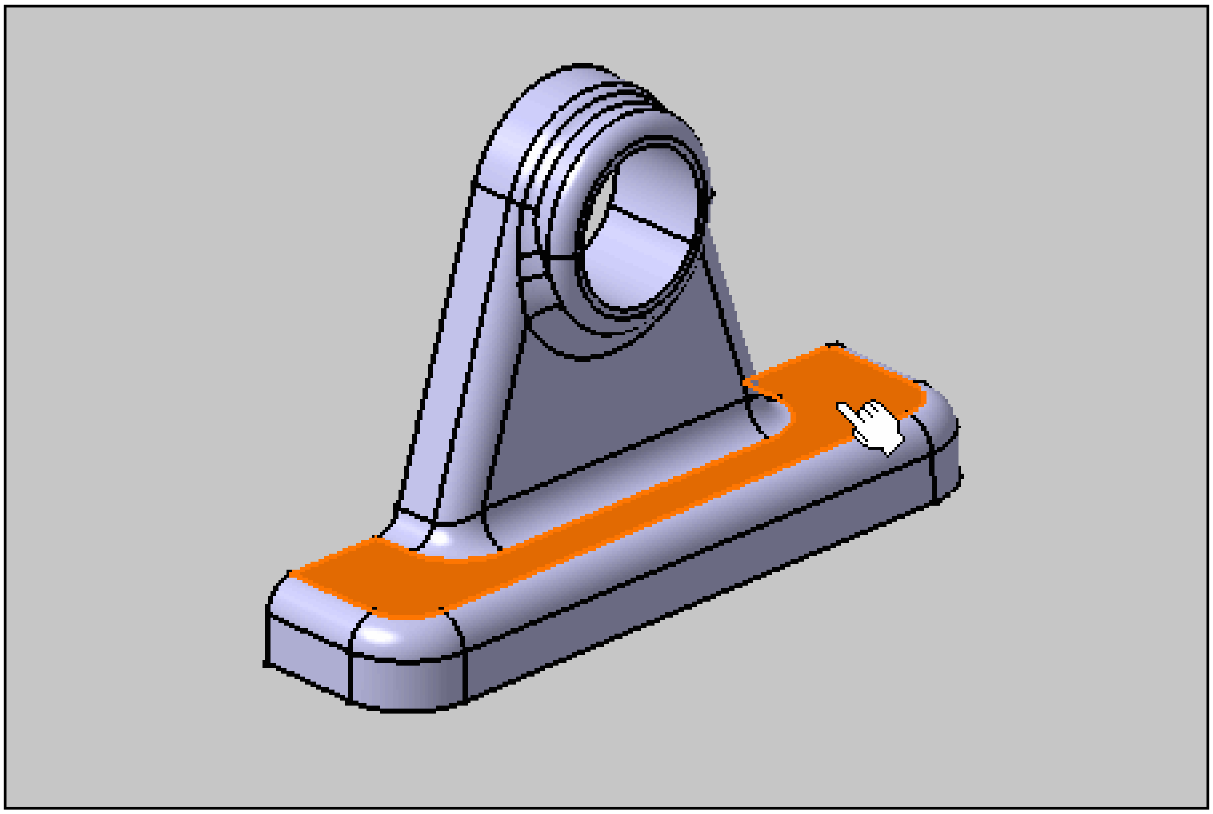
Note that this choice of standard must be expressed prior to specifying any tolerance. After any creation in the workbench, the standard may be modified but the corresponding syntax and semantic variation will not be taken into consideration.

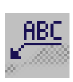
Creating Annotation Texts

This task shows you how to create two textual annotations related to the 3D geometry of the part.

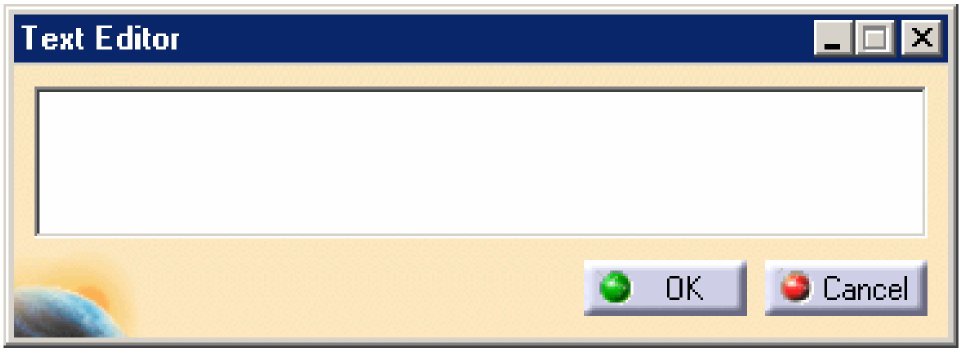
If you wish to improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation. All the screen captures have been performed according to this option.

1. Click the face as shown to define the surface and the location for the arrow head of the leader line.



2. Click the **Text** icon: 

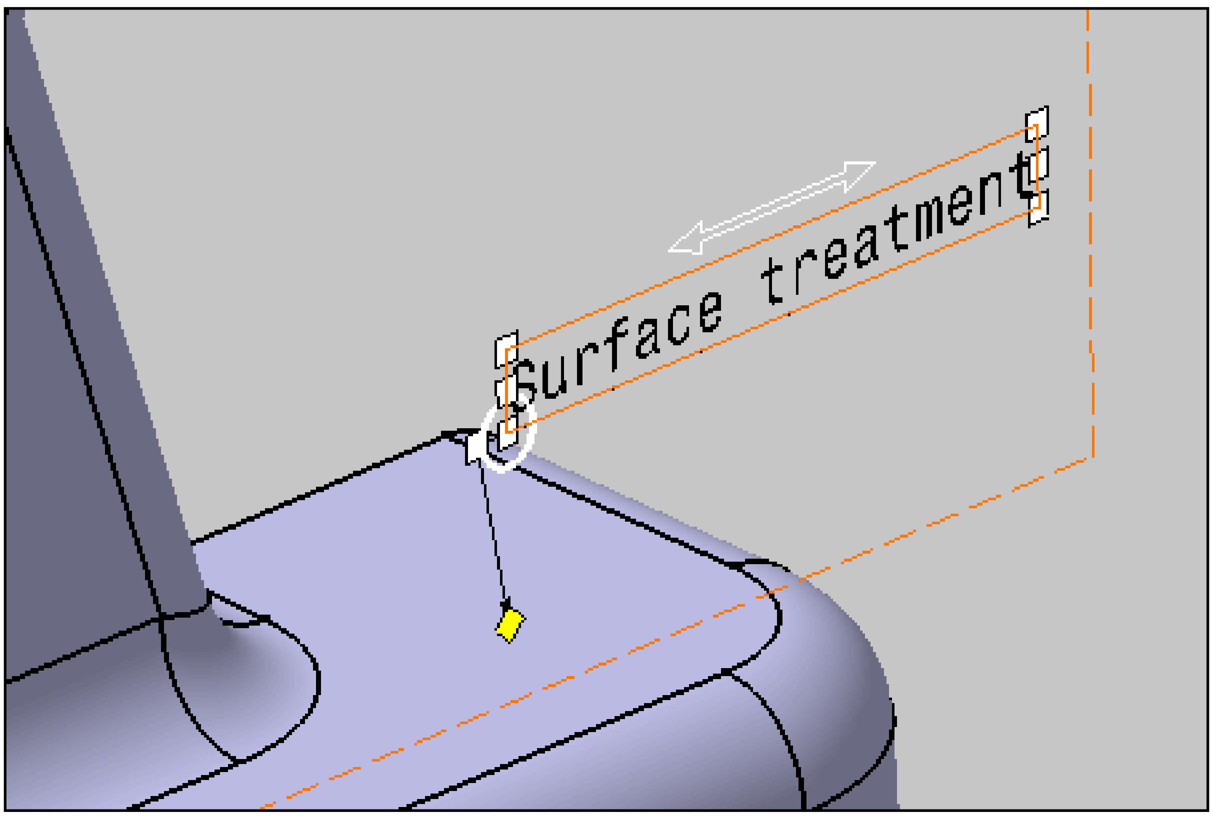
The **Text Editor** dialog box is now displayed.



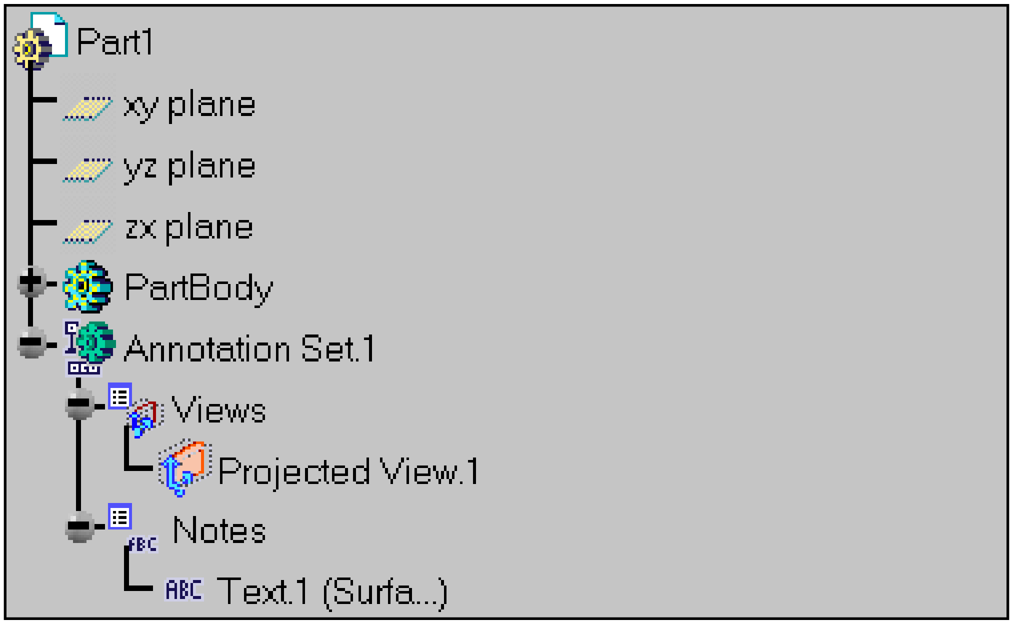
3. Enter **Surface treatment** in the dialog box.

4. Click **OK** to end the text creation.

The text is displayed in the 3D space in an annotation plane.

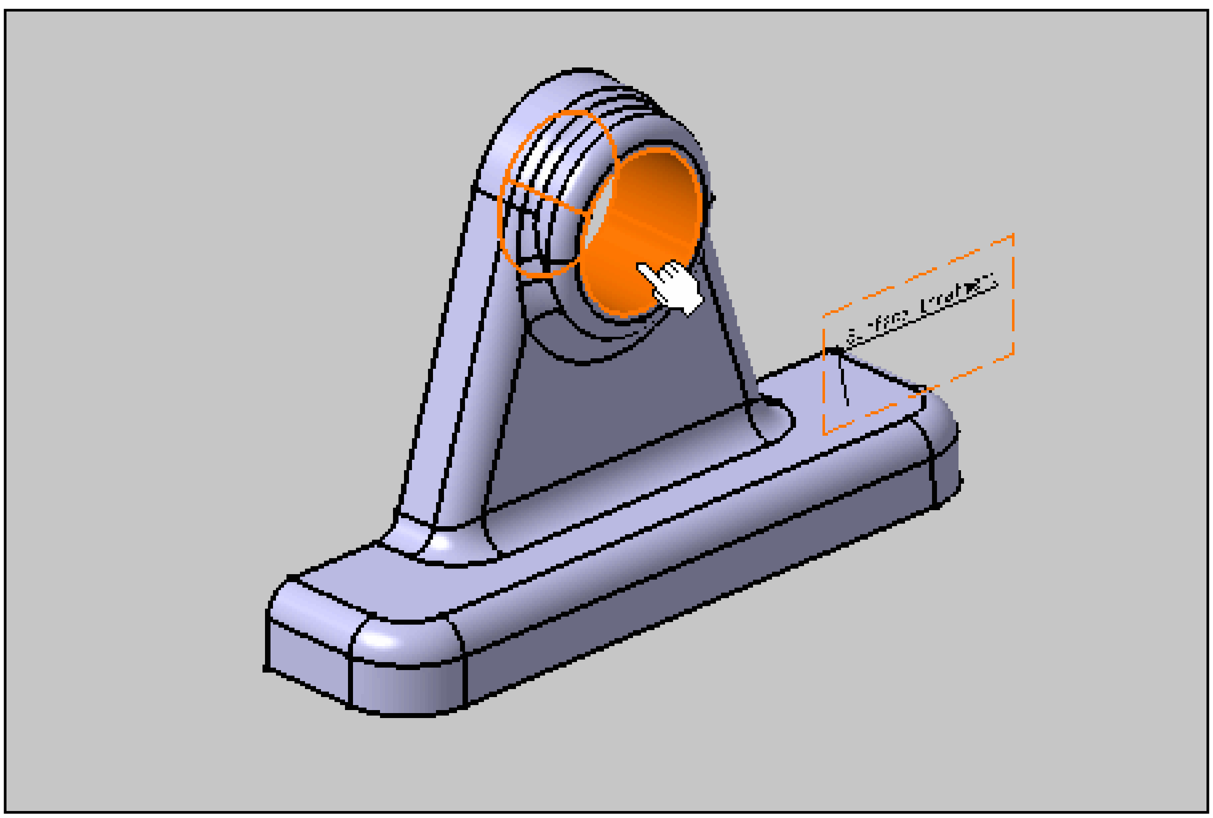


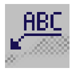
The textual annotation identified as Text.1 is added to the specification tree.
Note also that the application creates a projected view as shown in the tree. This view is automatically created when any view have been created yet or any existing view cannot be re-use.



Any front view created corresponds to an annotation plane (called "Front view annotation plane" too in the workbench).
If no annotation plane still exists, one is by default created when specifying the first annotation.

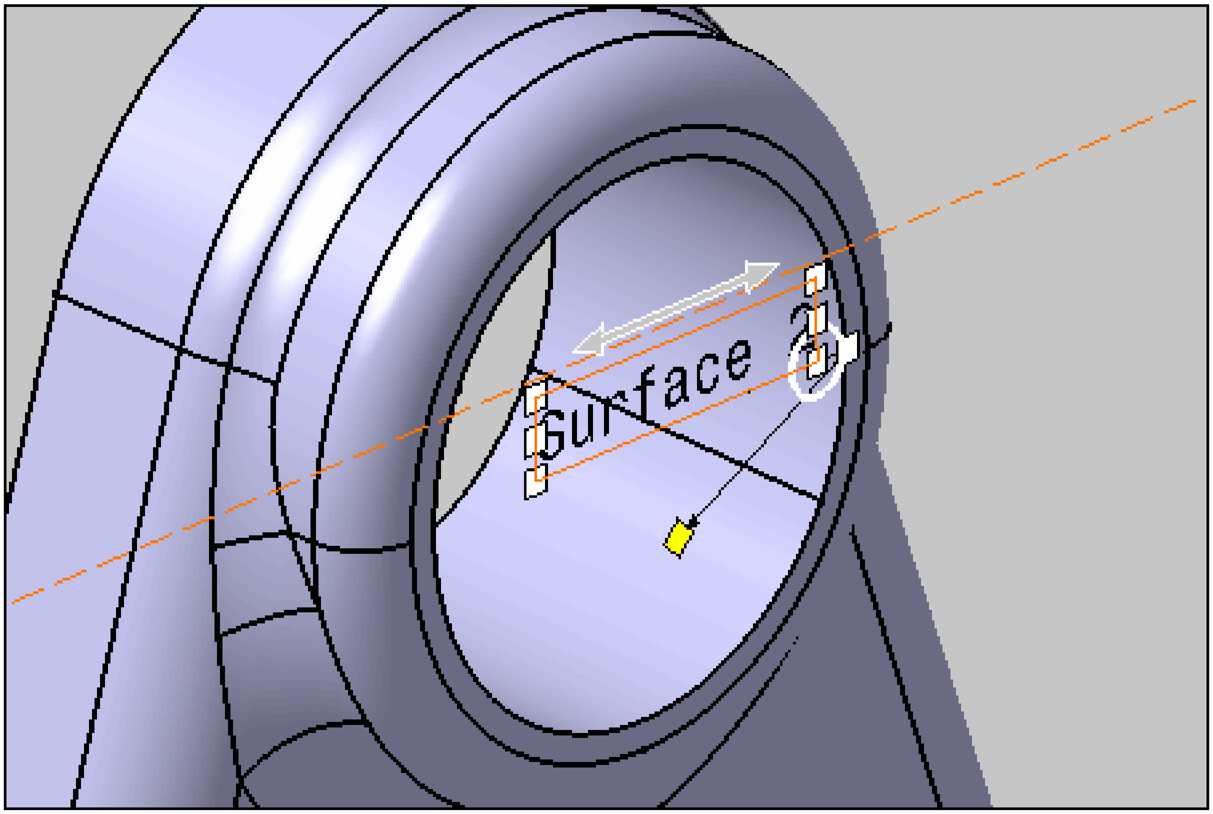
5. To create another text annotation, select the inner cylindrical face:



6. Click the **Text** icon: 

7. Enter "Surface 2" in the dialog box and click **OK**.

You have created a second textual annotation in the same front view.



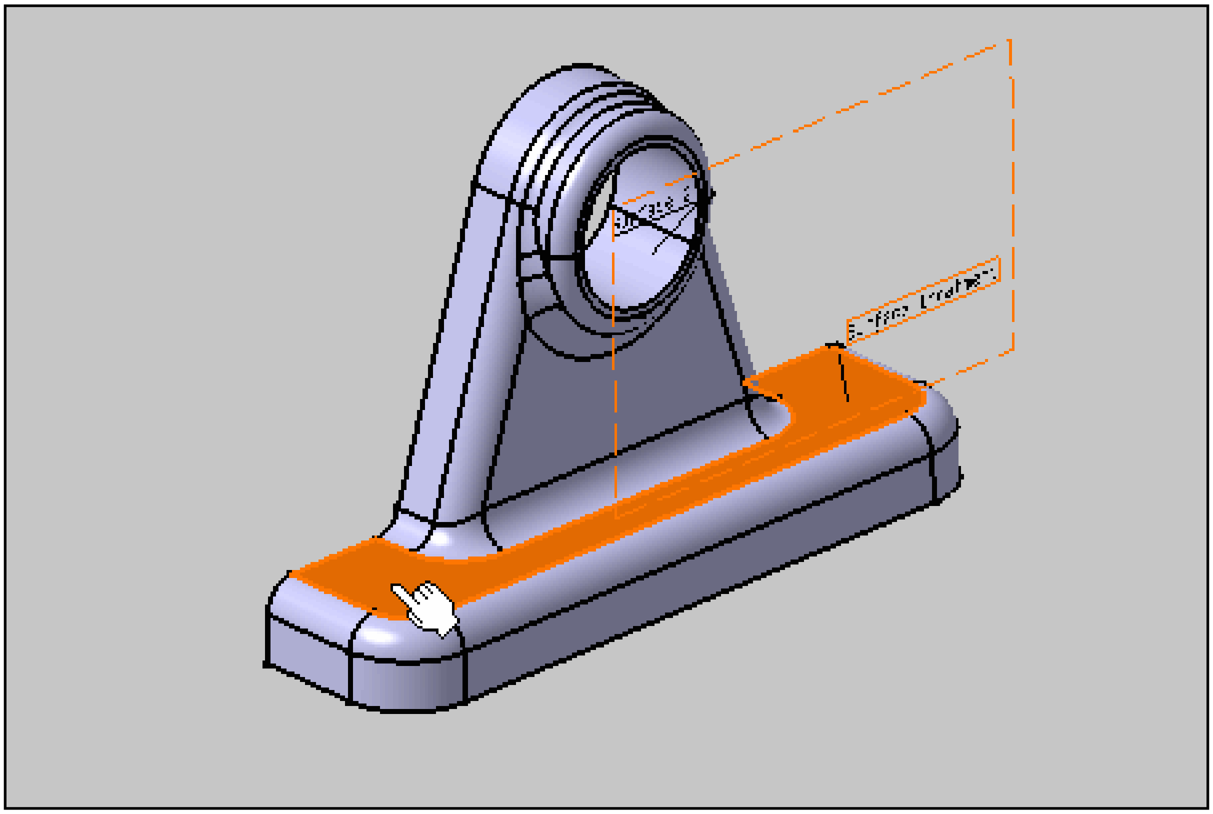
For more about textual annotations, refer to Specifying textual annotations.

Creating a Simple Datum

This task shows you how to create a simple datum on a surface.

Datum elements are involved in geometric tolerancing specifications. For instance, when specifying an orientation or a position tolerance, you need to refer to datum elements for the specification.

1. Select the attachment surface to be specified as datum.

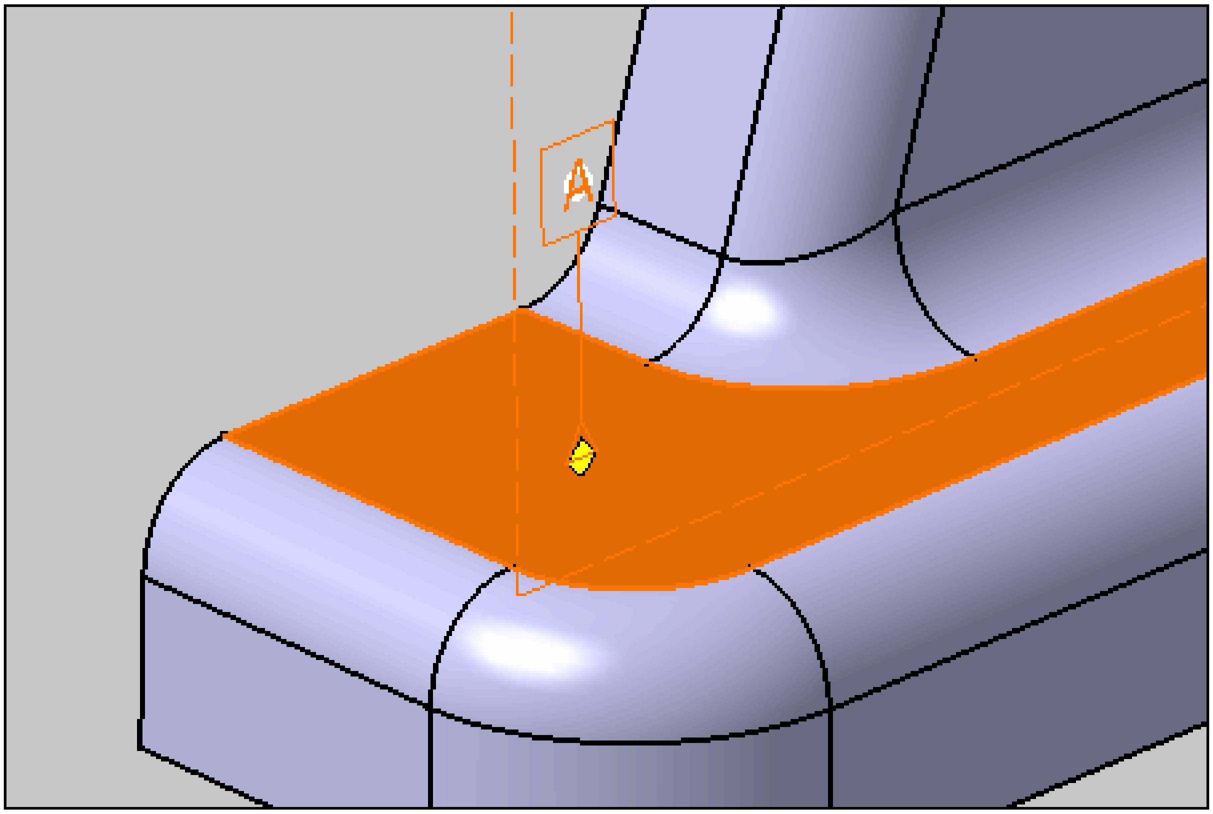


2. Click the **Datum Feature** icon: 

The **Datum Feature Modification** dialog box that appears displays "A" as the default identifier.

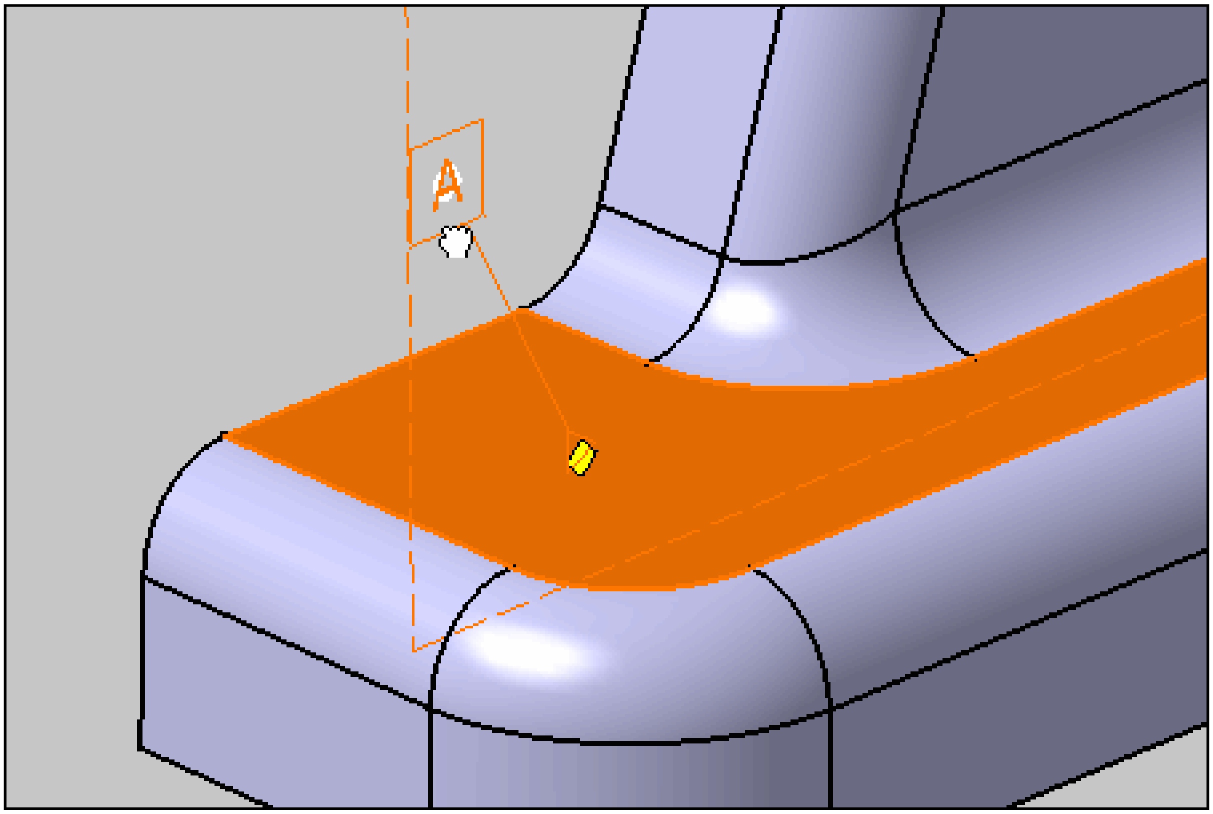
3. Click **OK** to create the datum if the identifier corresponds to your choice.

The datum feature is created in a specific annotation plane. The Datum entity is added to the specification tree. The datum is a 3D annotation without any semantic link to the geometrical tolerancing.



The display of this datum label corresponds to the ANSI normative reference.

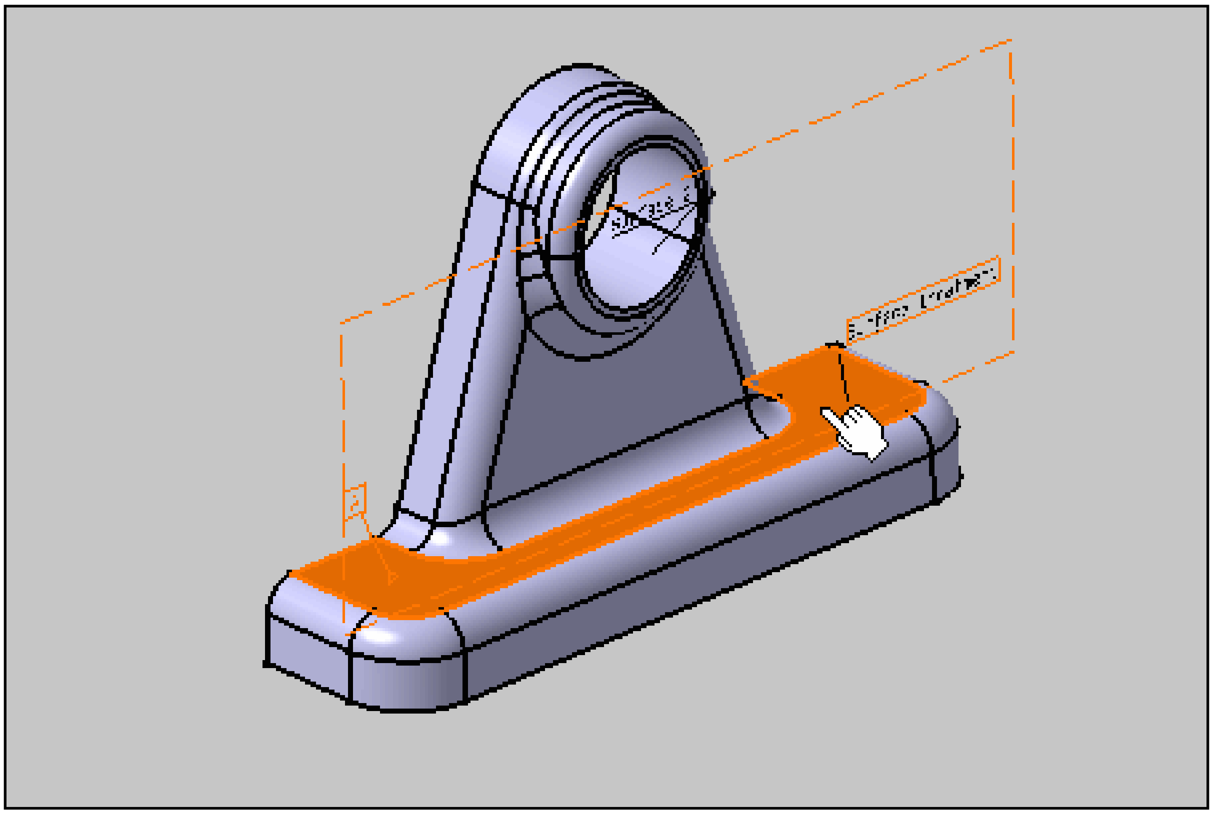
4. Select the datum and drag it. You can notice that it remains in the annotation plane.

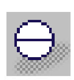

A

Creating Datum Targets

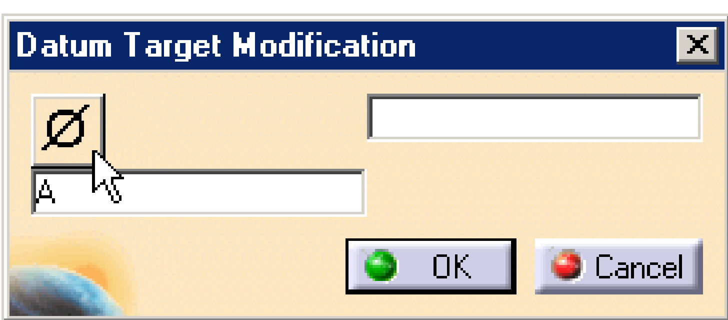
This task shows you how to create three datum targets.

1. Select the face as shown.

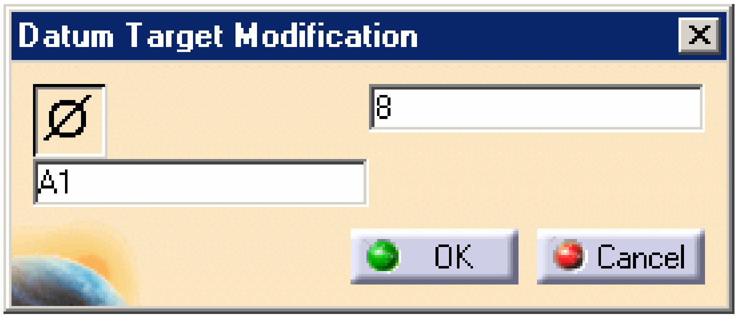


2. Click the **Datum Target** icon: 

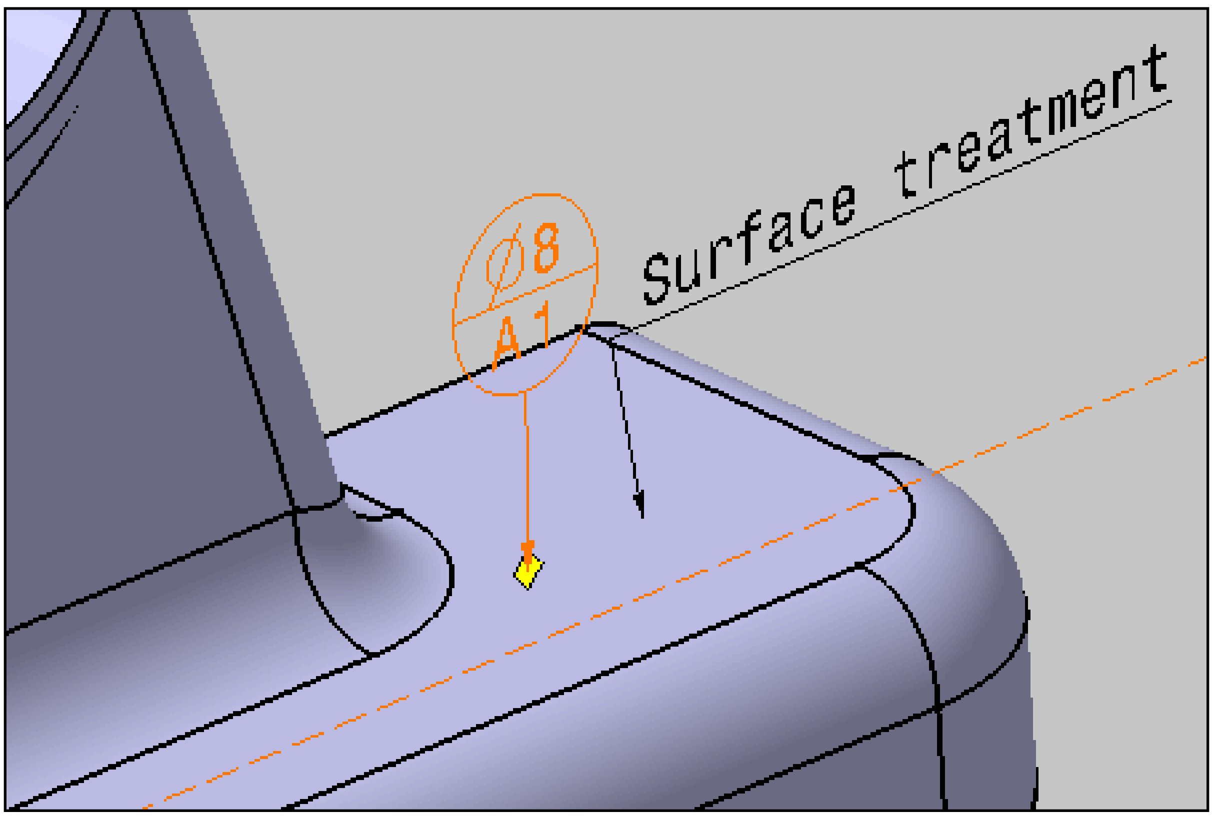
3. In the dialog box that appears, click the diameter icon.



4. Enter 8 in the field opposite and enter "A1" in the field to the left.

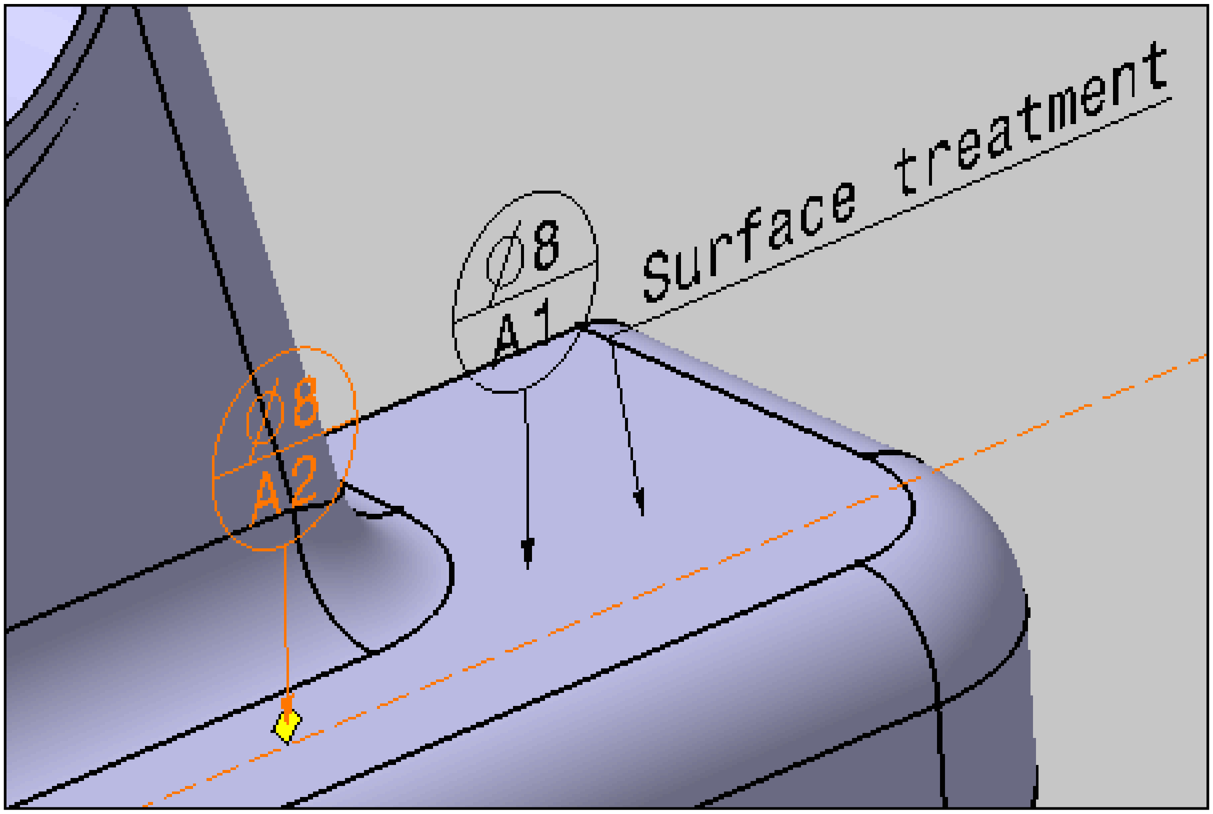


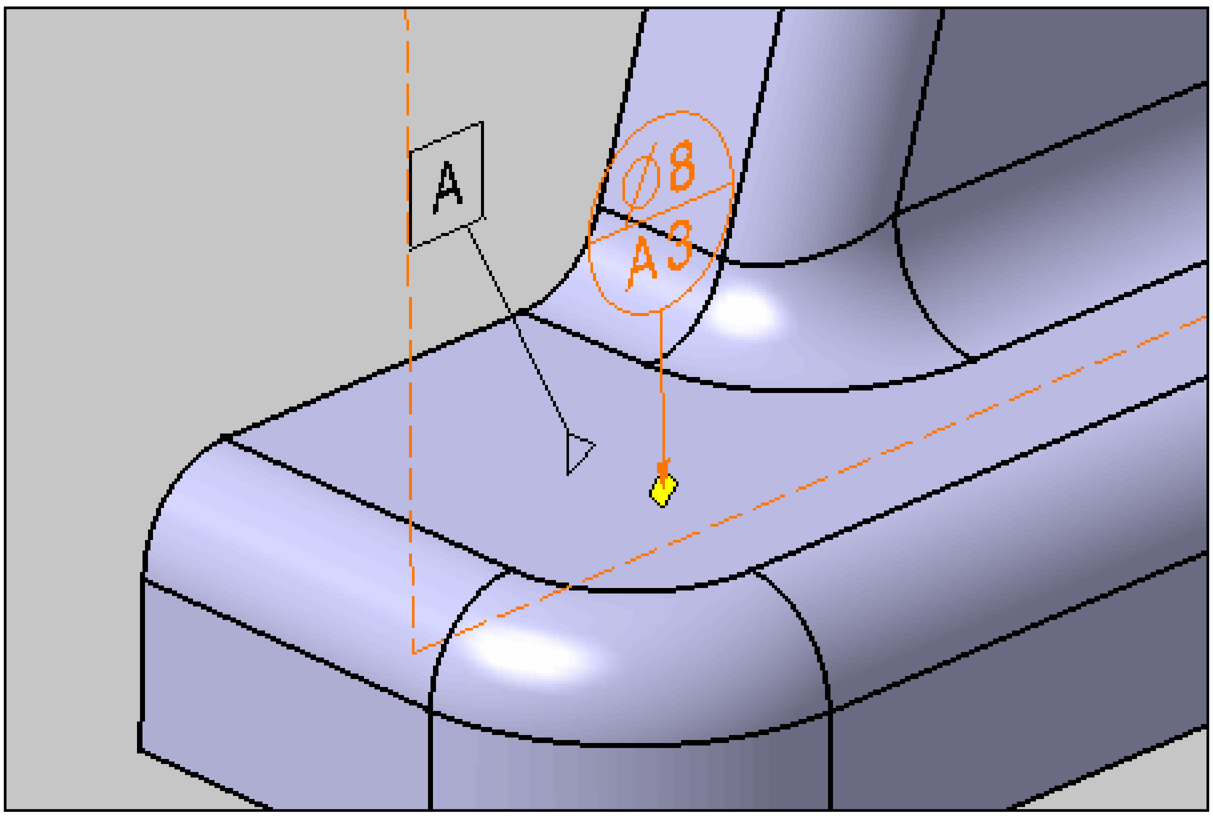
5. Click OK to validate.



You have created a datum target on datum plane A. The datum target corresponds to a 8mm-diameter surface. The name of the target is "A1".

6. Repeat the previous steps to specify two additional datum targets: A2 and A3.

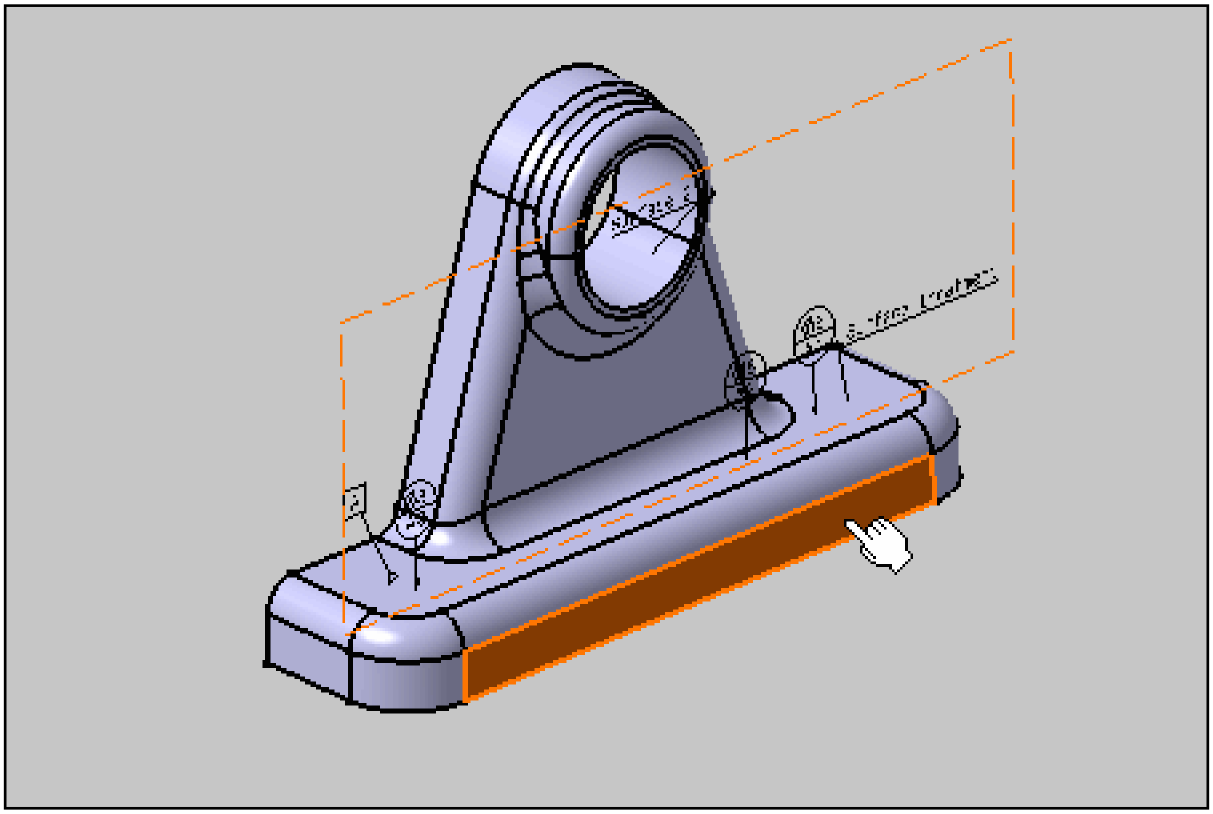

Surface treatment
Ø8
A1
Ø8
A2

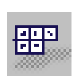

A
Ø8
A3

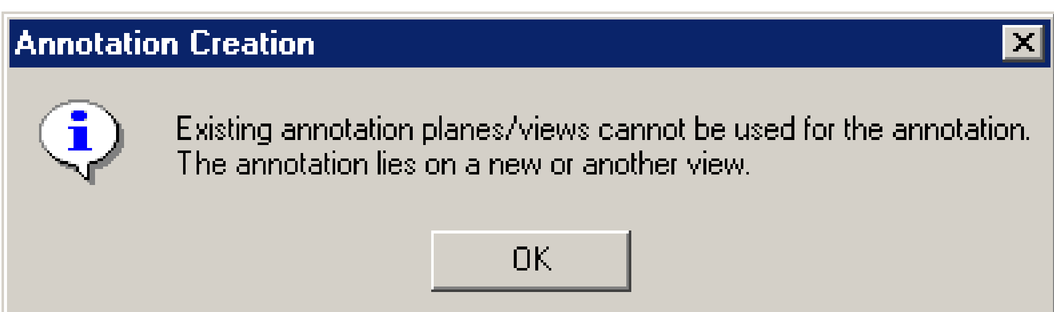
Creating a Geometrical Tolerance

This task shows you how to create a geometrical tolerance directly on the 3D geometry. Geometrical tolerances are specifications included in a tolerance frame.

1. Select the front planar surface as shown:

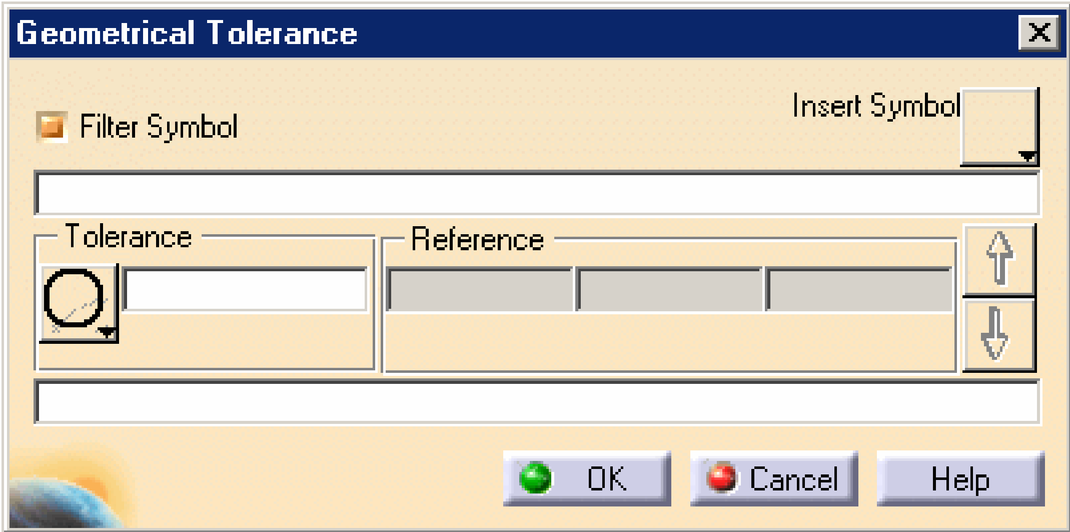


2. Click the **Geometrical Tolerance** icon: 



A message window appears informing you that you cannot use the active view. Therefore, the application is going to display the annotation in an annotation plane normal to the selected face.

3. Click **OK** to close the message window.



The **Geometrical Tolerance** dialog box is displayed.

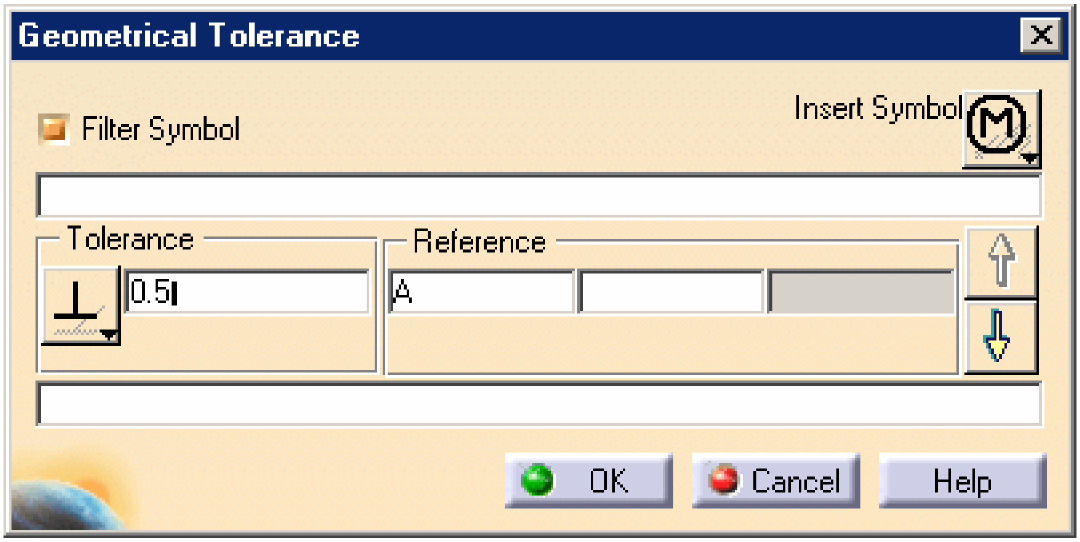
To create a geometrical tolerance you need to specify one geometric tolerancing symbol and fill in the tolerance value field.

When fulfilling the second line "Spec 2...", a second geometrical tolerance will be created. Both tolerancing specifications will be displayed as grouped.

4. Set the perpendicularity symbol to define the tolerance.

5. Enter the value of the tolerance: **0.5** and insert the **Maximum Material Condition** symbol modifier

6. Enter **A** as reference.

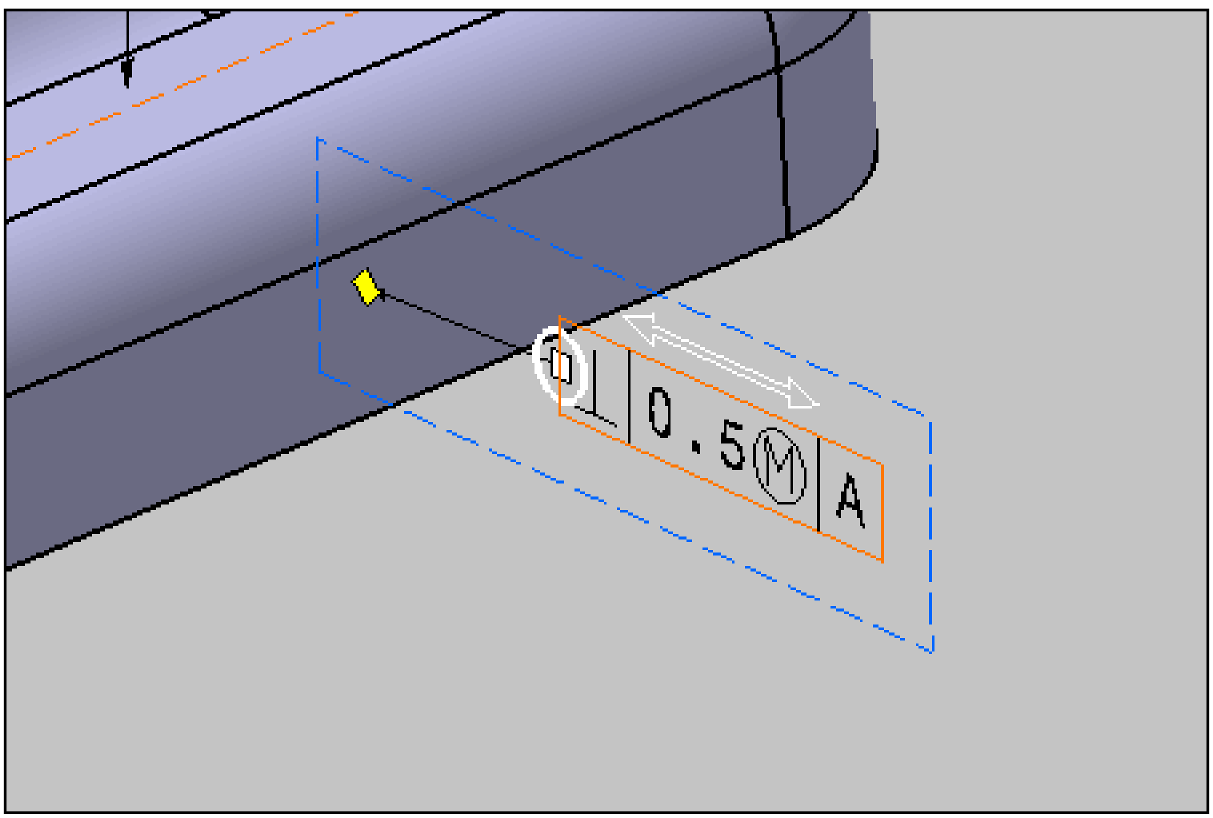


Modifiers are not displaying in tolerance and reference fields and appear with a "|" character.

7. Click **OK**.

The geometrical tolerancing annotation is attached to the 3D part. The geometrical tolerance entity is added to the specification tree.

You have specified a perpendicularity of the front planar surface regarding to A datum surface. This toleranced surface shall be in 0.5-wide tolerance zone, on which an Maximum Material Condition (MMC) is applied.

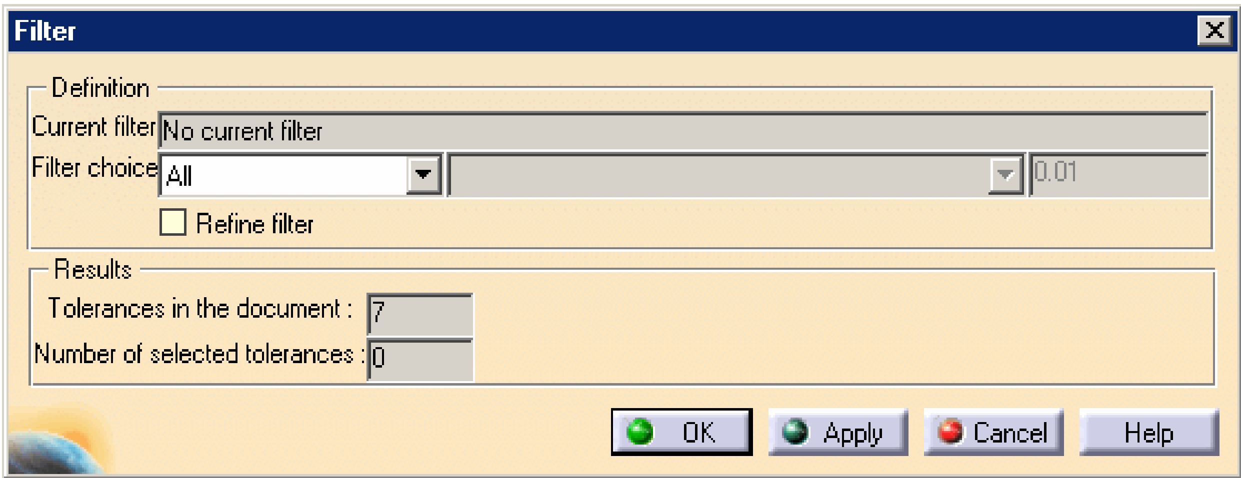


Filtering Annotations

This task shows you how to filter the display of annotations. These filtering options lets you focus on your area of investigation.

1. Click the **Filter** icon: 

The **Filter** dialog box is displayed.



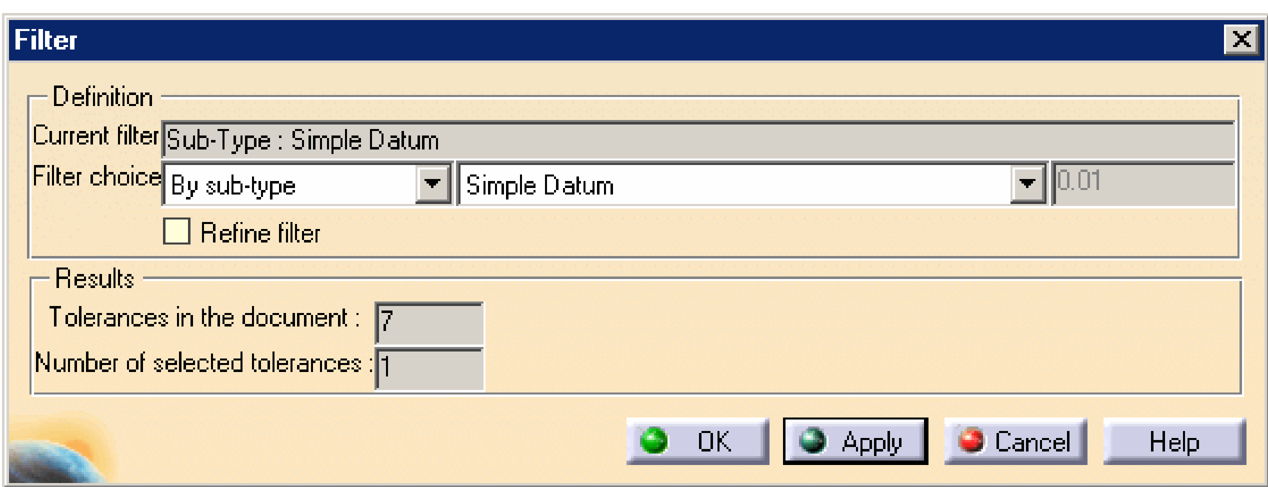
2. Set the **Filter** choice field to "By sub-type".

3. Set the **Simple Datum** sub-type.

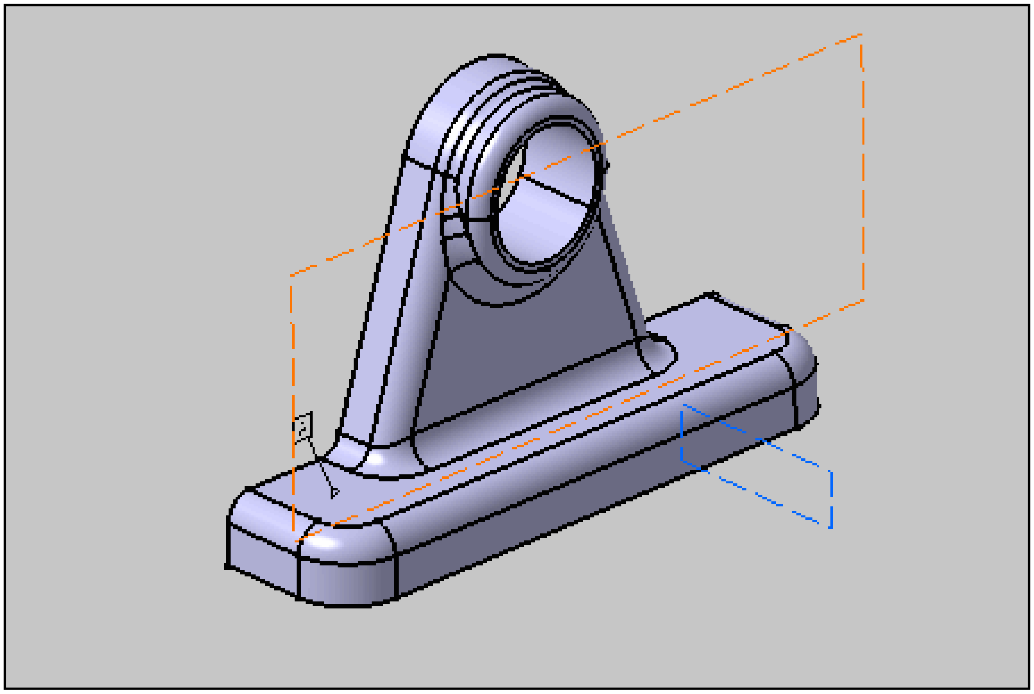
Checking the **Refine filter** option filters out tolerances still filtered.

4. Click **Apply**.

The Number of selected tolerances field displays "1".

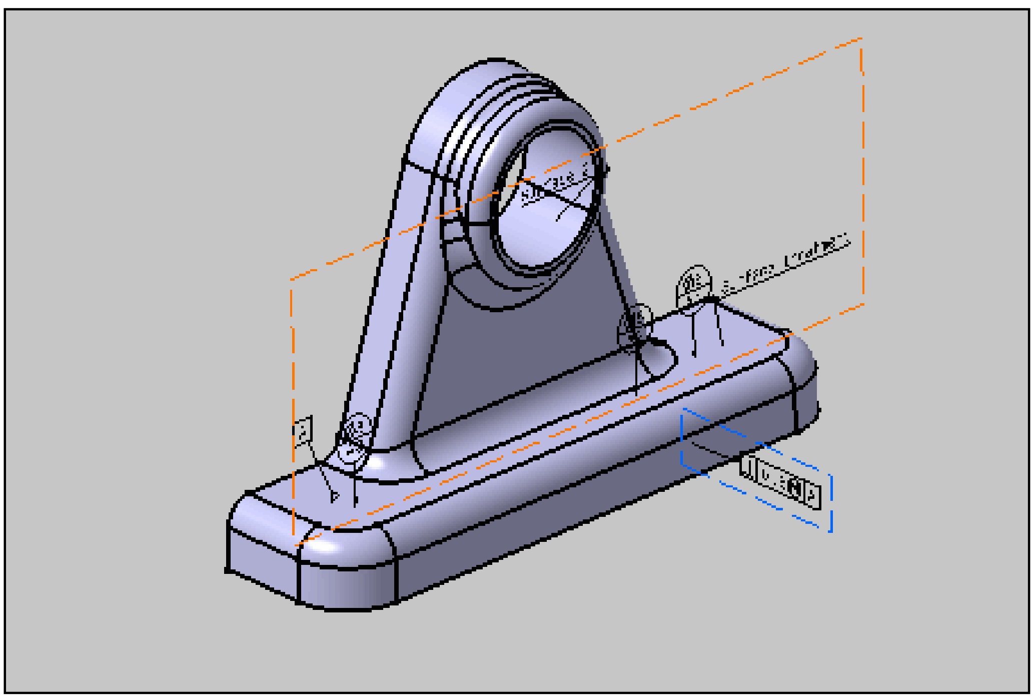


The only datum you created in this tutorial is consequently displayed attached to the geometry.



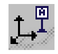
5. Click **Cancel** to cancel the operation and perform the rest of the scenario.

All annotations are visible again.

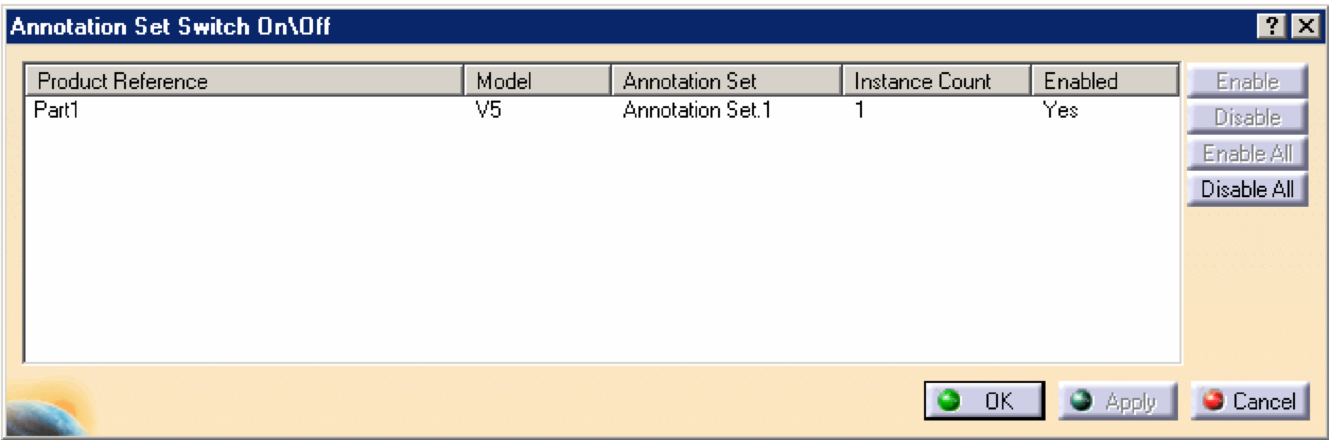


Disabling 3D Annotations

This task shows you how to disable the annotations you created.

1. Click the **List Annotation Set Switch On/Switch Off** icon: 

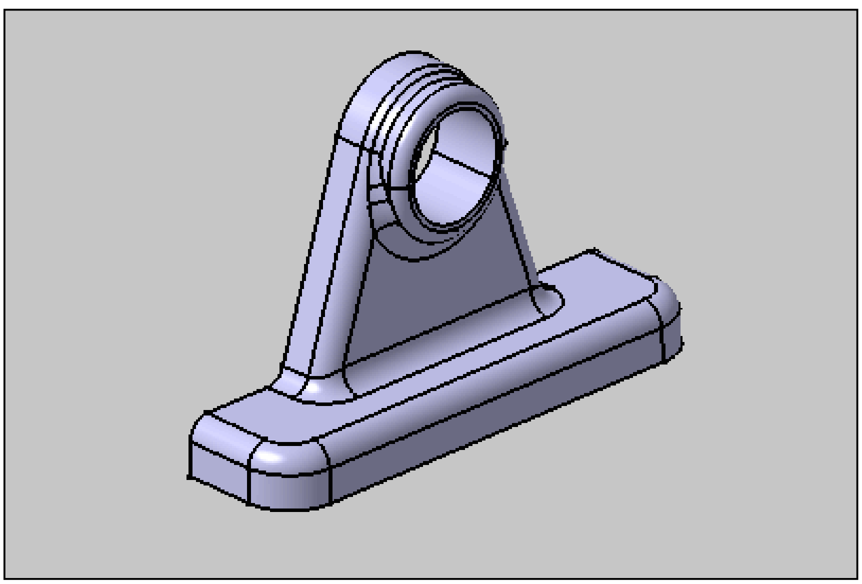
   The **Annotation Set Switch On/Off** dialog box is displayed.



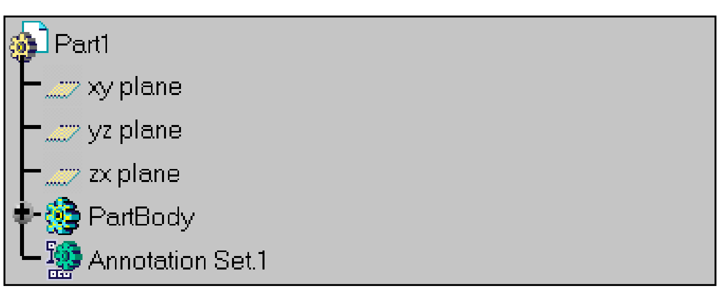
2. Click **Part1** then the **Disable All** button.

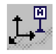
3. Click **OK** in the dialog box.

   The annotations are disabled in the geometry.



The specification tree no longer displays them.



4. Click the **List Annotation Set Switch On/Switch Off** icon  again to restore the previous state.

5. Click **Part1** then the **Enable All** button.

Accessing the Set Properties

This task shows you how to access the set properties and edit the set name.

1. Select **Annotation Set.1** in the specification tree.

2. Right-click and select the **Properties** contextual command.

3. Click **Tolerancing & Annotations** tab.

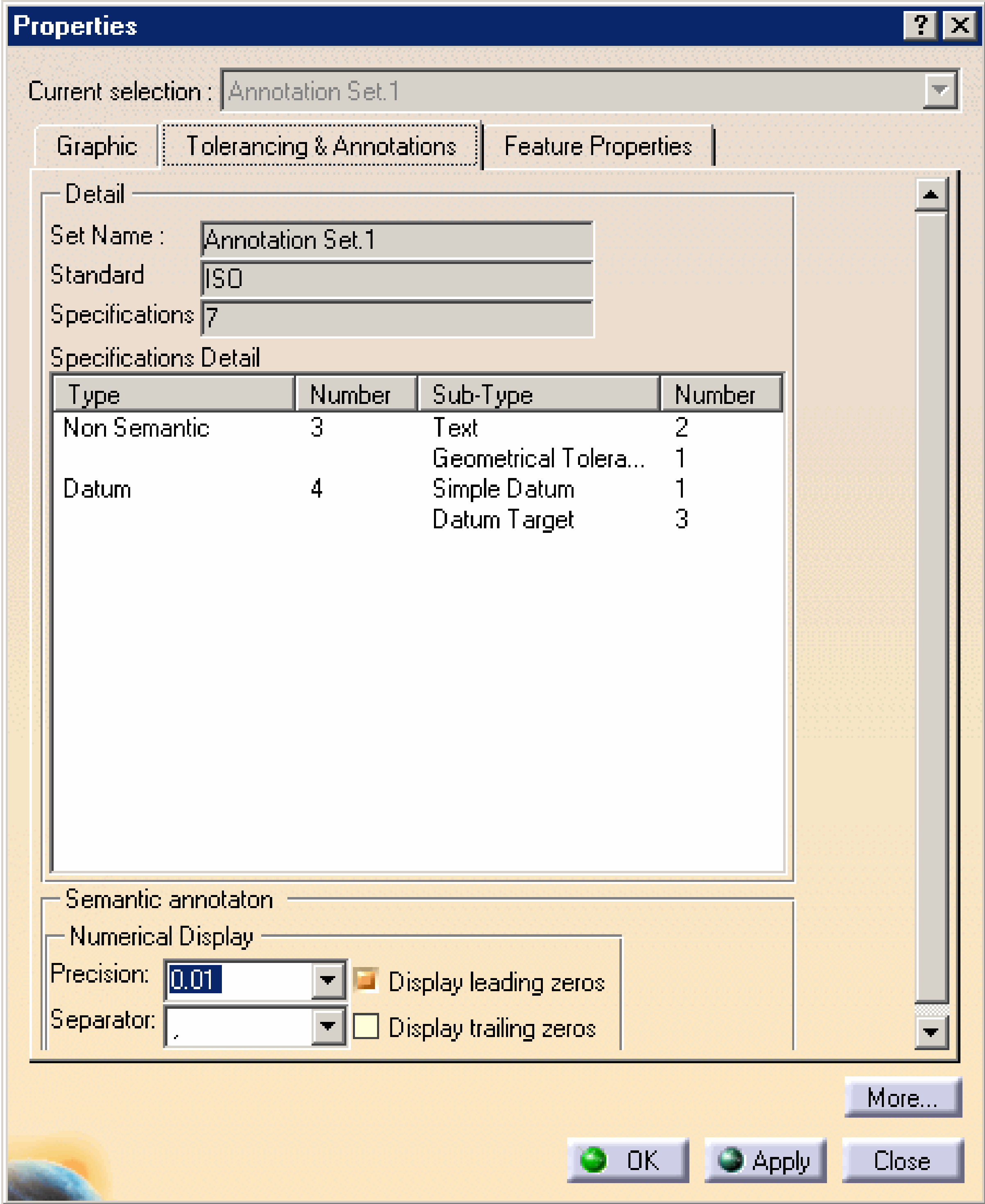
The dialog box that appears displays information about the set, namely:

The selected set name: as displayed in the specification tree

The standard used: ANSI

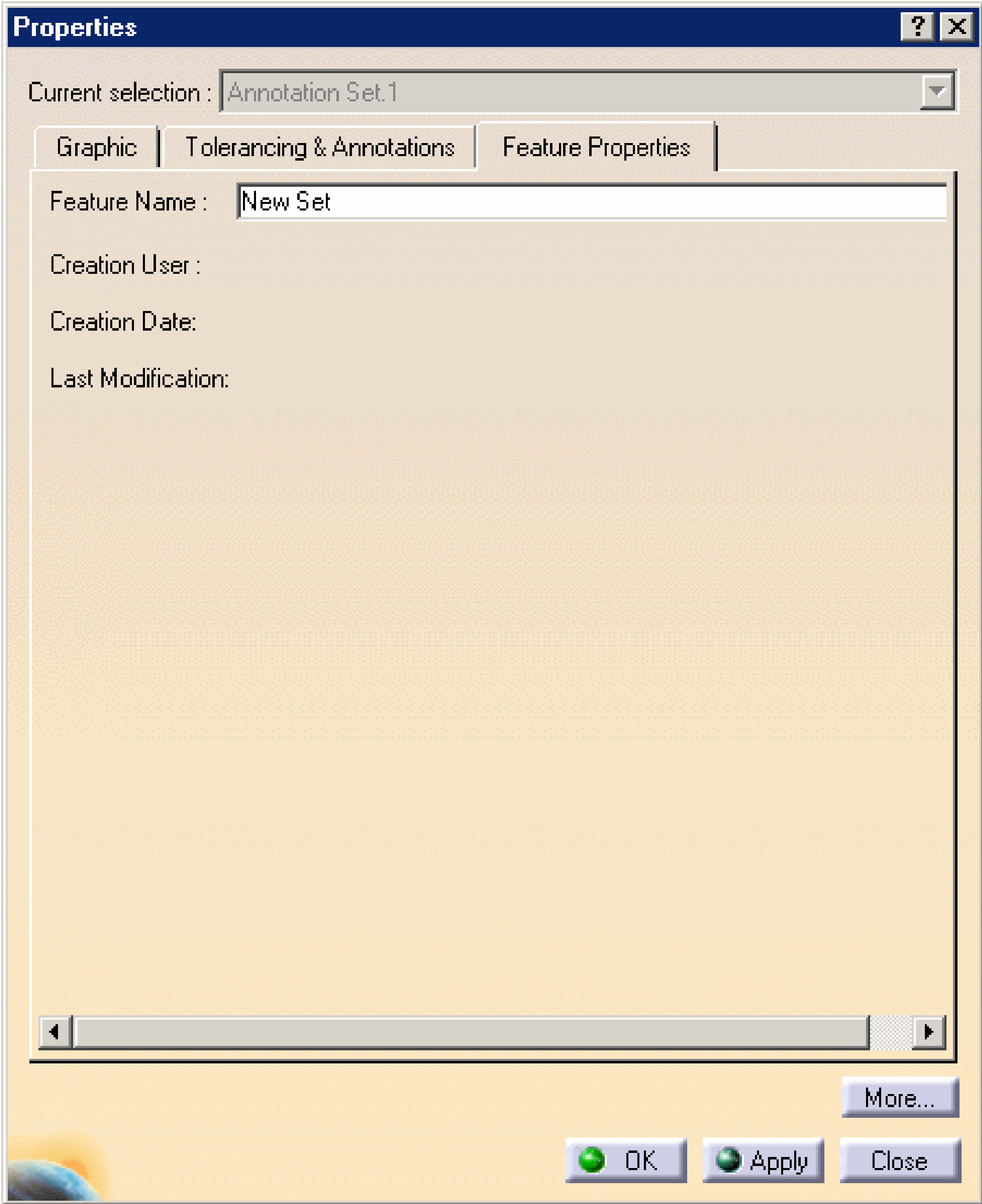
The number of specifications: you have created seven specifications.

The detail of these specifications: you have created two textual annotations, three datum targets, one GD&T and one datum.



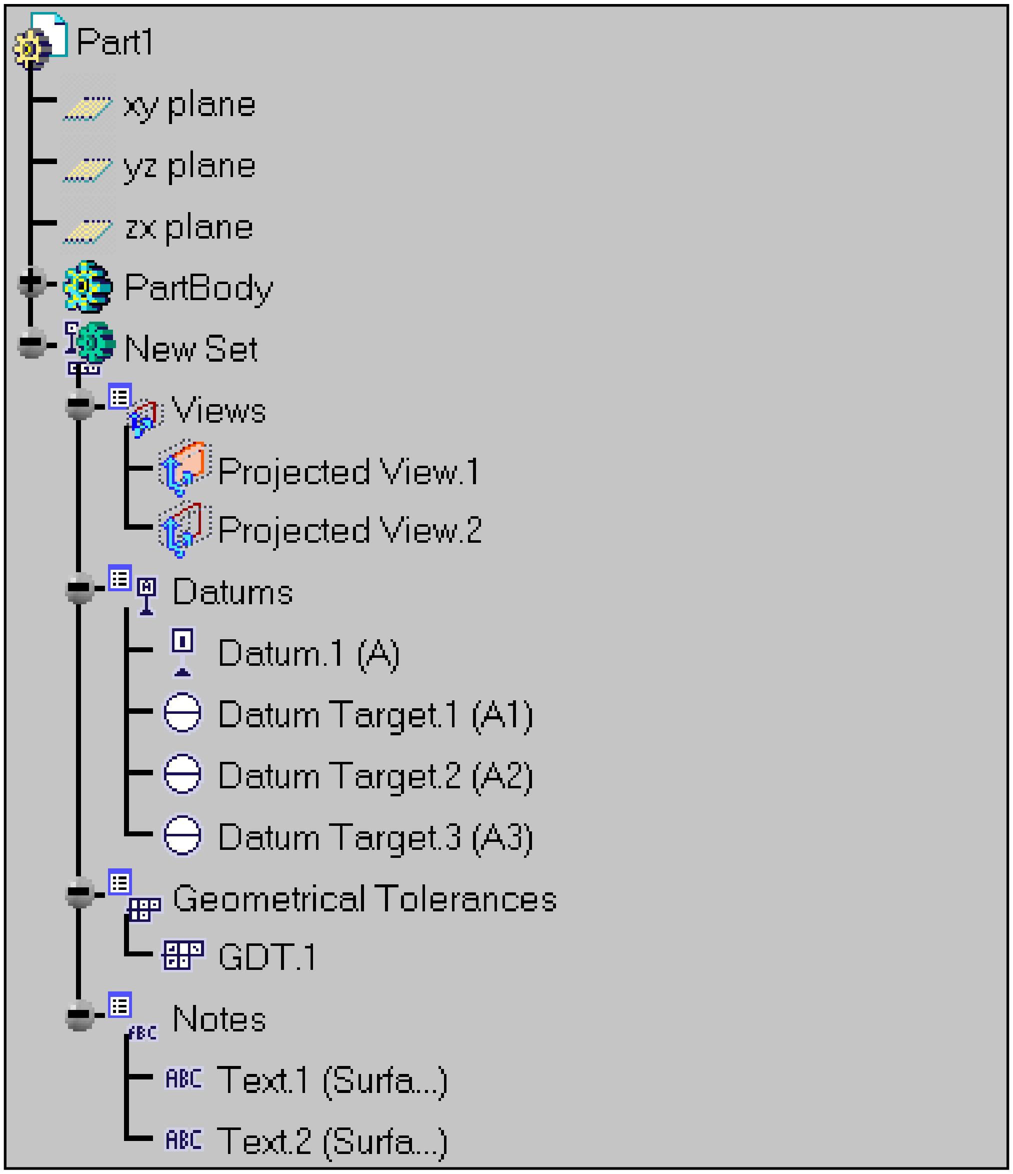
4. Click the **Feature Properties** tab (if not available, click **More**) and enter a new name in the **Feature Name** field.

For instance, enter **New Set**.



5. Click **OK** to validate and close the dialog box.

The new name is displayed in the specification tree.


Part1
xy plane
yz plane
zx plane
PartBody
New Set
Views
Projected View.1
Projected View.2
Datums
Datum.1 (A)
Datum Target.1 (A1)
Datum Target.2 (A2)
Datum Target.3 (A3)
Geometrical Tolerances
GDT.1
Notes
Text.1 (Surfa...)
Text.2 (Surfa...)

User Tasks

3D Functional Tolerancing & Annotation allows you to perform the following tasks:

Choosing a Standard

This task shows you how to set the standard you need for tolerancing your part or assembly.

You must choose a standard before creating the first annotation in a document. See also Standards.

1. Select the **Tools -> Options** command.

   The **Options** dialog box is displayed

2. Click **Mechanical Design** then **Functional Tolerancing** in the left-hand column.

   The Default **standard at creation** option provides five conventional standards:

   - ASME (American Society for Mechanical Engineers)
   - ASME 3D (American Society for Mechanical Engineers)
   - ANSI (American National Standards Institute)
   - ISO (International Organization for Standardization)
   - JIS (Japanese Industrial Standard)

   See Tolerancing setting for further detail.

3. Click **OK** to validate and close the dialog box.

Note that this choice of standard must be expressed prior to specifying any tolerance. After any creation in the workbench, the standard may be modified but the corresponding syntax and semantic variation will not be taken into consideration.

Using the Tolerancing Advisor

Introducing the Tolerancing Advisor: get started with the Tolerancing Advisor.

Dimensioning and Tolerancing Threads using the Tolerancing Advisor: see how the Tolerancing Advisor lets you create dimensions and tolerances for threads.

Introducing the Tolerancing Advisor

This task introduces the Tolerancing Advisor.

The Tolerancing Advisor lets you create allowed annotations according to the selected geometrical element or existing annotation.

Allowed annotations are:

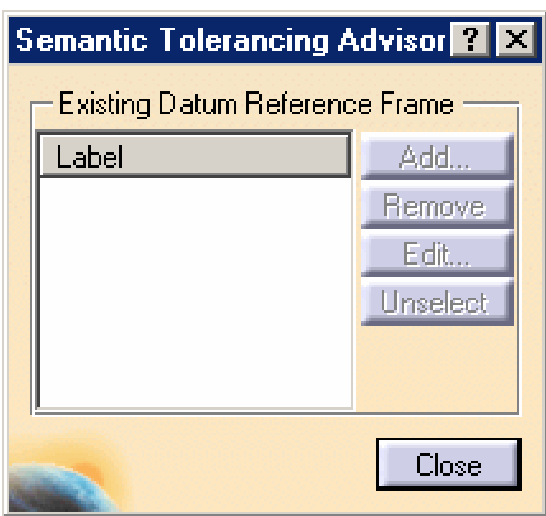
- Semantic and non semantic annotations (Text, Roughness, Flag note). See Tolerancing Settings.
- Datum.
- Datum target.
- Datum reference frame.

Open the Tolerancing_Annotations_01 CATPart document:

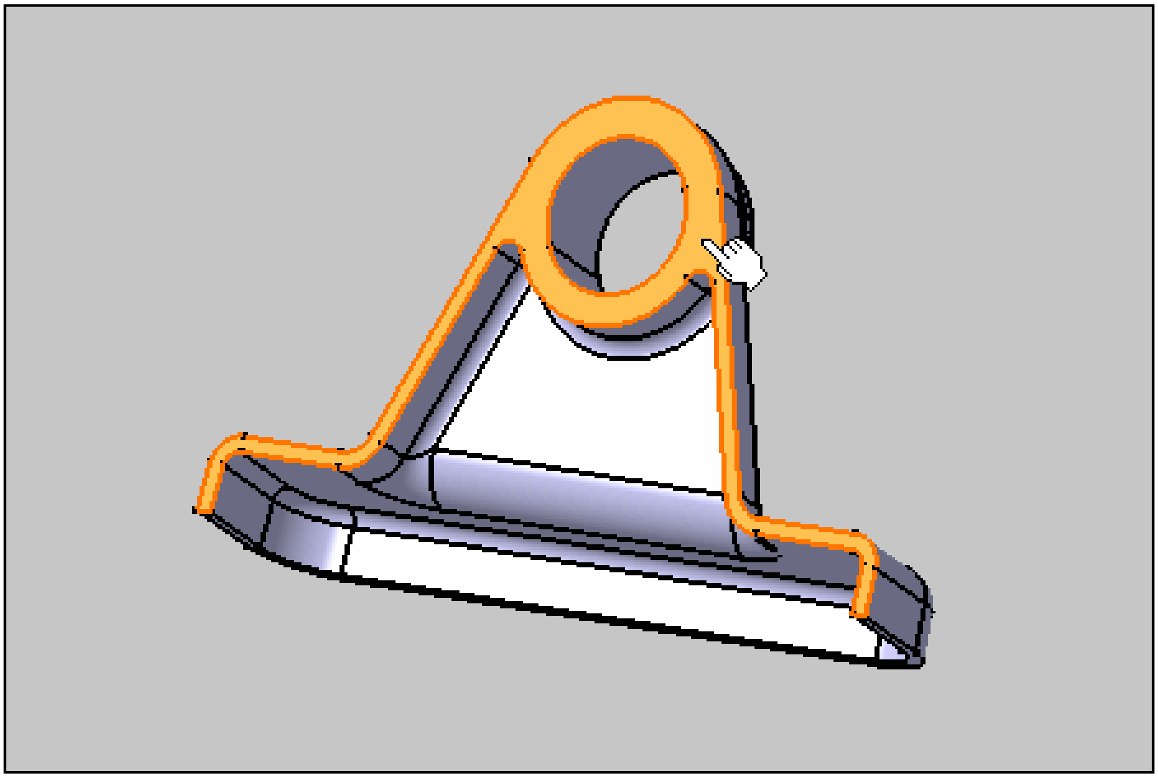
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Click the **Tolerancing Advisor** icon: 

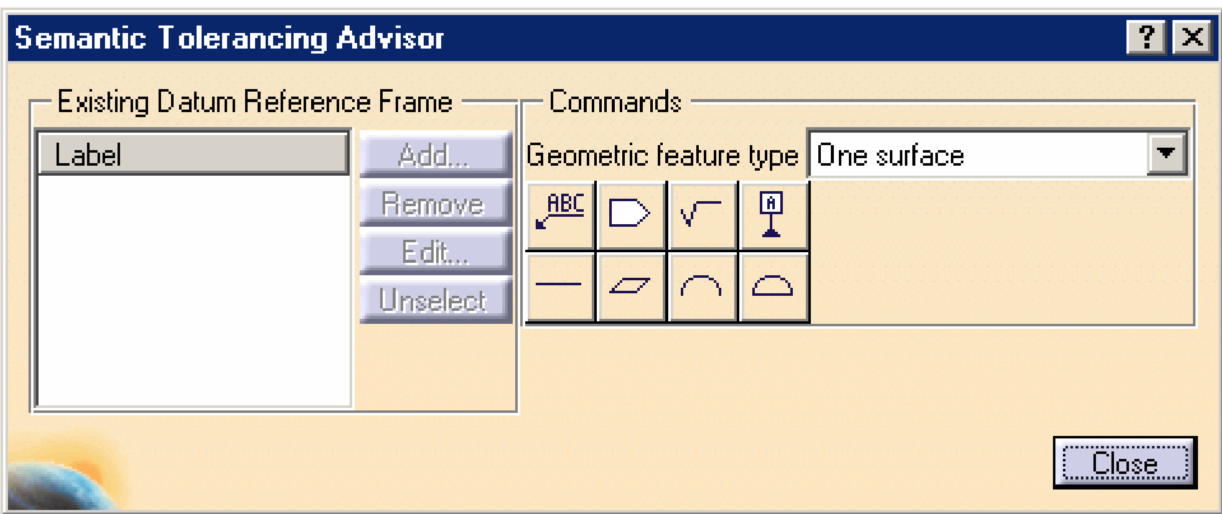
   The **Semantic Tolerancing Advisor** dialog box appears.
   This is the minimal appearance for this dialog box because no geometrical element or annotation has been selected and no datum reference frame has been created yet.



2. Select the surface as shown on the part.



The **Semantic Tolerancing Advisor** dialog box is updated according to the selected surface.

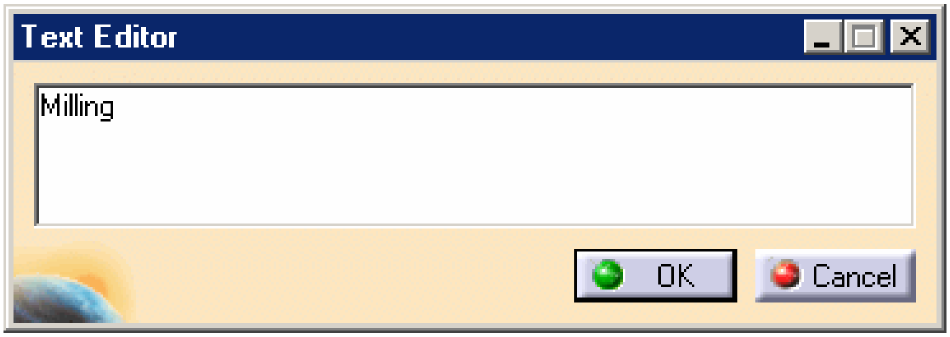


The **Commands** frame contains all the semantic annotations that will be created in relation with the selected element and the geometrical feature type.

The **Commands** frame contains a combo list for all capabilities applying for the selection.

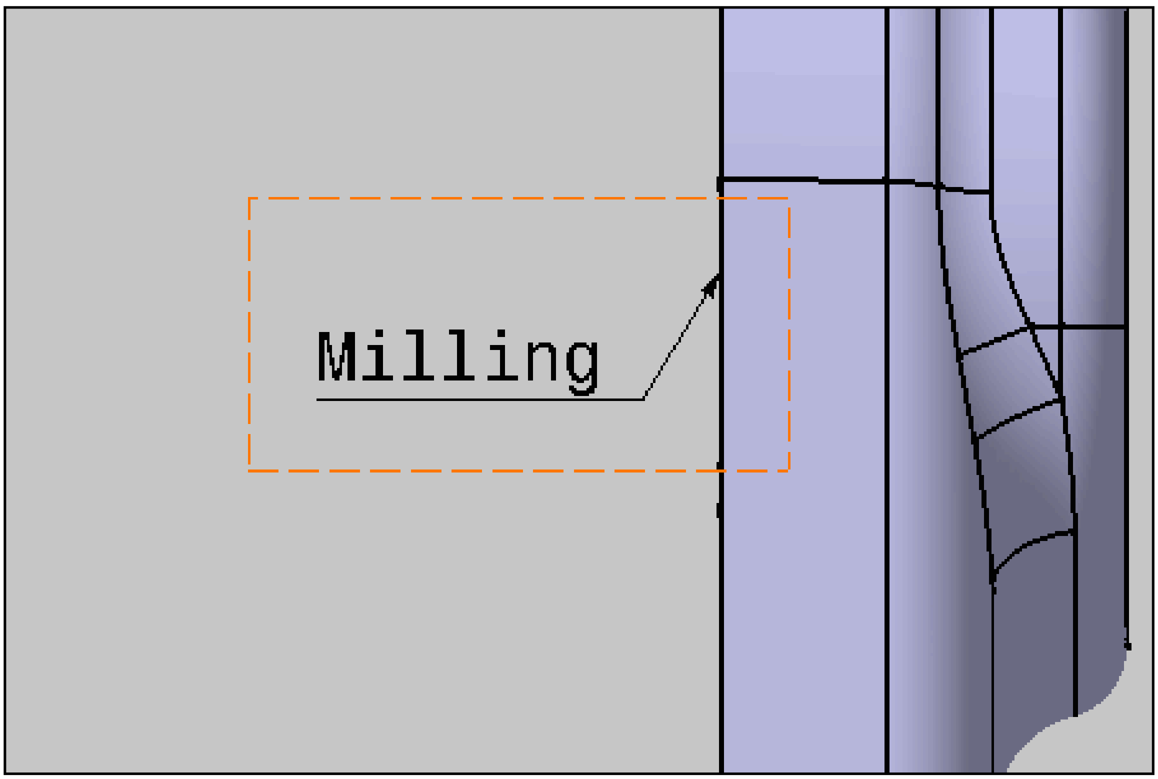
3. Click the **Text with Leader** icon (One surface): 

4. Enter Milling in the **Text Editor** dialog box when it appears.

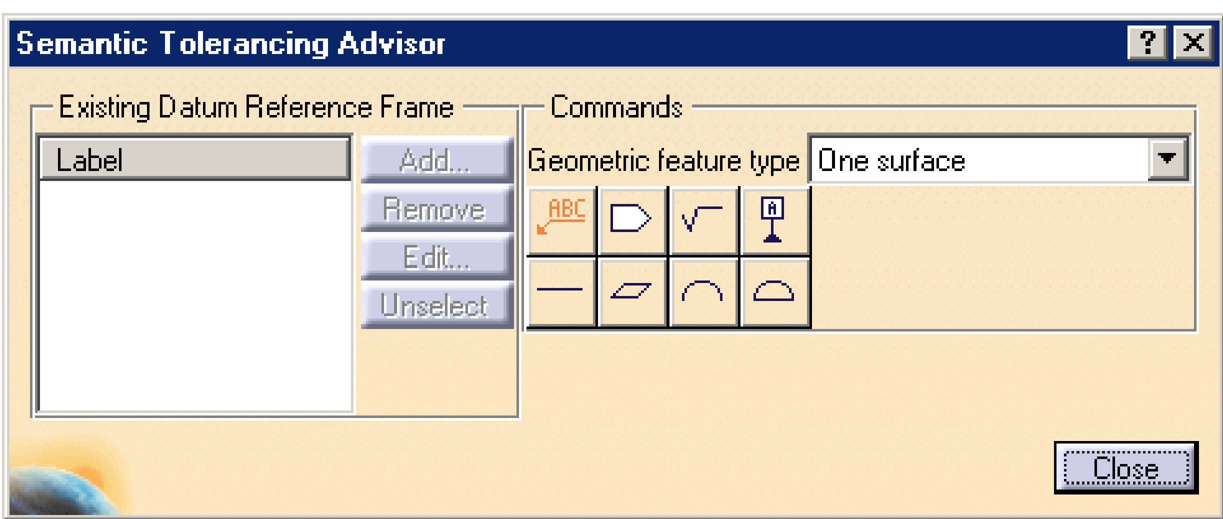


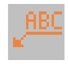
5. Click **OK** in the dialog box.

The annotation text is created.



The **Semantic Tolerancing Advisor** dialog box is updated.



The **Text with Leader** icon is orange-colored: 
This color inform you that an annotation has been created; you can still create other annotations.

6. Click **Close** in **Semantic Tolerancing Advisor** dialog box.

Dimensioning and Tolerancing Threads

This task shows how to create dimensions and tolerances for threads using the Tolerancing Advisor.

For a general introduction of the Tolerancing Advisor, refer to Introducing the Tolerancing Advisor.

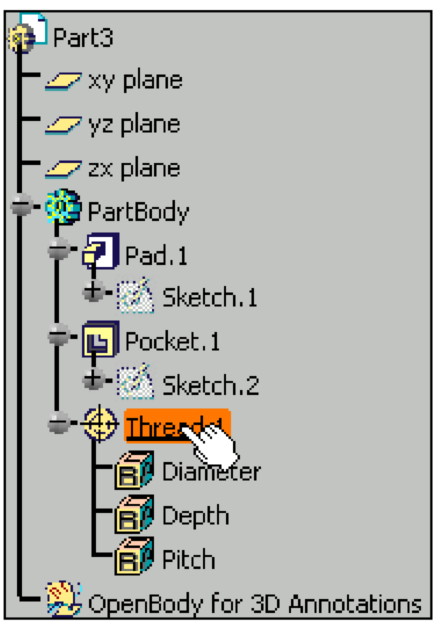
Open the Tolerancing_Annotations_12 CATPart document.

1. Click the **Thread Representation Creation** icon: 

   The **Thread Representation Creation** dialog box is displayed.

2. Select **Thread.1** in the specification tree.

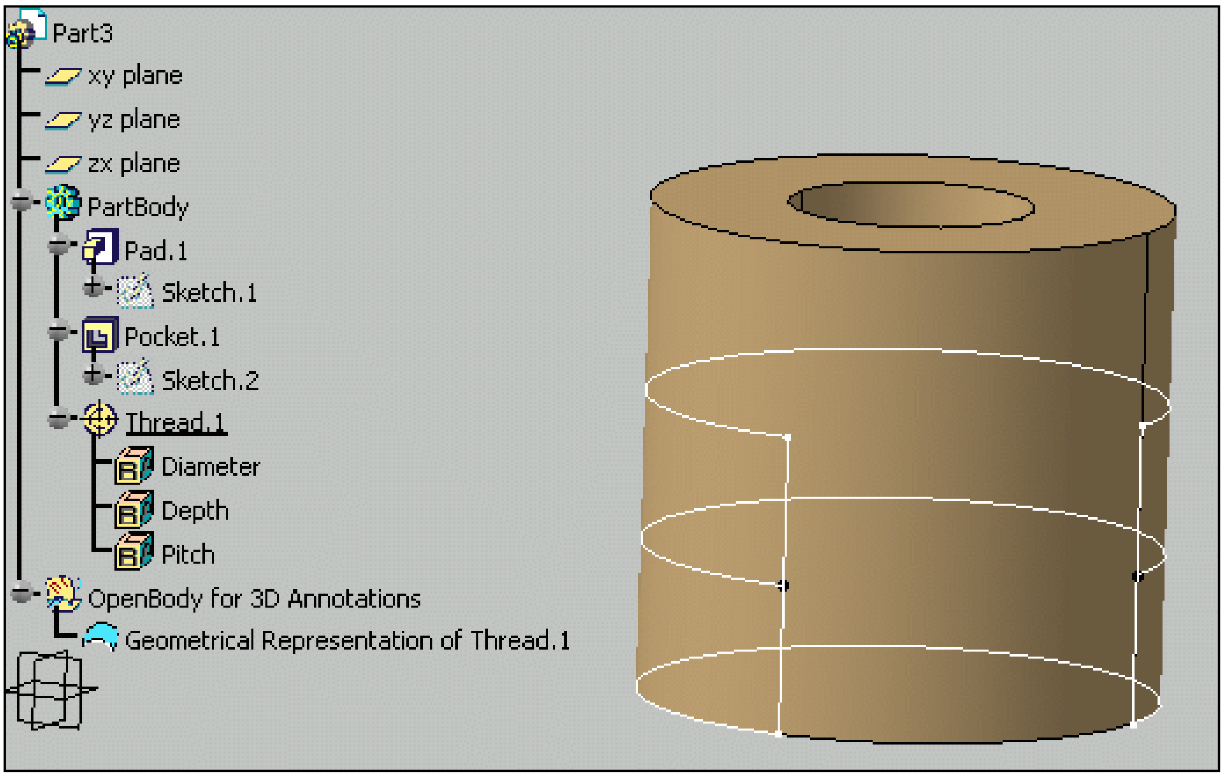
   

   The **Thread Representation Creation** dialog box is updated to indicate that the thread representation will be created for the selected thread.

In the case of numerous threads, selecting the **All threads** option lets you create the thread representations of all of them.

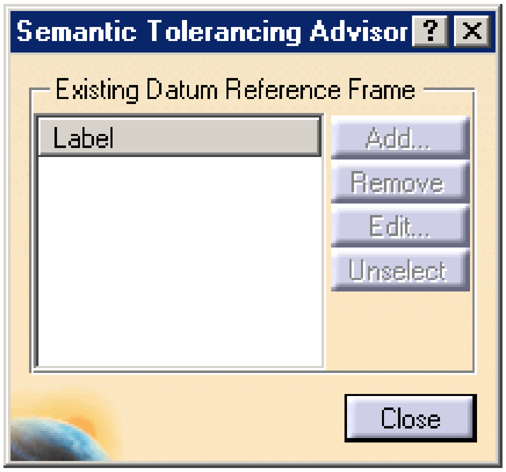
3. Click **OK** to validate and exit the dialog box.

The geometrical representation of the thread is displayed in the geometry, and an item is created in the specification tree.

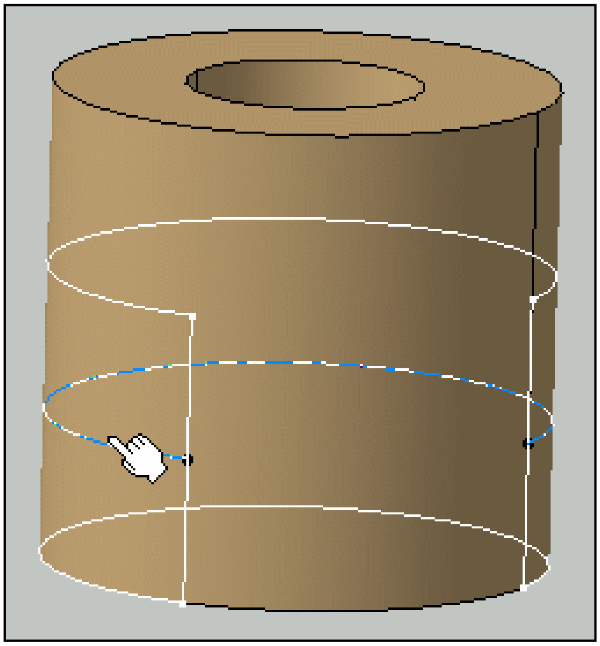


4. Click the **Tolerancing Advisor** icon: 

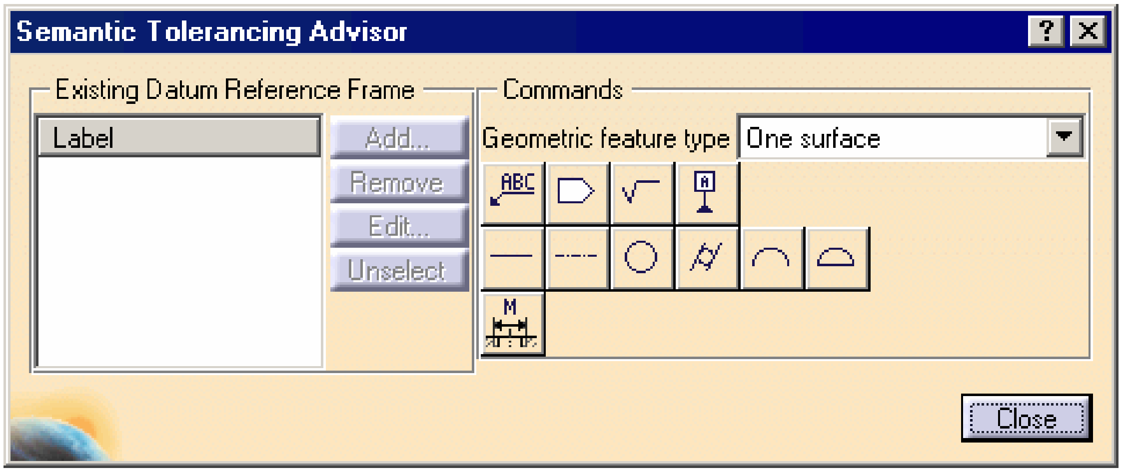
The **Semantic Tolerancing Advisor** dialog box is displayed.



5. Select the median 3/4 circle arc which symbolizes the thread helical surface.



The **Semantic Tolerancing Advisor** dialog box is updated according to the selected element.

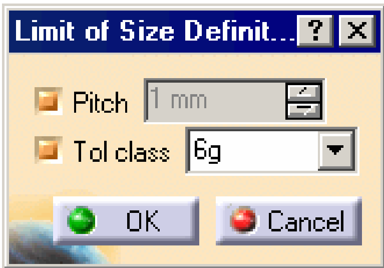


6. Click the **Diameter** icon (One surface): 

The thread diameter dimension is previewed and the **Limit of Size Definition** dialog box is displayed, offering the following options:

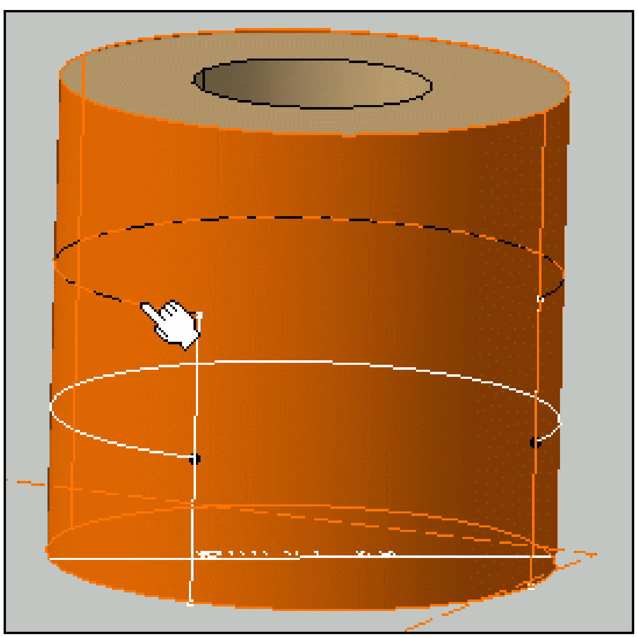
- **Pitch**: lets you display the pitch value in the thread dimension.
- **Tol class**: lets you define and display the tolerance class value in the thread dimension.

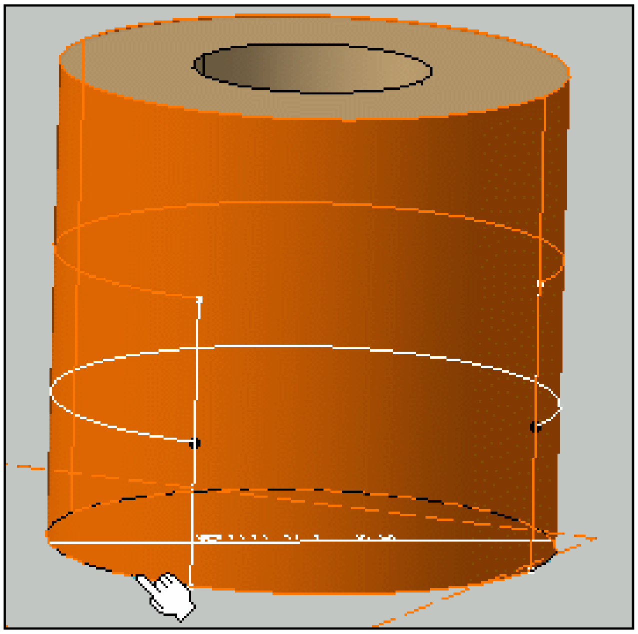
7. Select both the **Pitch** and the **Tol class** options and in the **Tol class** drop-down list, select **6g** as the tolerance class value.



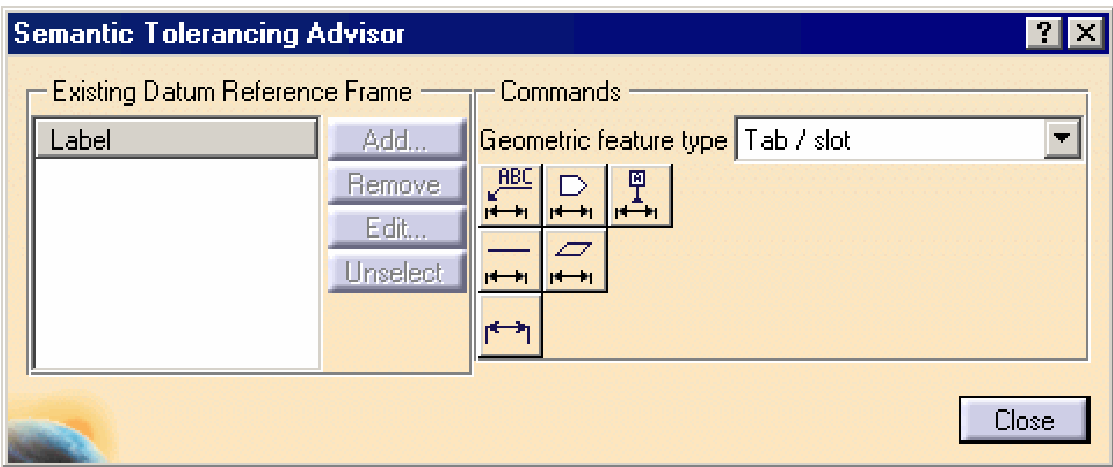
8. Click **OK** to validate. The thread diameter dimension is created.

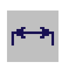
9. Back in the **Semantic Tolerancing Advisor** dialog box, multi-select (using the **Ctrl** key, for example) the 3/4 circle arcs which symbolize the thread starting and ending planes.





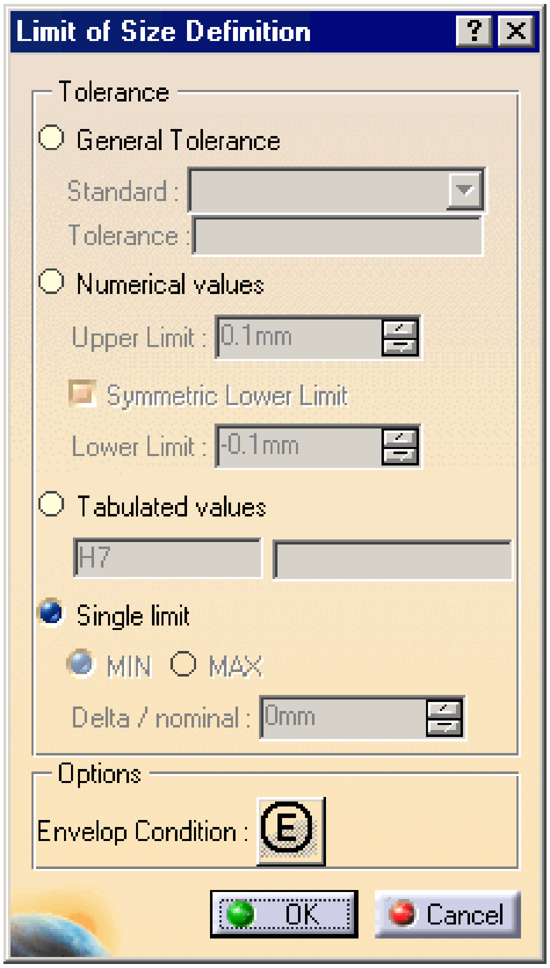
Once again, the **Semantic Tolerancing Advisor** dialog box is updated according to the selected elements.



10. Click the **Distance Creation** icon (Tab/slot): 

The thread length dimension is previewed and the **Limit of Size Definition** dialog box is displayed, offering the following options:

- **General Tolerance**: lets you define a pre-defined class of tolerance, see Tolerances for the default class setting.
- **Numerical values**: lets you define the Upper Limit and optionally the Lower Limit (provided you uncheck the Symetric Lower Limit option).
- **Tabulated values**: lets you define fitting tolerances. Refer to Normative References for more information: ISO 286, ANSI B4.2.
- **Single limit**: lets you enter a minimum or maximum tolerance value. Use the Delta / nominal field to enter a value in relation to the nominal diameter value. For example, if the nominal diameter value is 10 and if you enter 1, then the tolerance value will be 11.

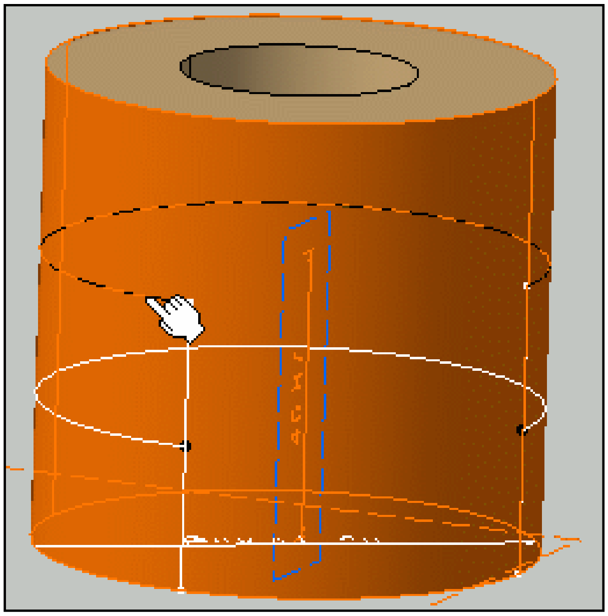


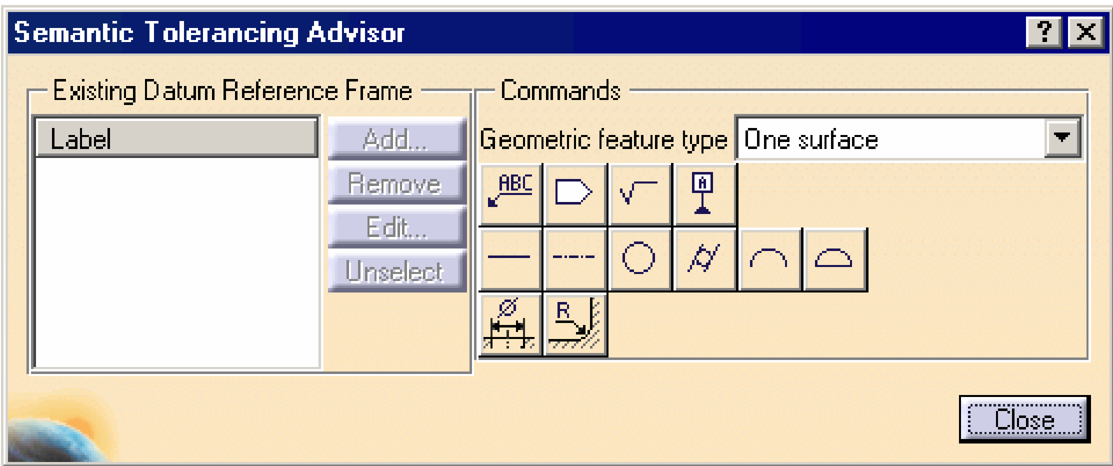
11. Select the **Tabulated values** option.

12. Click **OK** to validate.

13. When a dialog box appears, informing you that the annotation will be created on another view, click **OK**. The thread length dimension is created.

14. Back in the **Semantic Tolerancing Advisor** dialog box, select the 3/4 circle arc which symbolizes the thread starting plane.





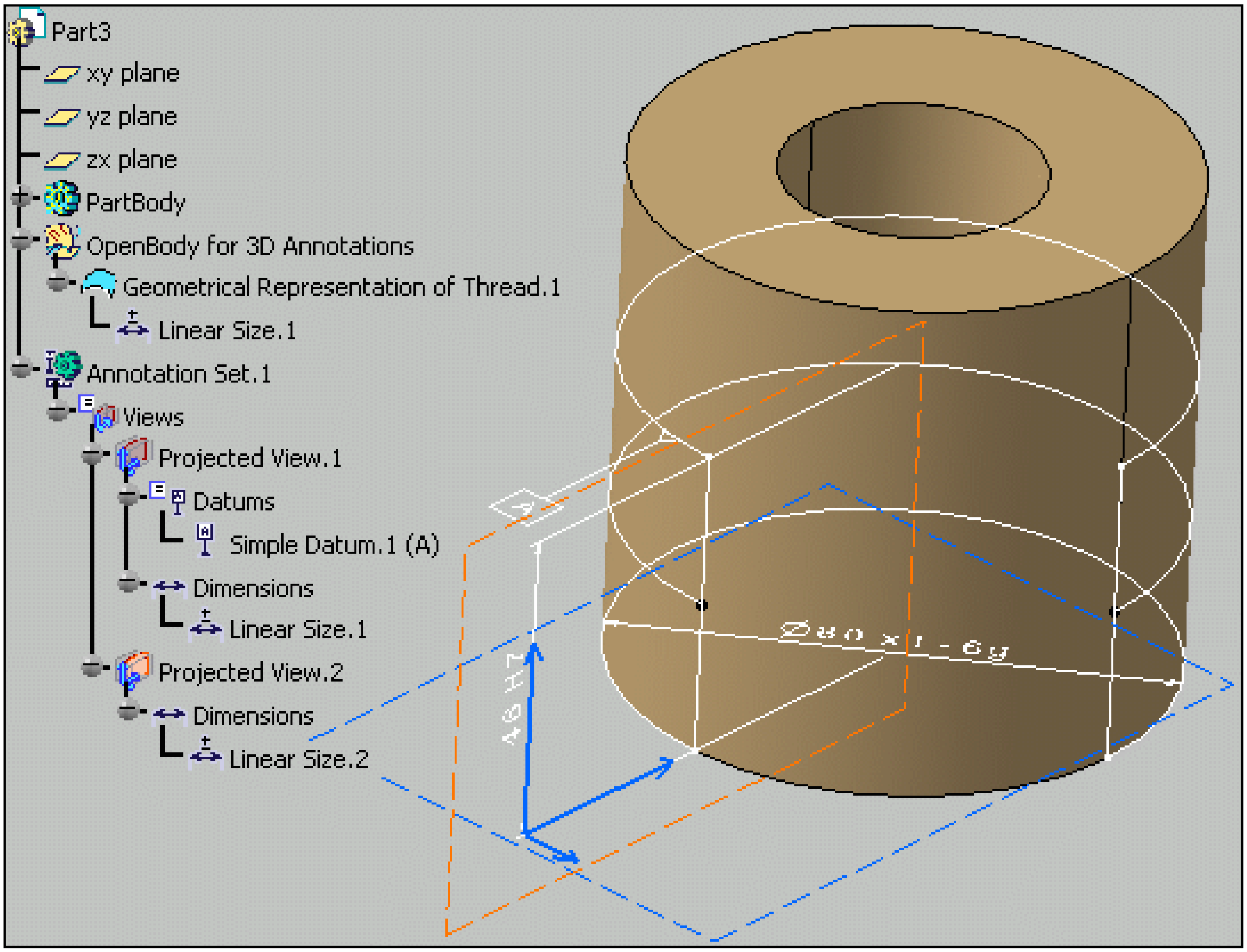
15. Click the **Semantic Datum** icon (One surface): 

The **Datum Definition** dialog box is displayed.

16. Leave the options as is, and click **OK** to validate. The datum feature is created.

Note that the 3/4 circle arcs which symbolize the thread starting and ending planes are recognized as planar surfaces, which means that you can tolerance them just like any other planar surface. At this stage, you create other tolerances (such as flatness for example) using the icons available in the **Semantic Tolerancing Advisor** dialog box.

17. Click **Close** in the **Semantic Tolerancing Advisor** dialog box. The thread dimensions and tolerances are displayed in the geometry as well as in the specification tree.



Part3
xy plane
yz plane
zx plane
PartBody
OpenBody for 3D Annotations
Geometrical Representation of Thread.1
Linear Size.1
Annotation Set.1
Views
Projected View.1
Datums
Simple Datum.1 (A)
Dimensions
Linear Size.1
Projected View.2
Dimensions
Linear Size.2
A
∅40 x1 - 6g
46H7

Tolerancing Body in White

The tasks described in the following scenario are meant to be performed step by step.

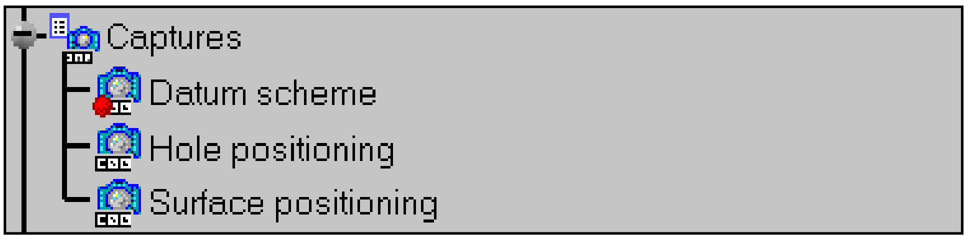
Creating a Datum and Datum Targets

This task shows you how to create a datum and datum targets on body in white surfaces.

Open the Tolerancing_Annotations_06 CATPart document:

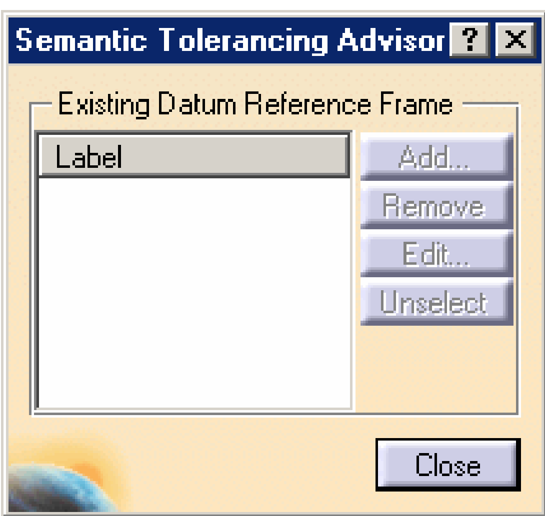
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Right-click the **Datum scheme** capture and select **Set Current** form the contextual menu: all created annotations will be added to this capture as long as it is current.

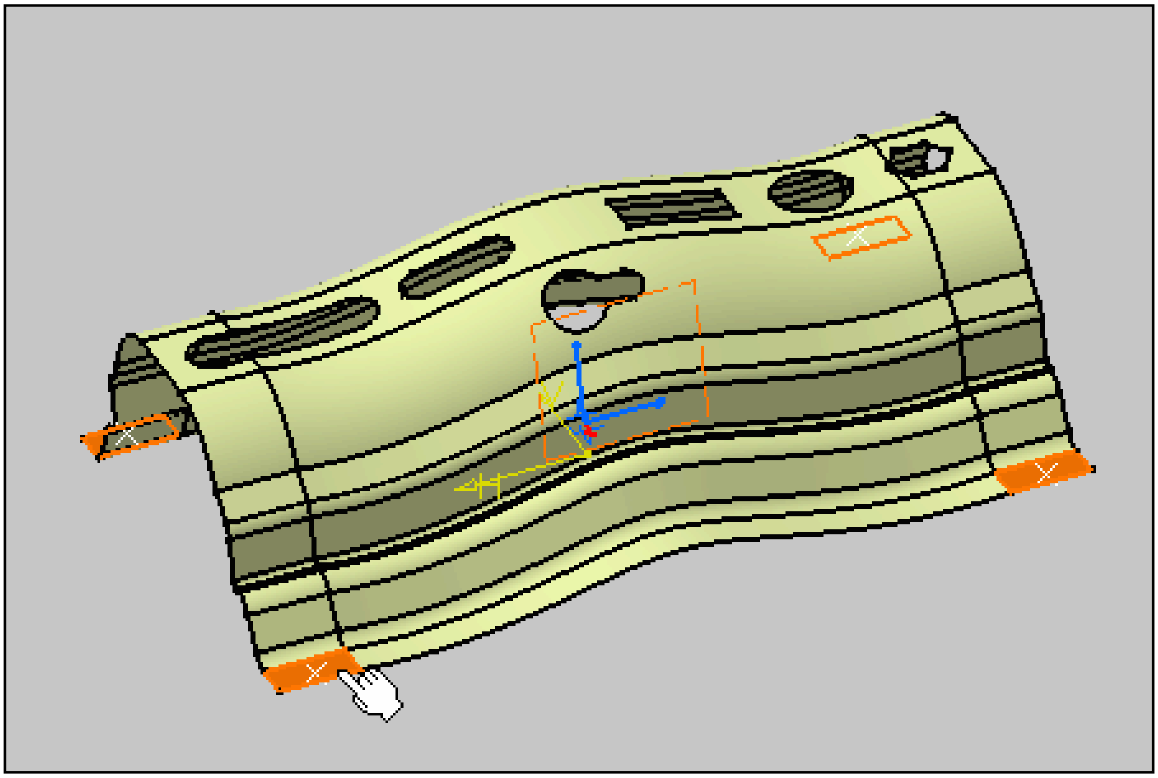


2. Click the **Tolerancing Advisor** icon: 

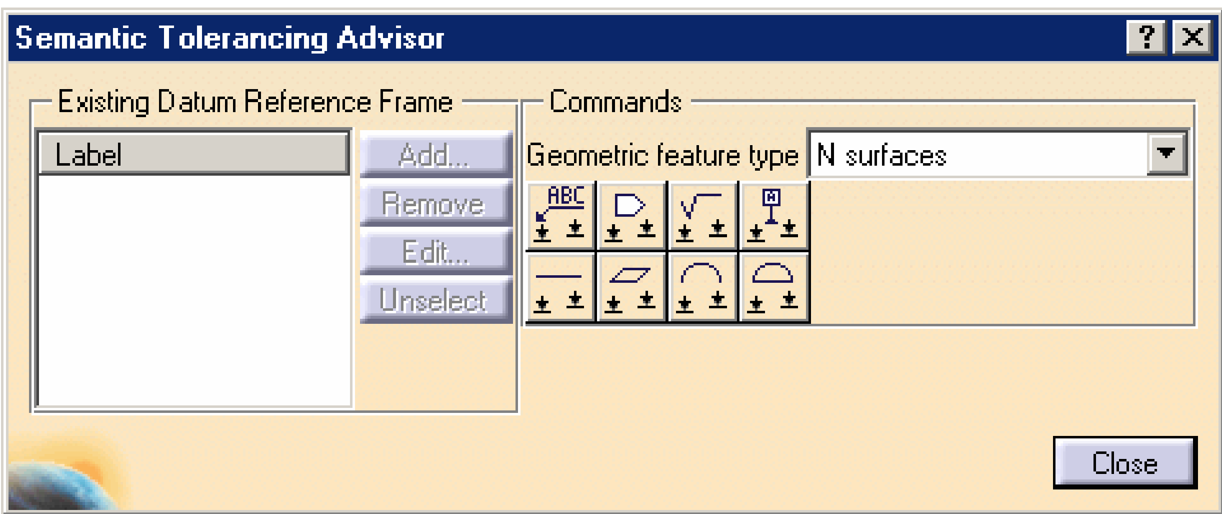
The **Semantic Tolerancing Advisor** dialog box appears.

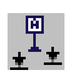


3. Select the four surfaces as shown on the part. The last selected surface will support the datum.

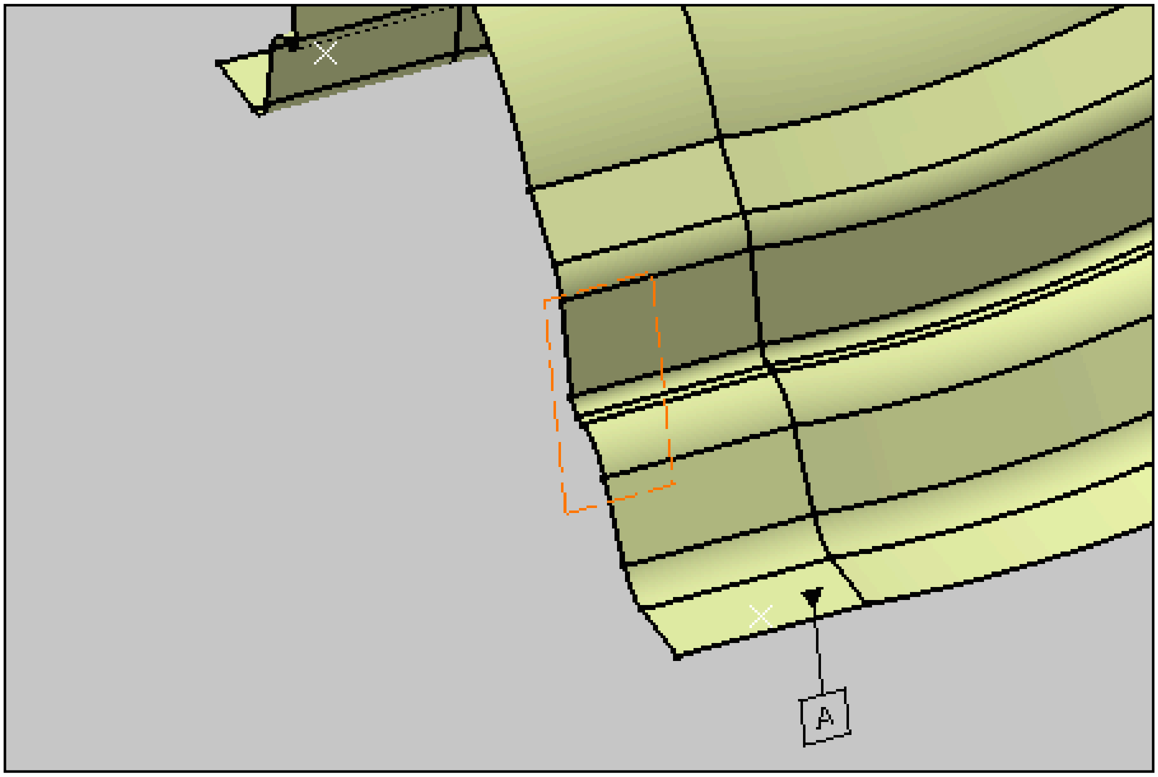


The **Semantic Tolerancing Advisor** dialog box appears.
The buttons and options available in the dialog box depend on your selection.

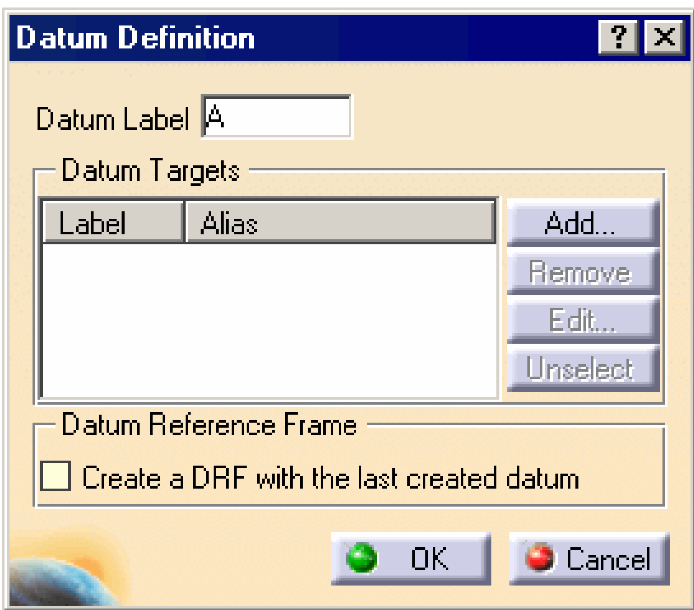


4. Click the **Semantic Datum** icon (N surfaces): 

The datum is created.

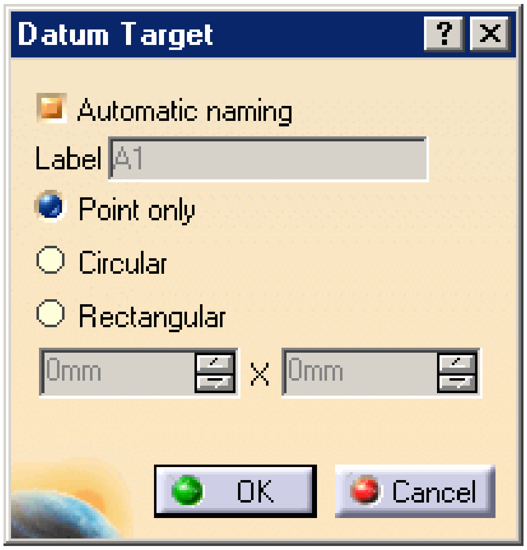


The **Datum Definition** dialog box appears.

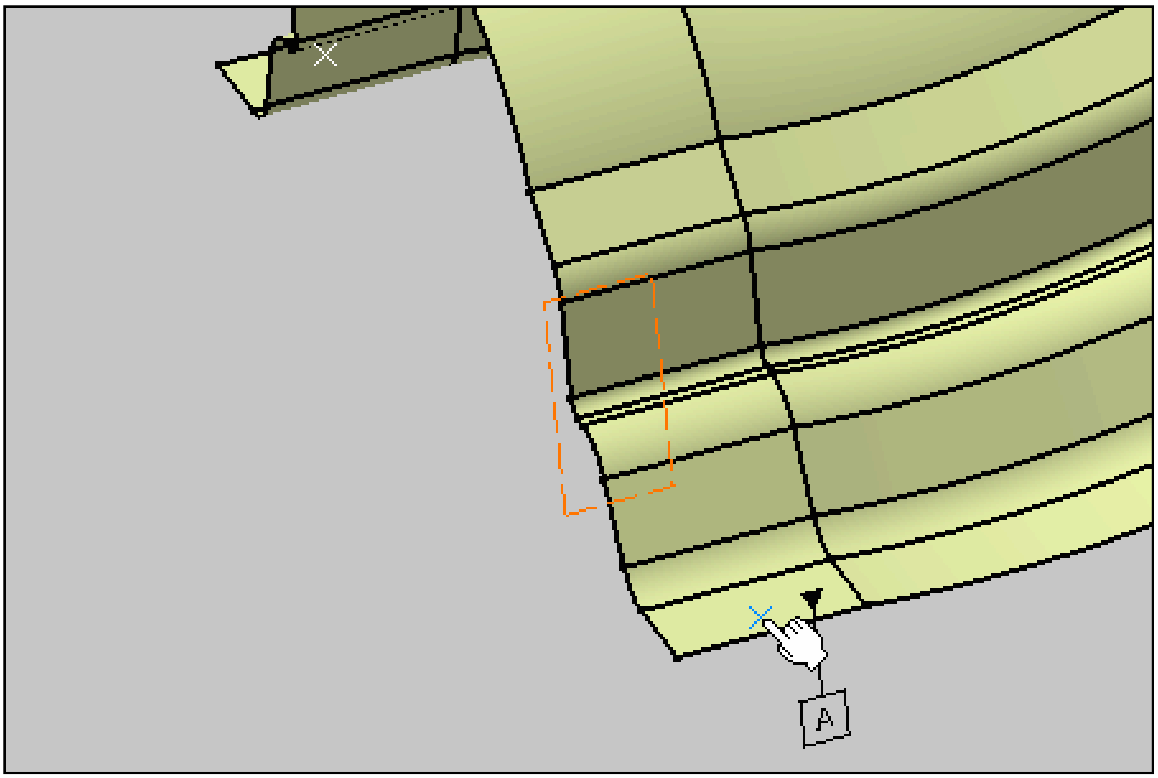


5. Click **Add** in the **Datum Definition** dialog box.

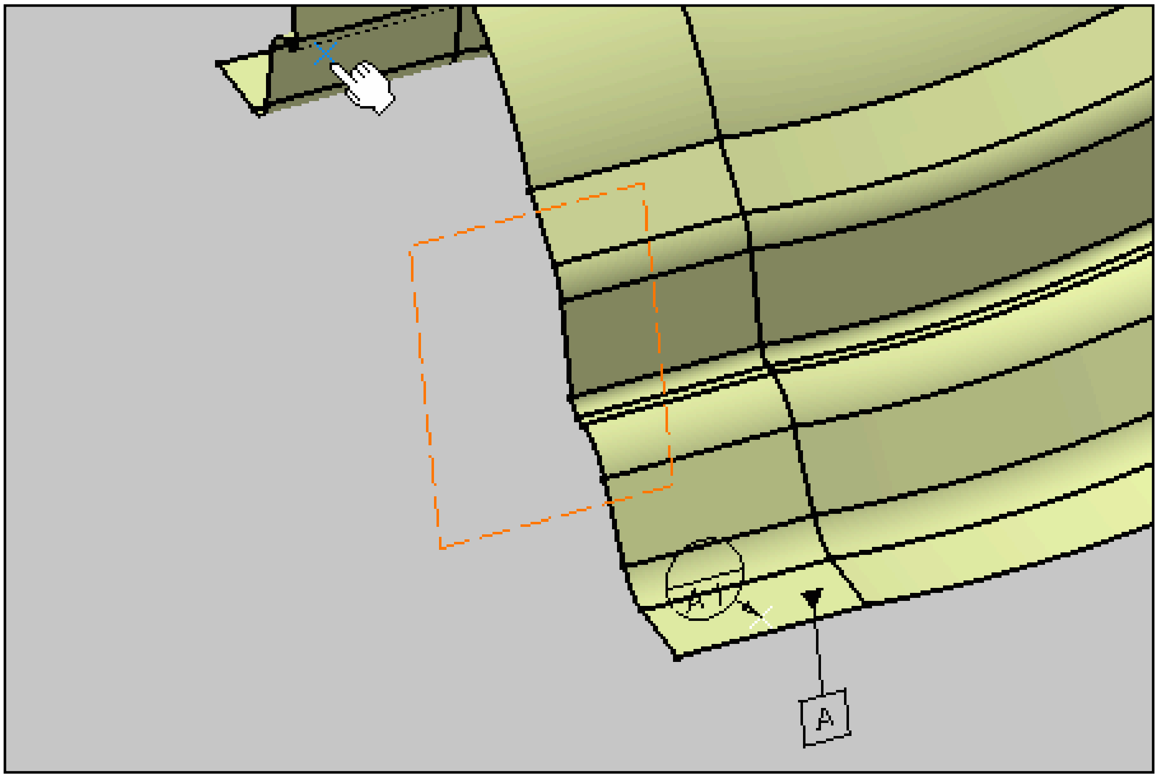
The **Datum Target** dialog box appears. Keep the options as is.

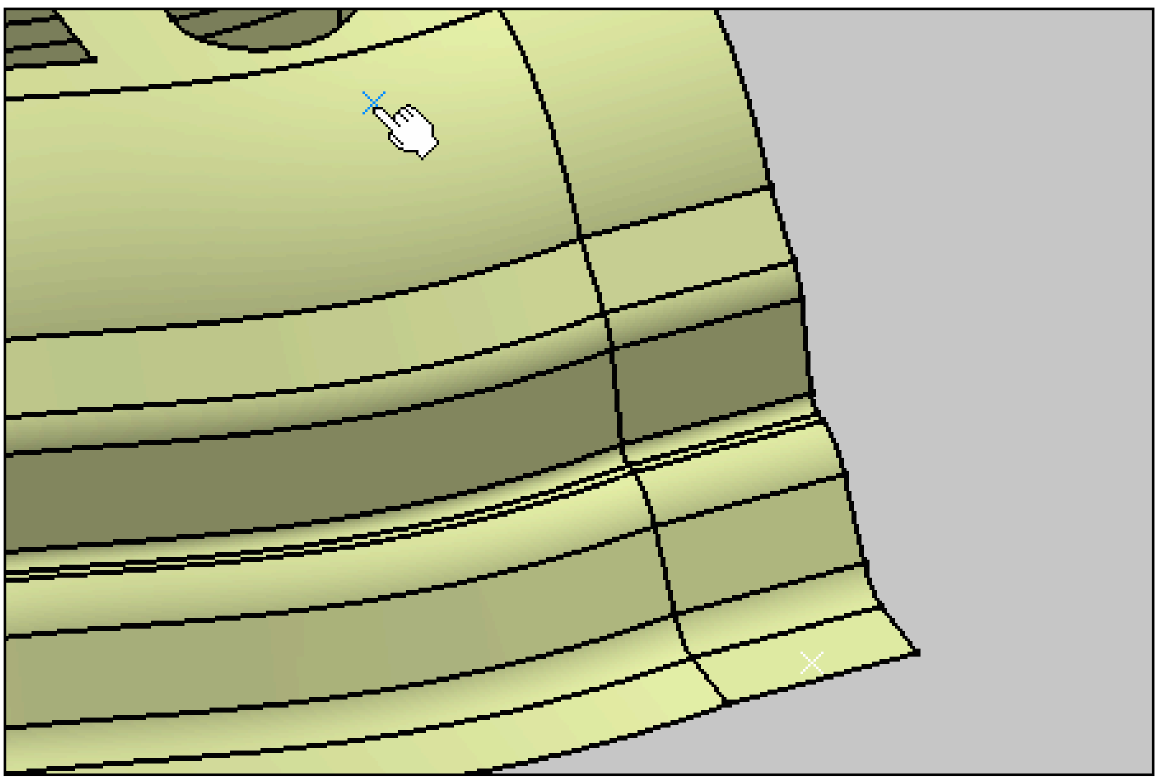


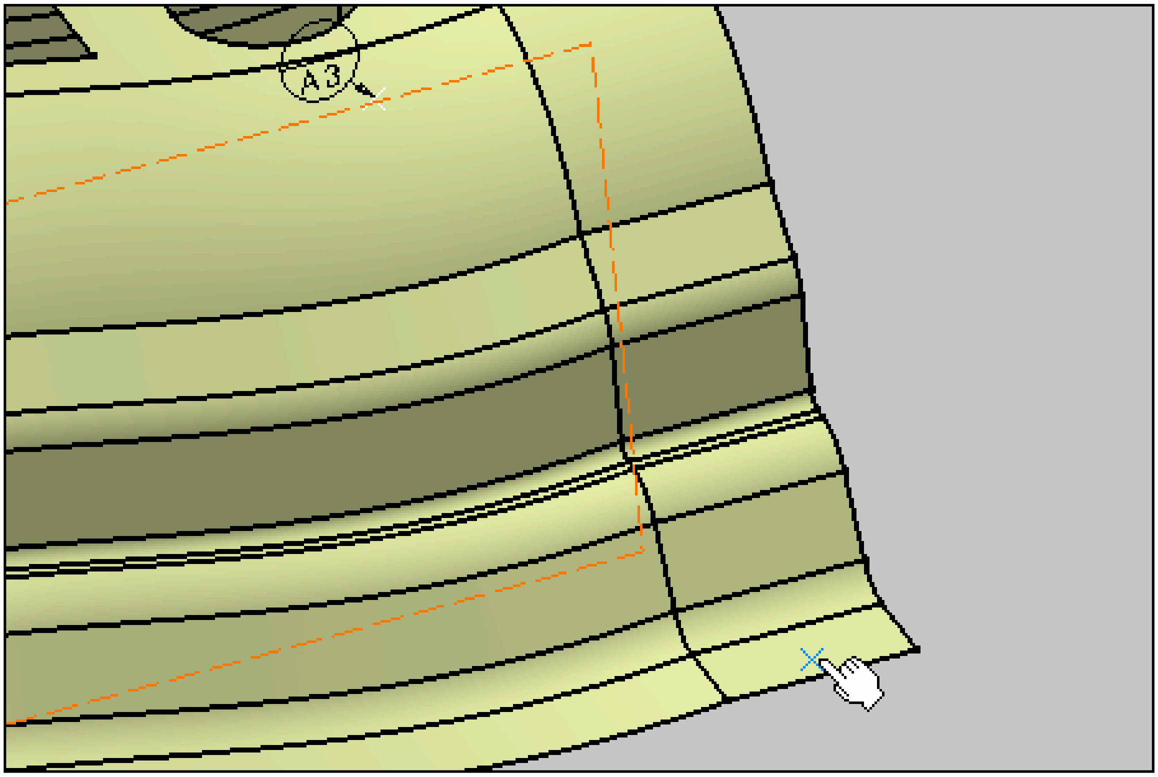
6. Select the target points as shown on the part.



You do not have to click **OK** in the **Datum Target** dialog box, selecting another point automatically validates the datum target creation.

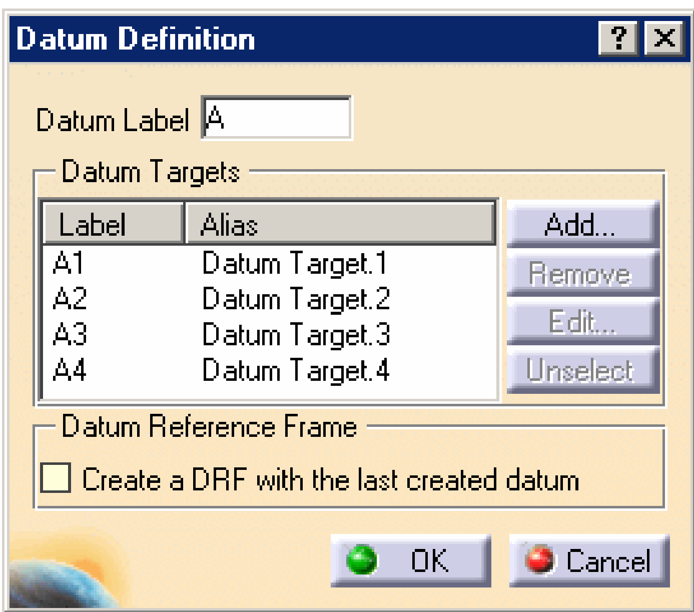

A1
A





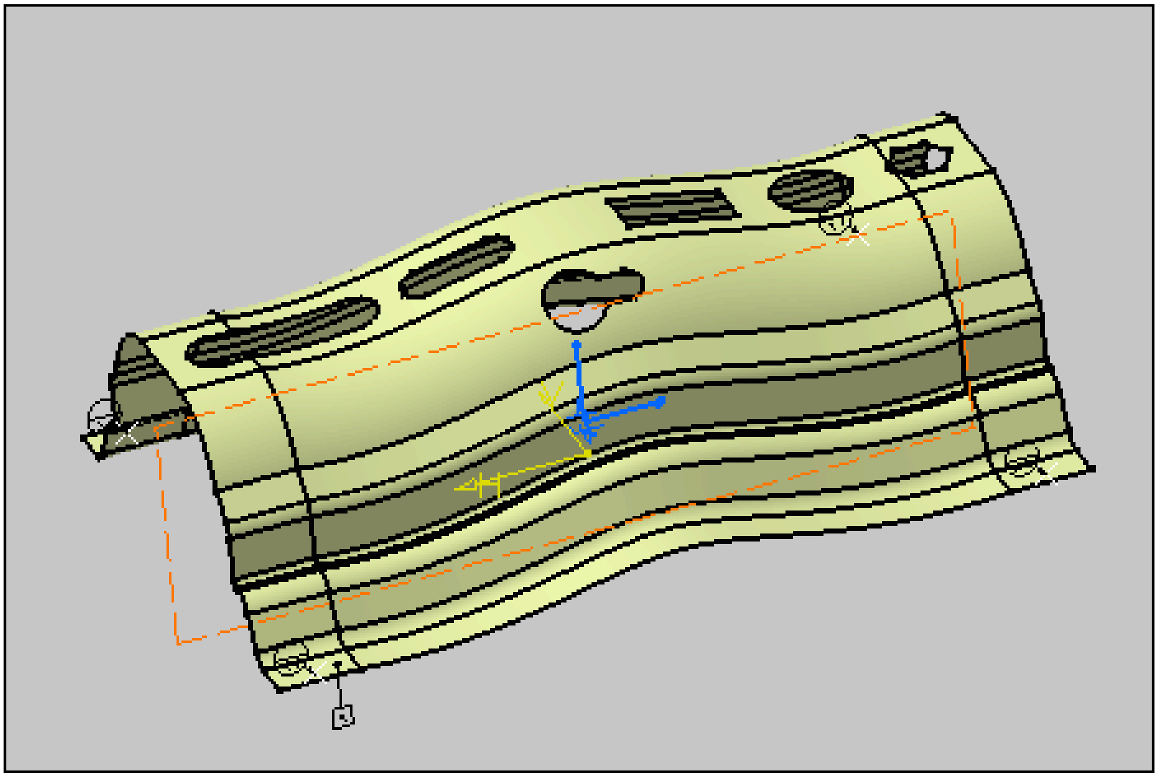
7. Click **OK** in the last **Datum Target** dialog box to end the datum target creation.

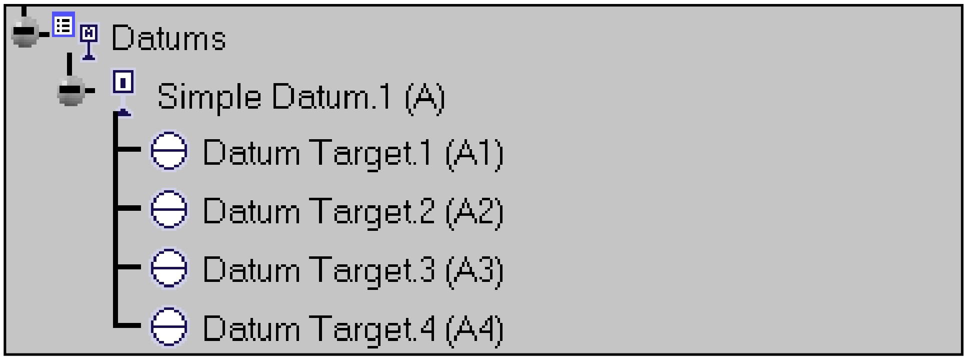
The **Datum Definition** dialog box is updated.



8. Click **OK** in the **Datum Definition** dialog box.

The datum and datum targets are created in the geometry and specification tree.





9. Do not close the **Semantic Tolerancing Advisor** dialog box to perform the next task.

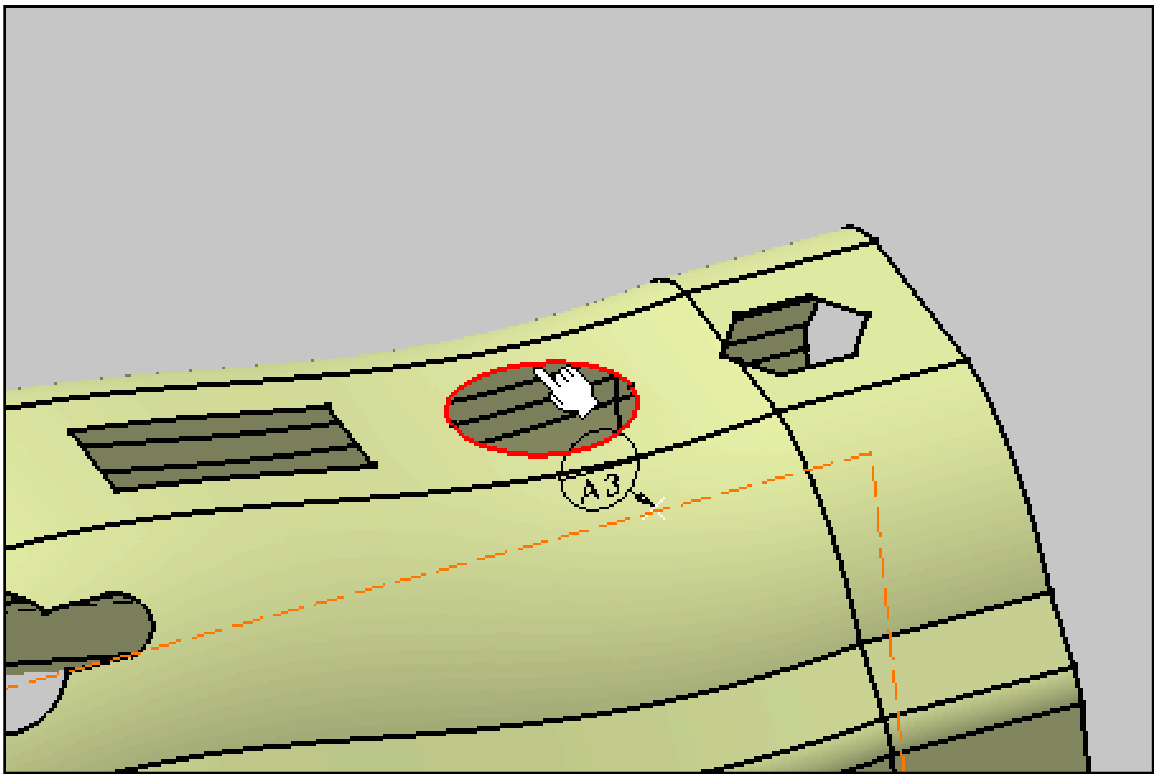
Creating Dimensions and Associated Datums

This task shows you how to create dimensions and associated datums on body in white surfaces.

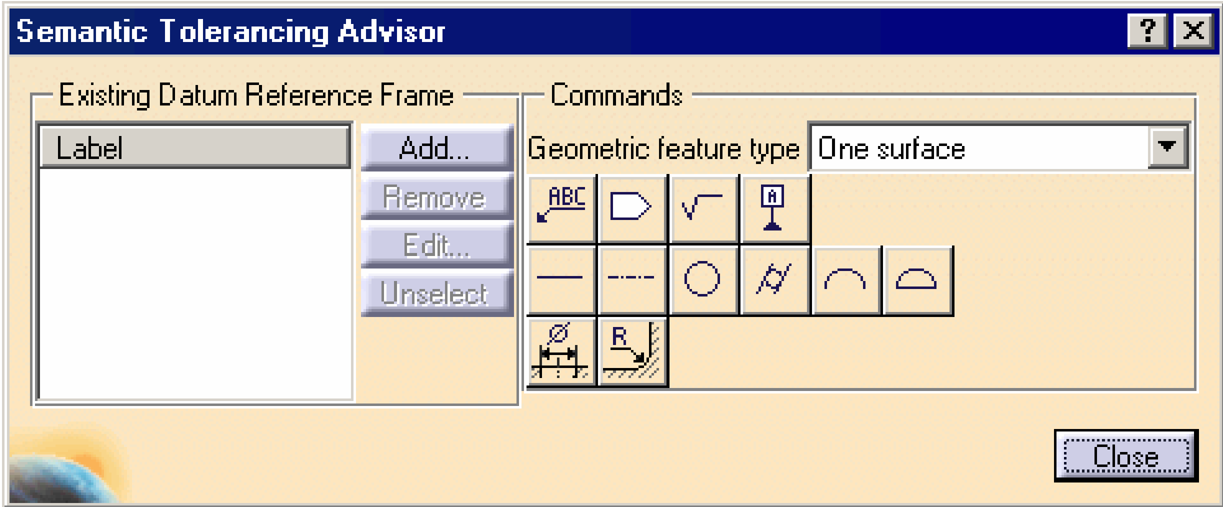
Five hole types are available for tolerancing:

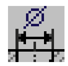
- Cylindrical hole.
- Elongated hole.
- Rectangular hole with angular corners.
- Rectangular hole with rounded corners.
- Other hole from previous.

1. Select the hole edge as shown on the part.

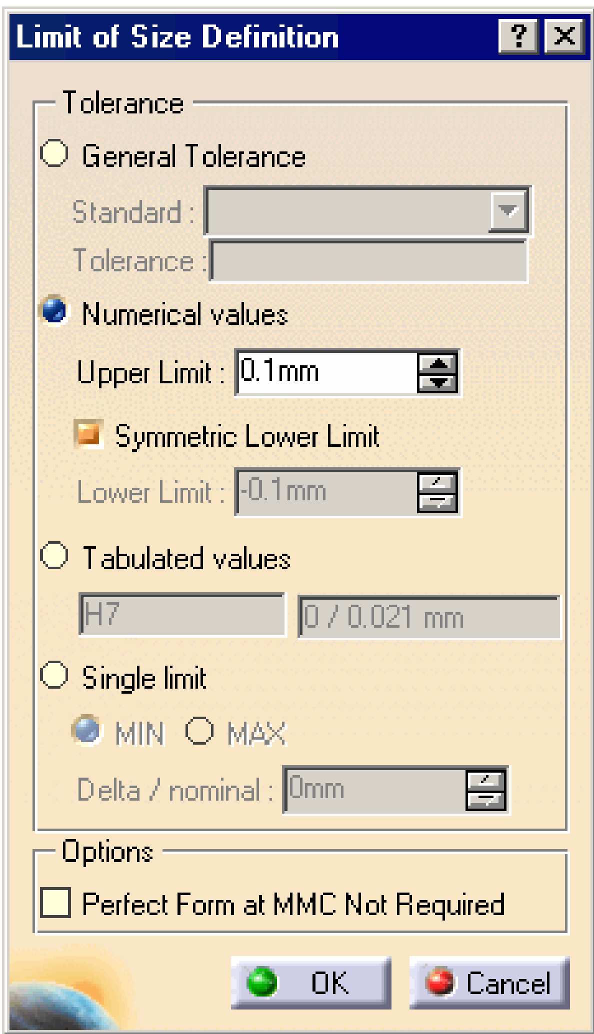


The **Semantic Tolerancing Advisor** dialog box is updated.



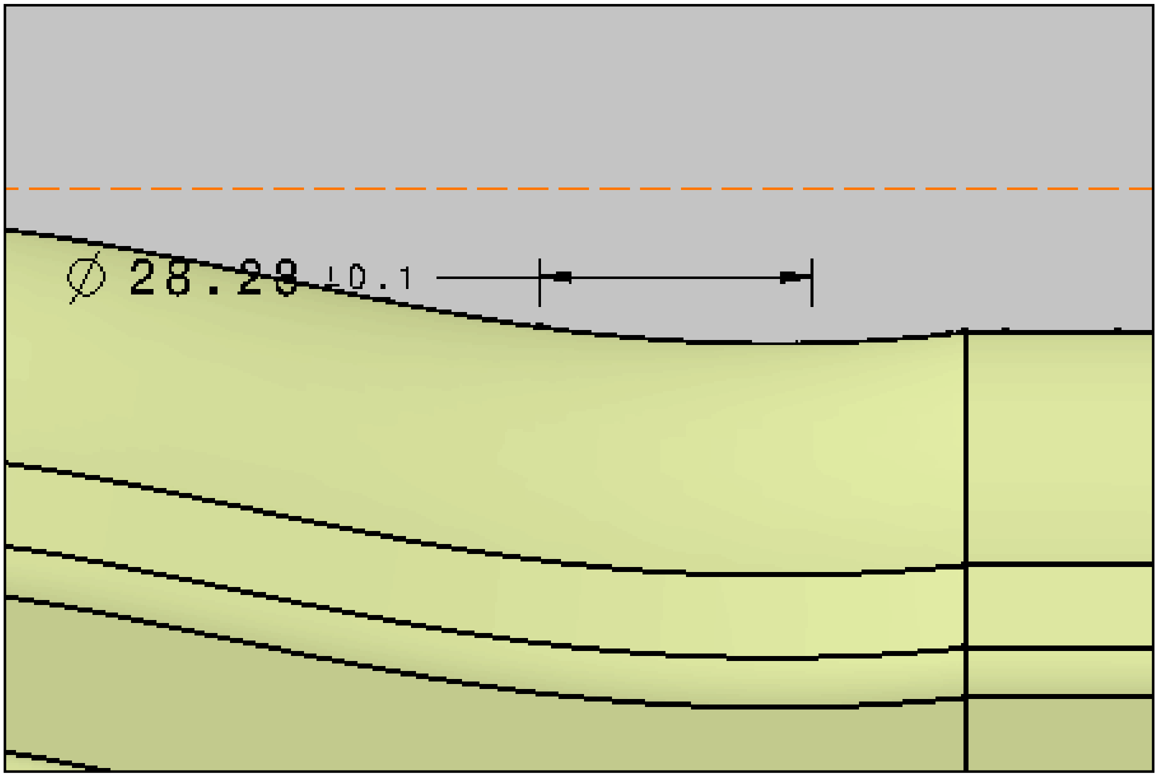
2. Click the **Diameter** icon (One surface): 

The **Limit of Size Definition** dialog box appears. Keep options as is.



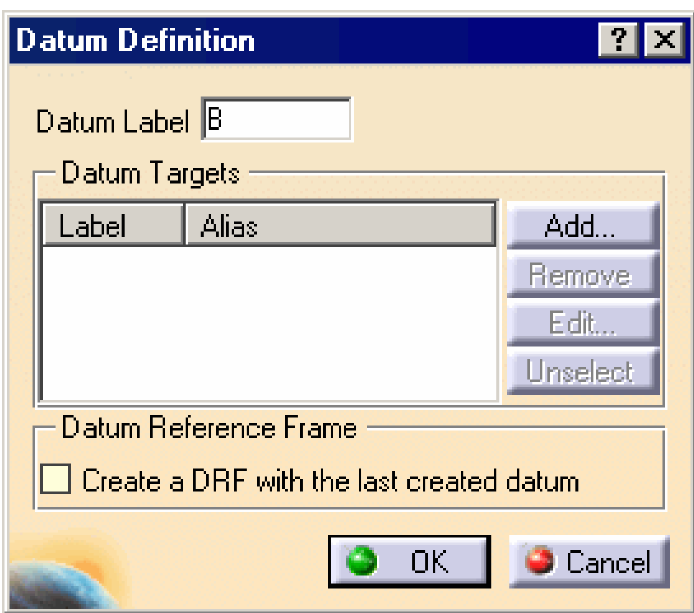
3. Click the **OK** in the dialog box.

The dimension is created.



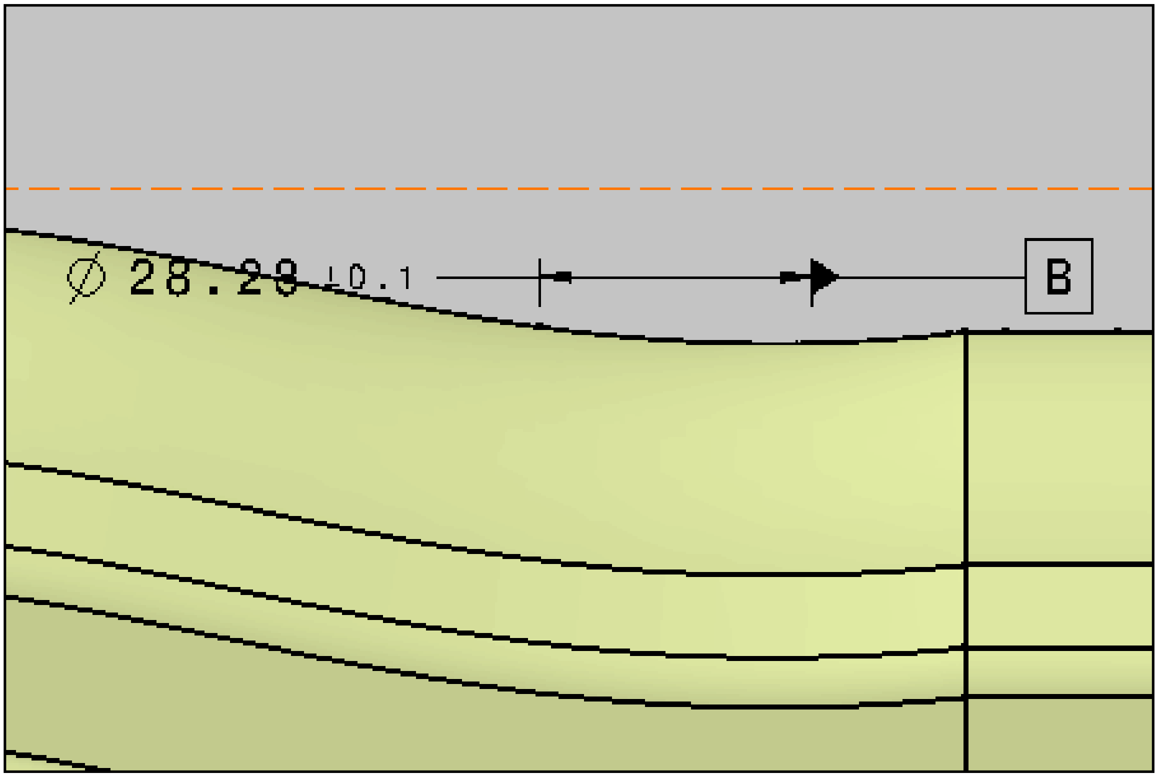
4. Click the **Semantic Datum** icon (One surface): 

The **Datum Definition** dialog box appears.

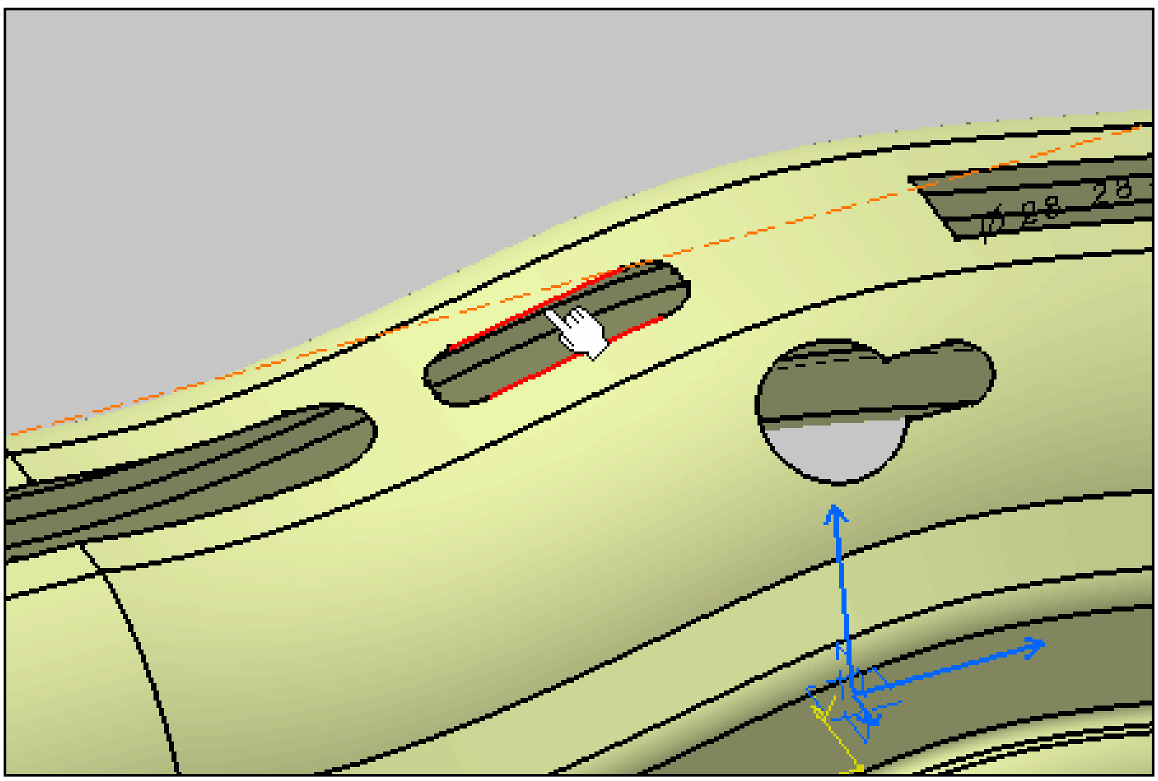


5. Click the **OK** in the dialog box.

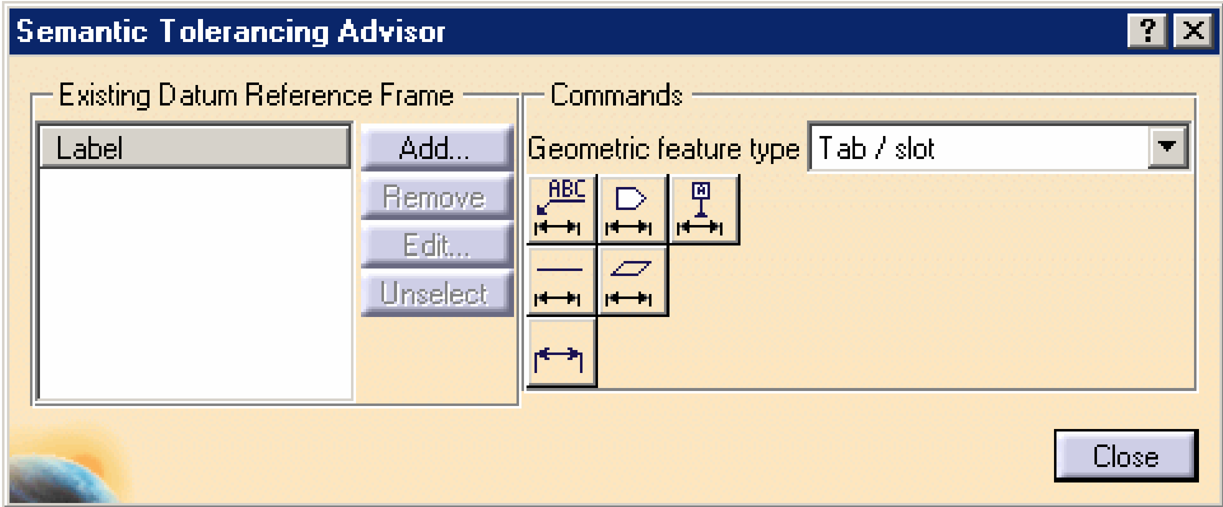
The datum is added to the dimension.

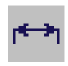


6. Select the two hole edges as shown on the part.



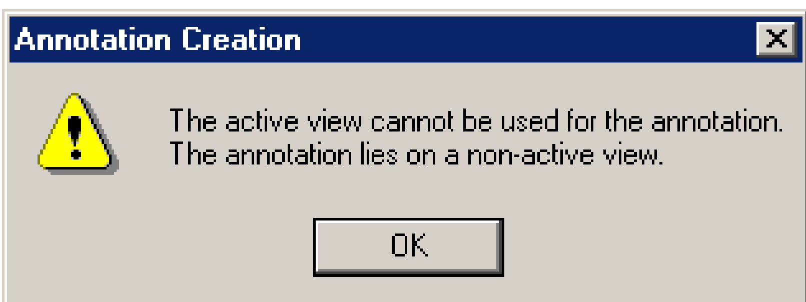
The **Semantic Tolerancing Advisor** dialog box is updated.



7. Click the **Distance Creation** icon (Tab/Slot): 

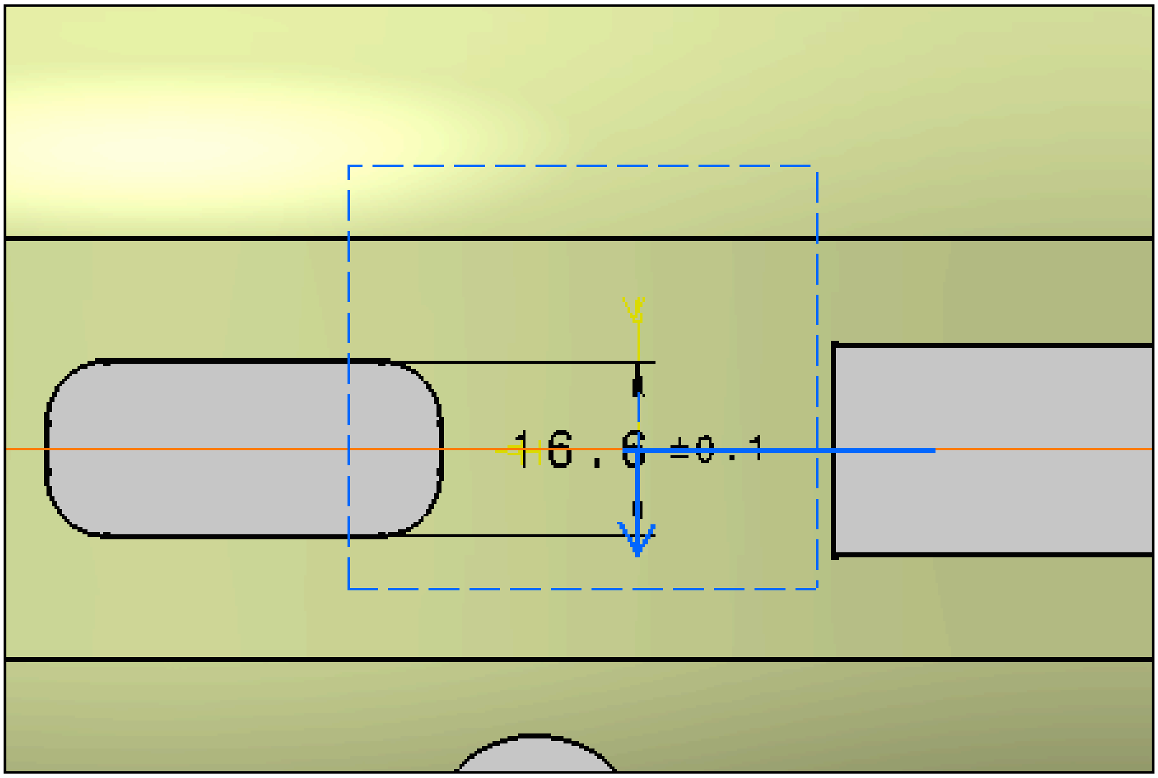
The **Limit of Size Definition** dialog box appears. Keep options as is.

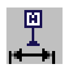
8. A message box appears: click **OK**.



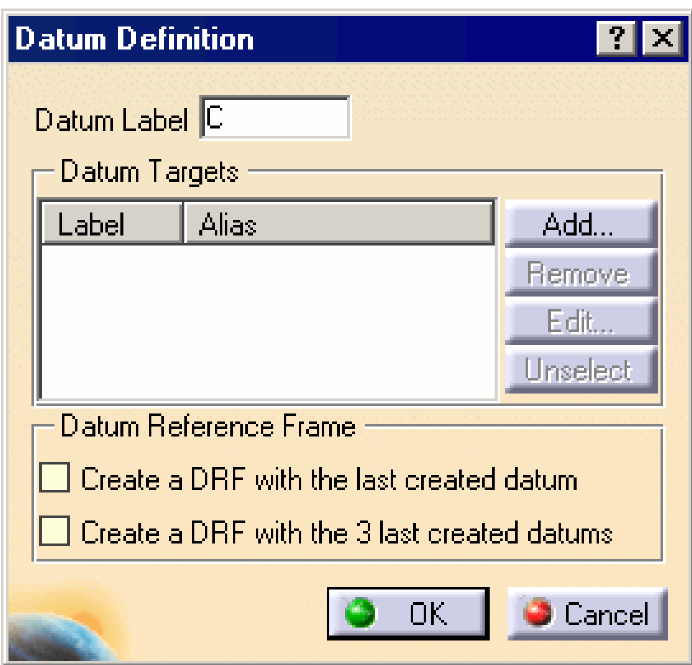
9. Click the **OK** in the dialog box.

The dimension is created.



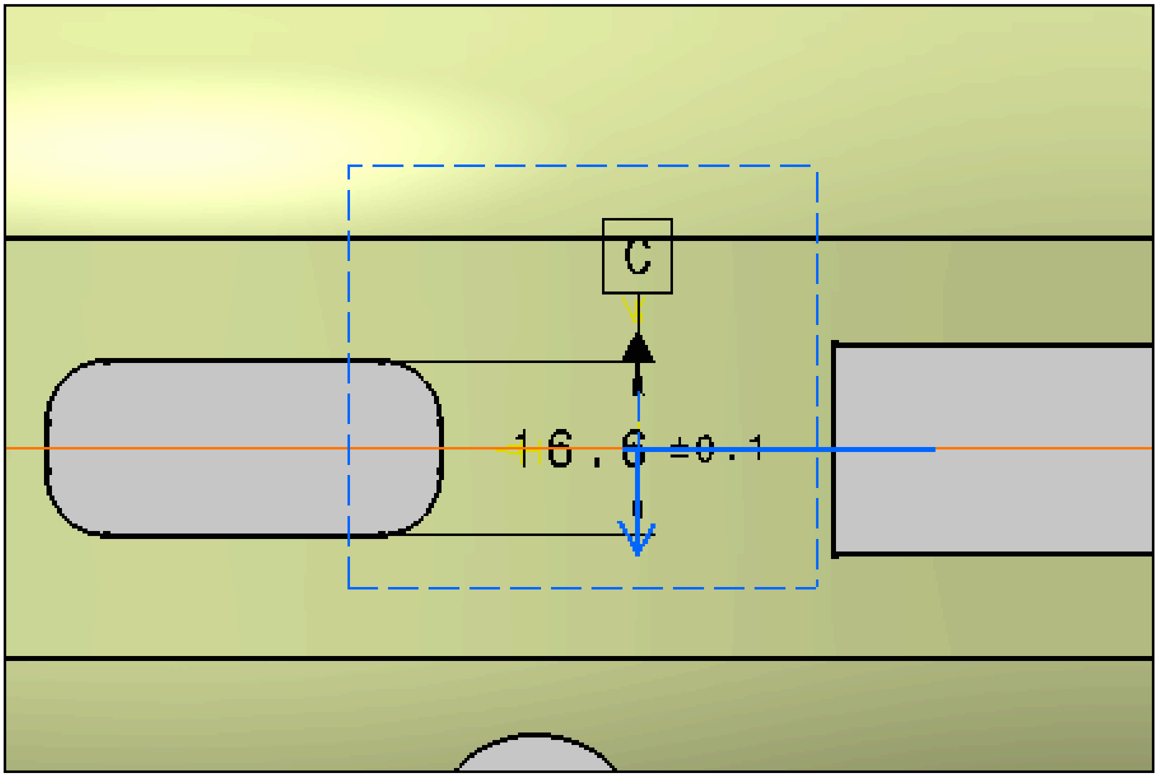
10. Click the **Semantic Datum** icon (Tab/Slot): 

The **Datum Definition** dialog box appears.

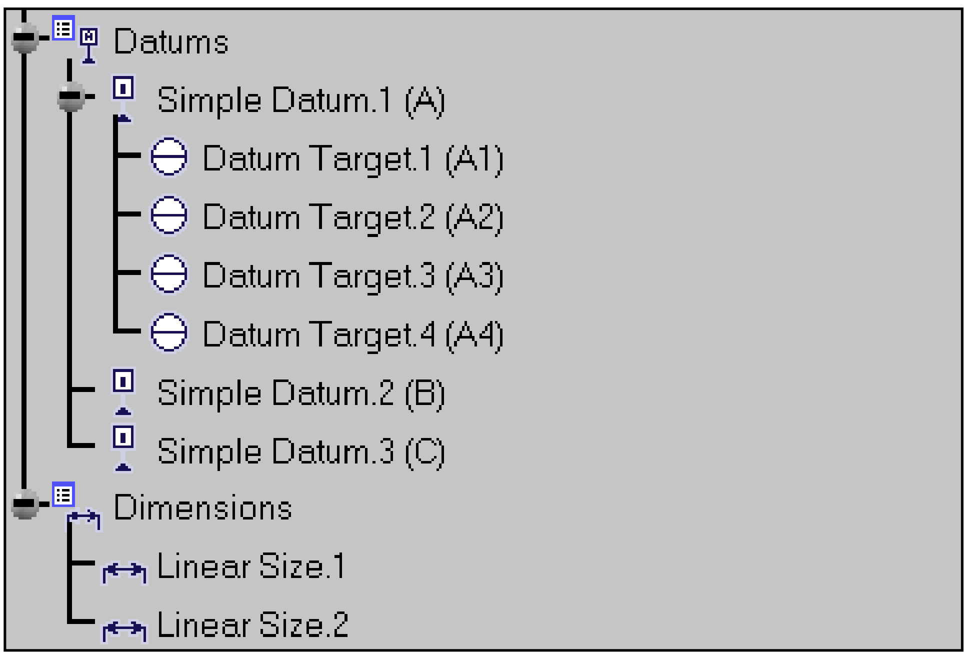


11. Click the **OK** in the dialog box.

The datum is added to the dimension.



The dimensions and datum are created in the geometry and specification tree.



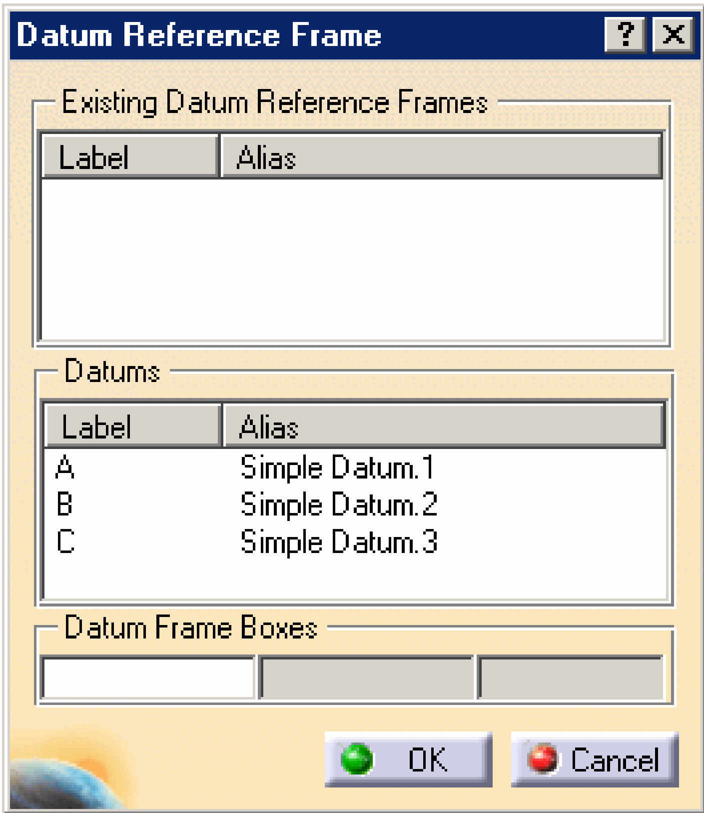
12. Do not close the **Semantic Tolerancing Advisor** dialog box to perform the next task.

Creating a Datum Reference Frame

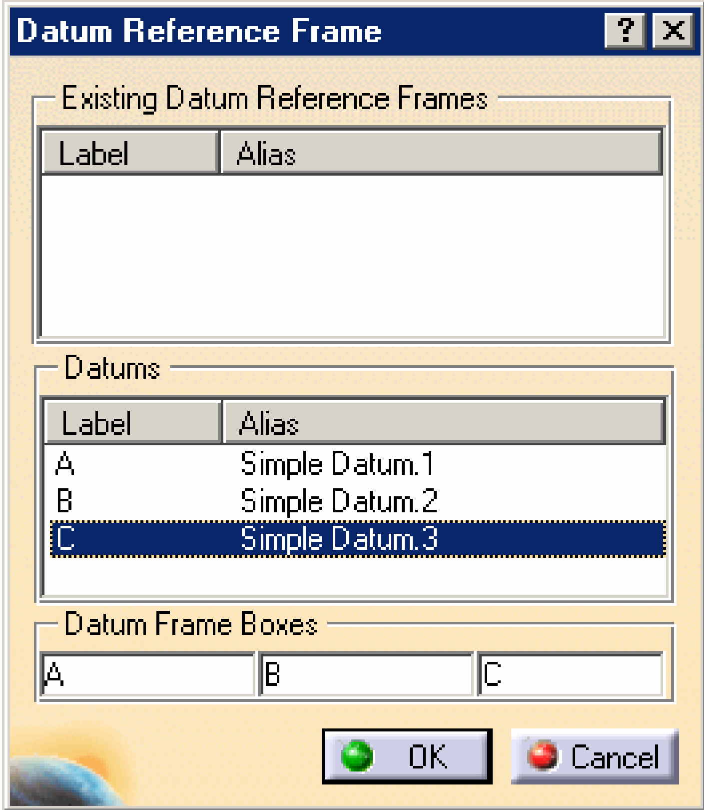
This task shows you how to create a datum reference frame on body in white surfaces.

1. Click the **Add** button in the **Semantic Tolerancing Advisor** dialog box .

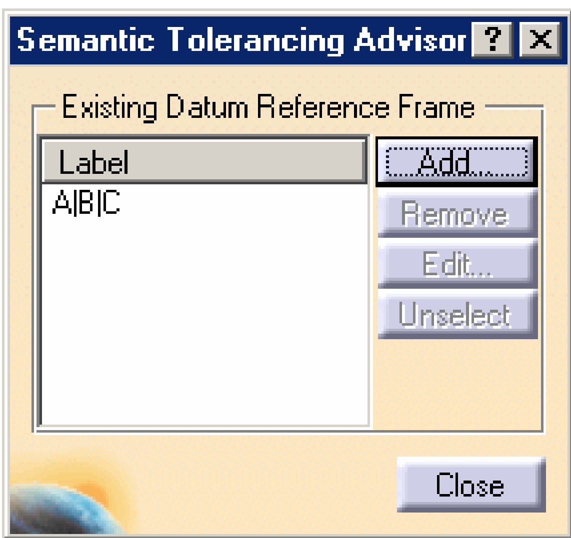
The **Datum Reference Frame** dialog box appears.



2. Select the each datum and specify its box as is.



3. Click the **OK** in the dialog box.



The **Semantic Tolerancing Advisor** dialog box is updated.

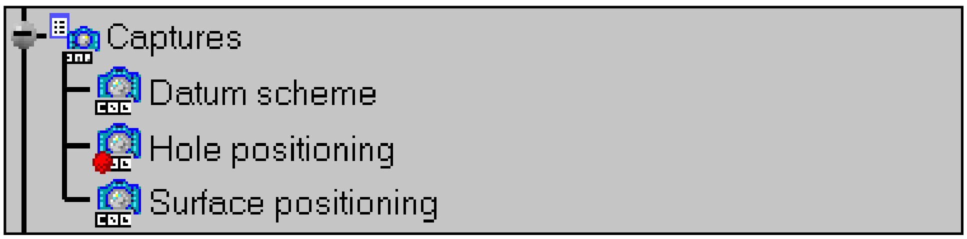
The datum reference frame is created in the specification tree.

4. Do not close the **Semantic Tolerancing Advisor** dialog box to perform the next task.

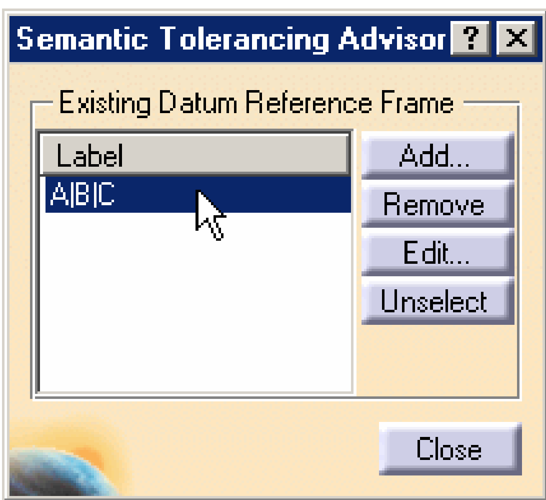
Tolerancing Body in White Holes

This task shows you how to create geometrical tolerances on body in white holes.

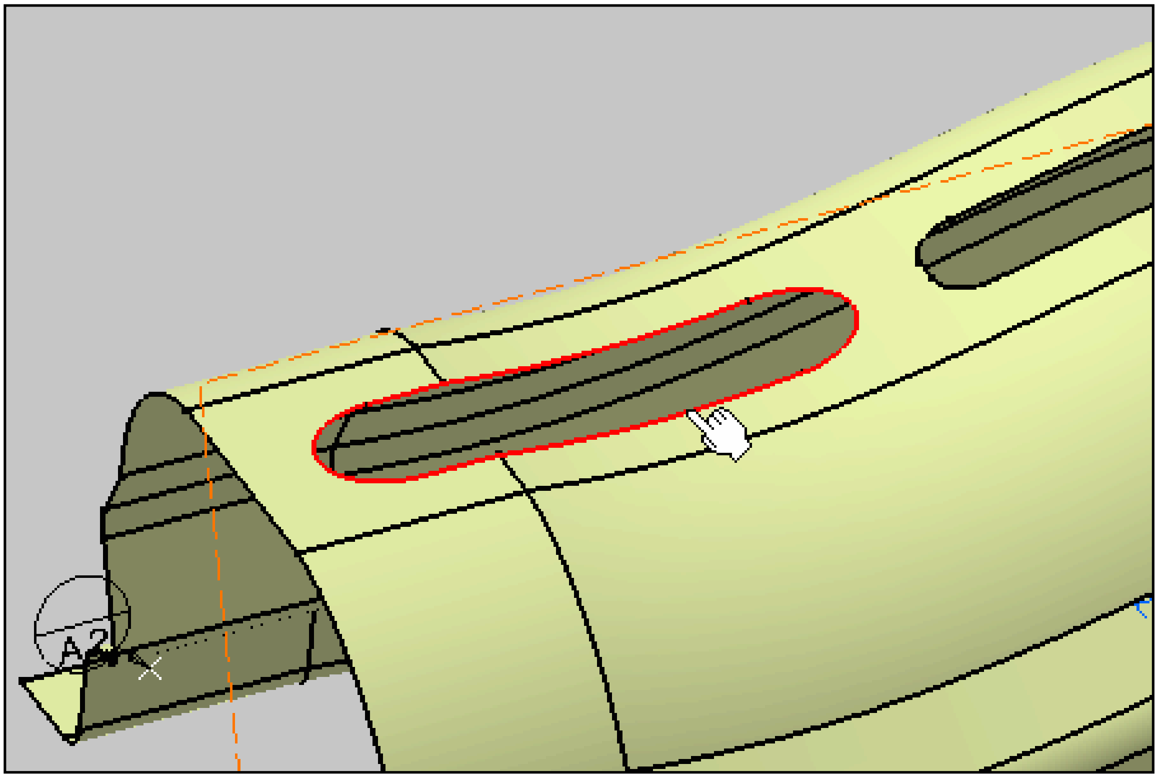
1. Right-click the **Datum scheme** capture and select **Unset Current** form the contextual menu, right-click the **Hole positioning** capture and select **Set Current** form the contextual menu.

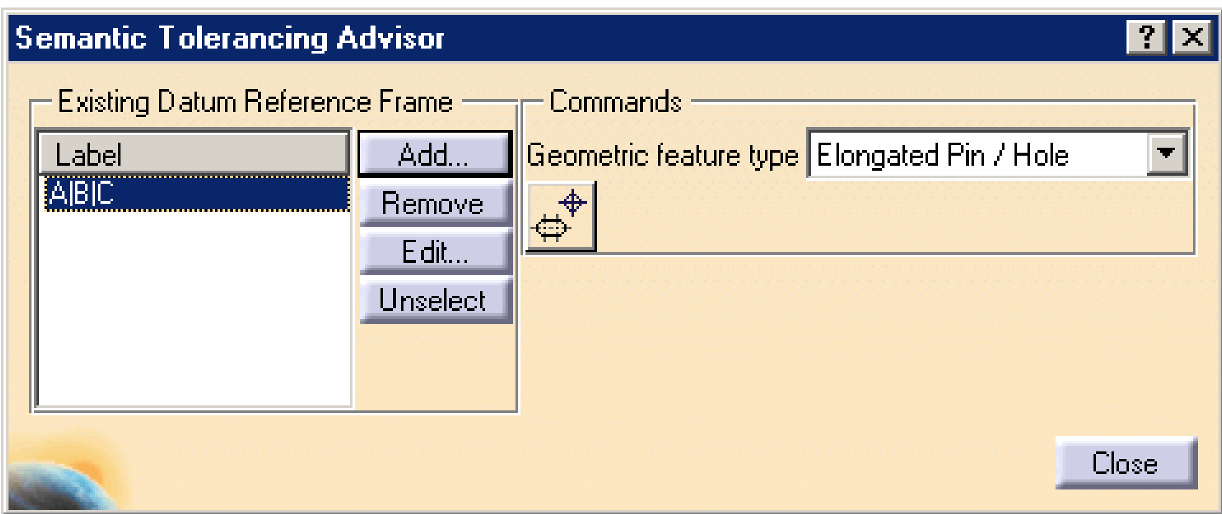


2. Select the datum reference frame in the **Semantic Tolerancing Advisor** dialog box.

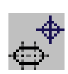


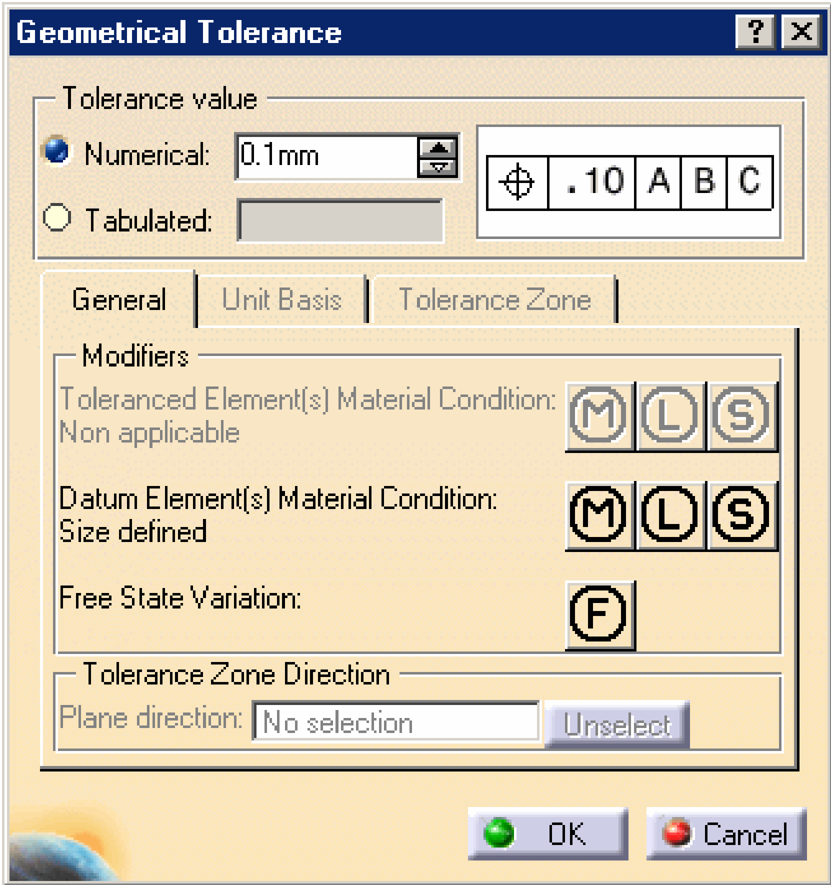
3. Select the whole hole edges as shown on the part.





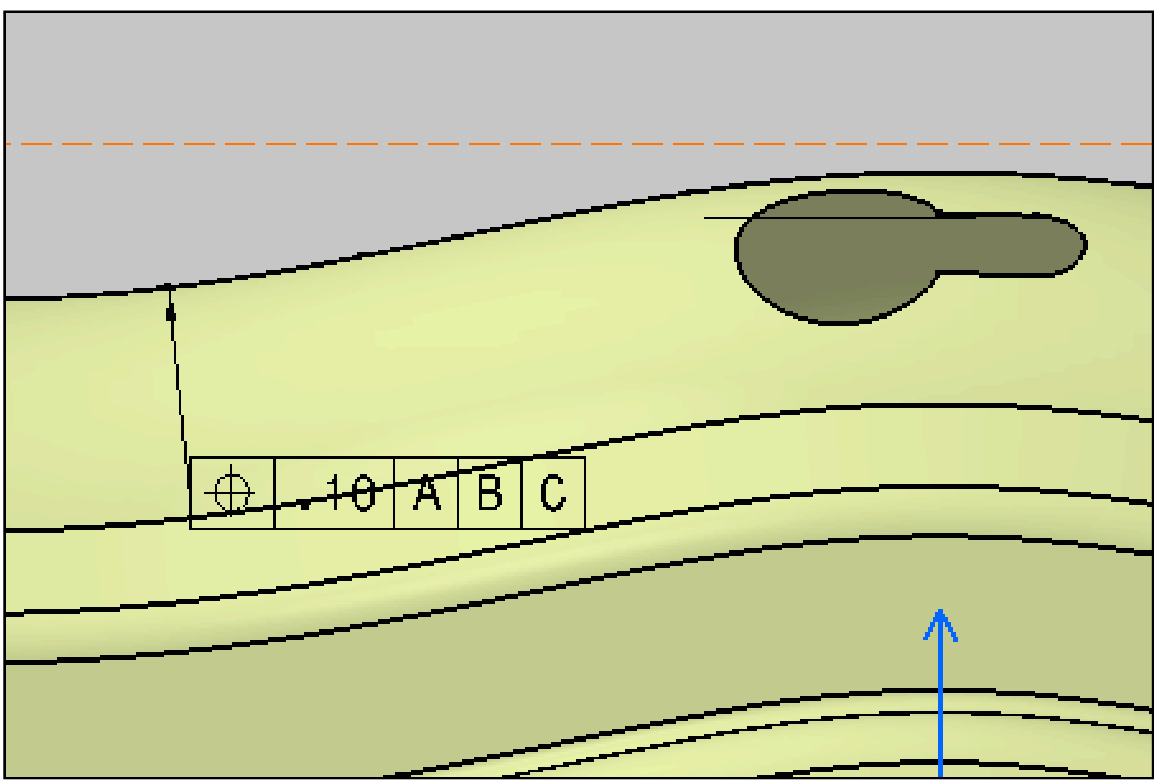
The **Semantic Tolerancing Advisor** dialog box is updated. Note the geometric feature type: **Elongated Pin/Hole**.

4. Click the **Position with DRF Specification** icon: 



The **Geometrical Tolerance** dialog box appears. Keep options as is.

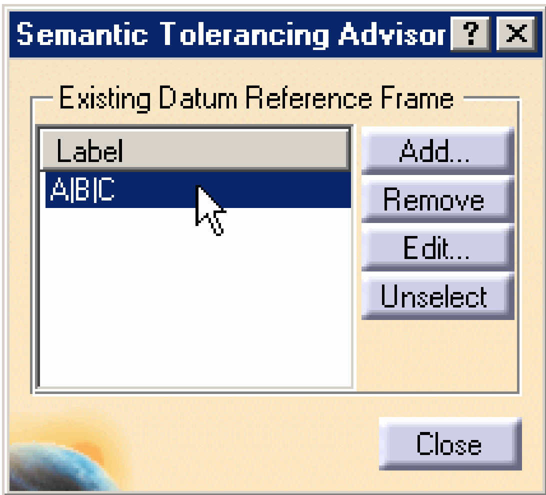
5. Click the **OK** in the dialog box.



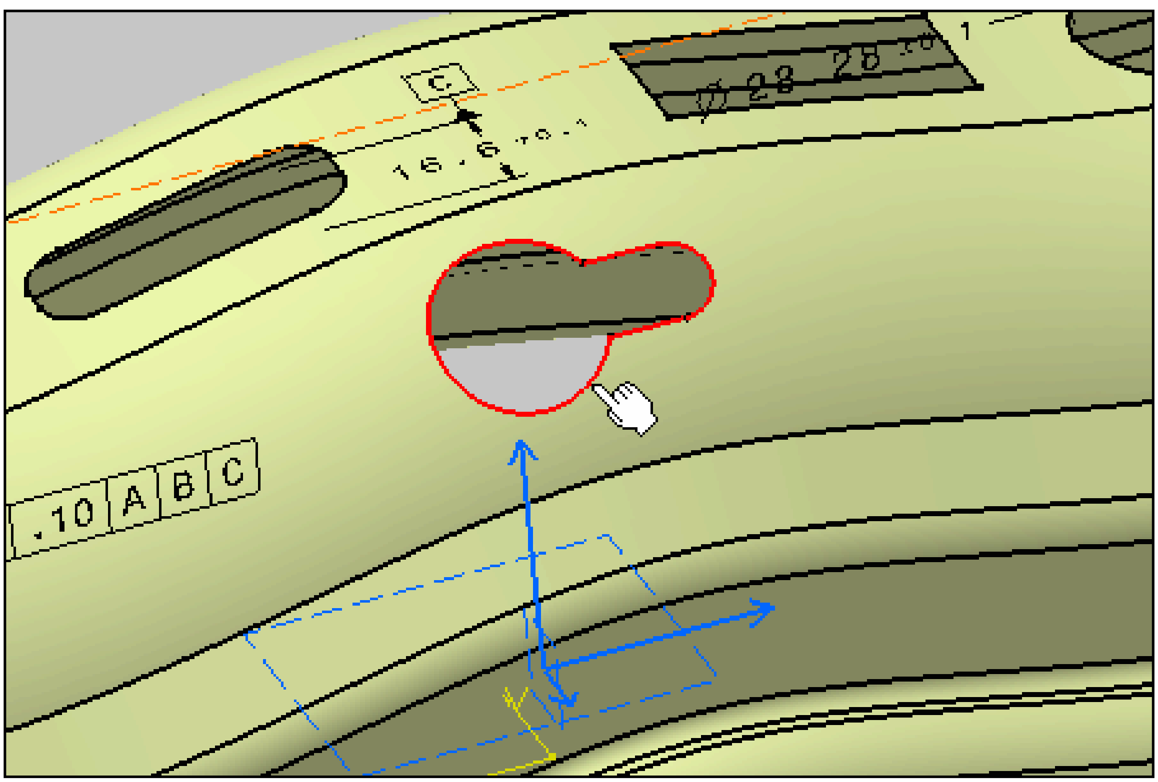
The geometrical tolerances is created.

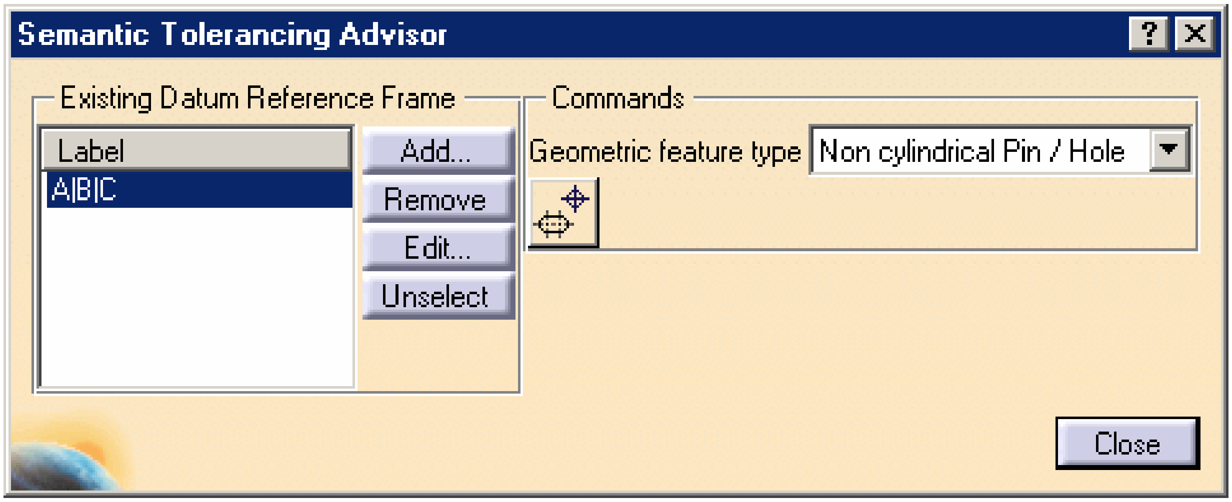
The geometrical tolerances is created in the specification tree.

6. Select the datum reference frame in the **Semantic Tolerancing Advisor** dialog box.

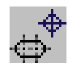


7. Select the whole hole edges as shown on the part.



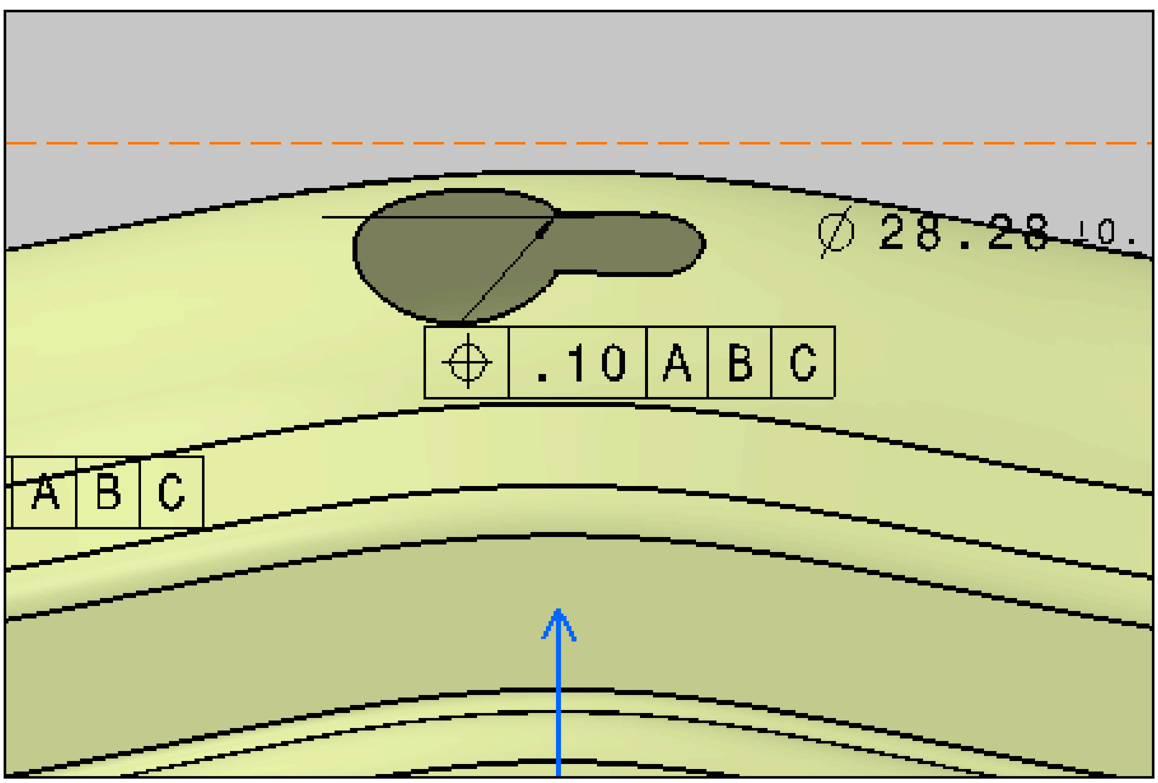


The **Semantic Tolerancing Advisor** dialog box is updated. Note the geometric feature type: **Non cylindrical Pin/Hole**.

8. Click the **Position with DRF Specification** icon: 

The **Geometrical Tolerance** dialog box appears. Keep options as is.

9. Click the **OK** in the dialog box.



The geometrical tolerance is created.

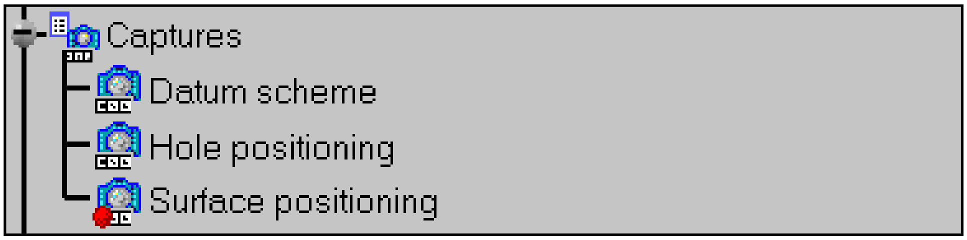
The geometrical tolerance is created in the specification tree.

10. Do not close the **Semantic Tolerancing Advisor** dialog box to perform the next task.

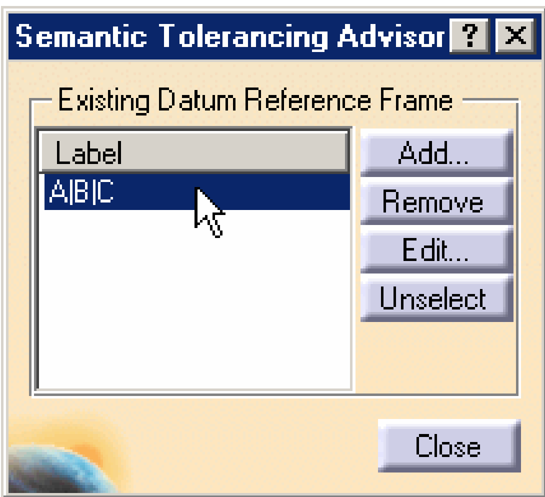
Tolerancing Body in White Surface

This task shows you how to create geometrical tolerances on a body in white surface.

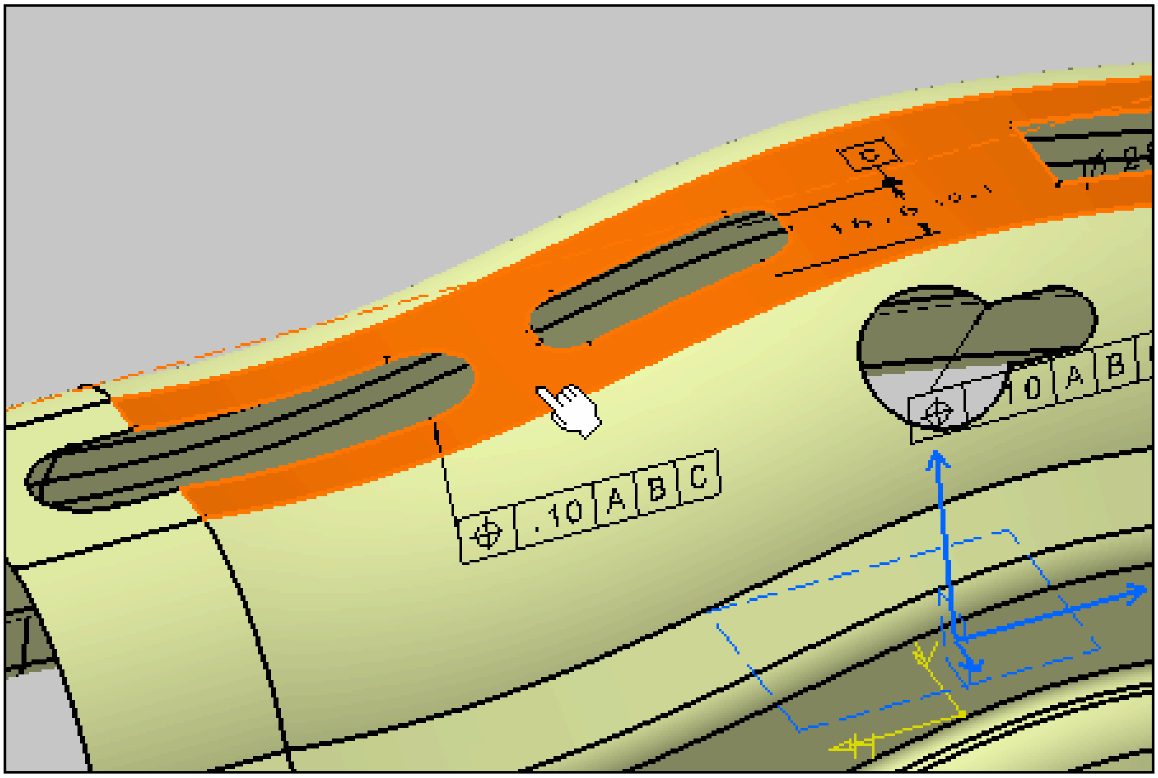
1. Right-click the **Hole positioning** capture and select **Unset Current** form the contextual menu, right-click the **Surface positioning** capture and select **Set Current** form the contextual menu.

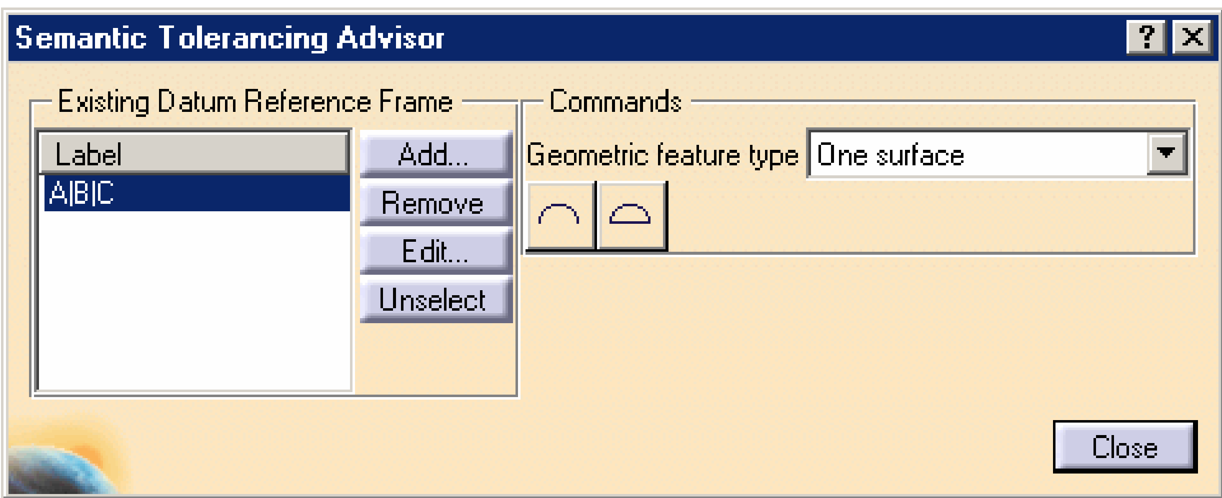


2. Select the datum reference frame in the **Semantic Tolerancing Advisor** dialog box.

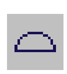


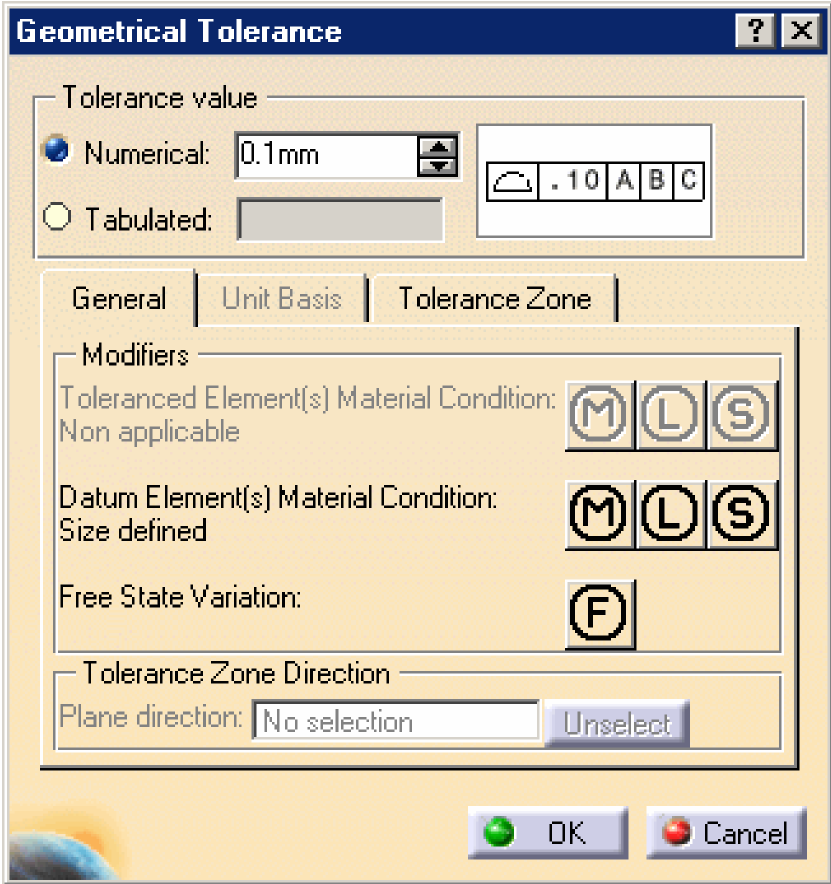
3. Select the surface as shown on the part.





The **Semantic Tolerancing Advisor** dialog box is updated. Note the geometric feature type: **One surface**.

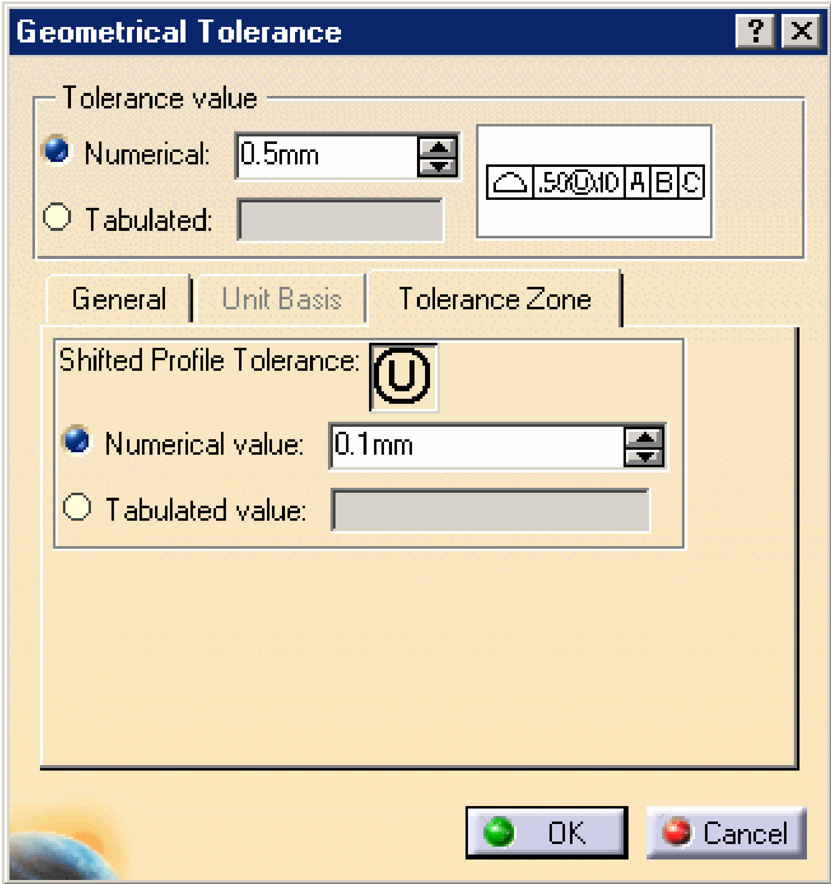
4. Click the **Profile of a Surface** icon: 



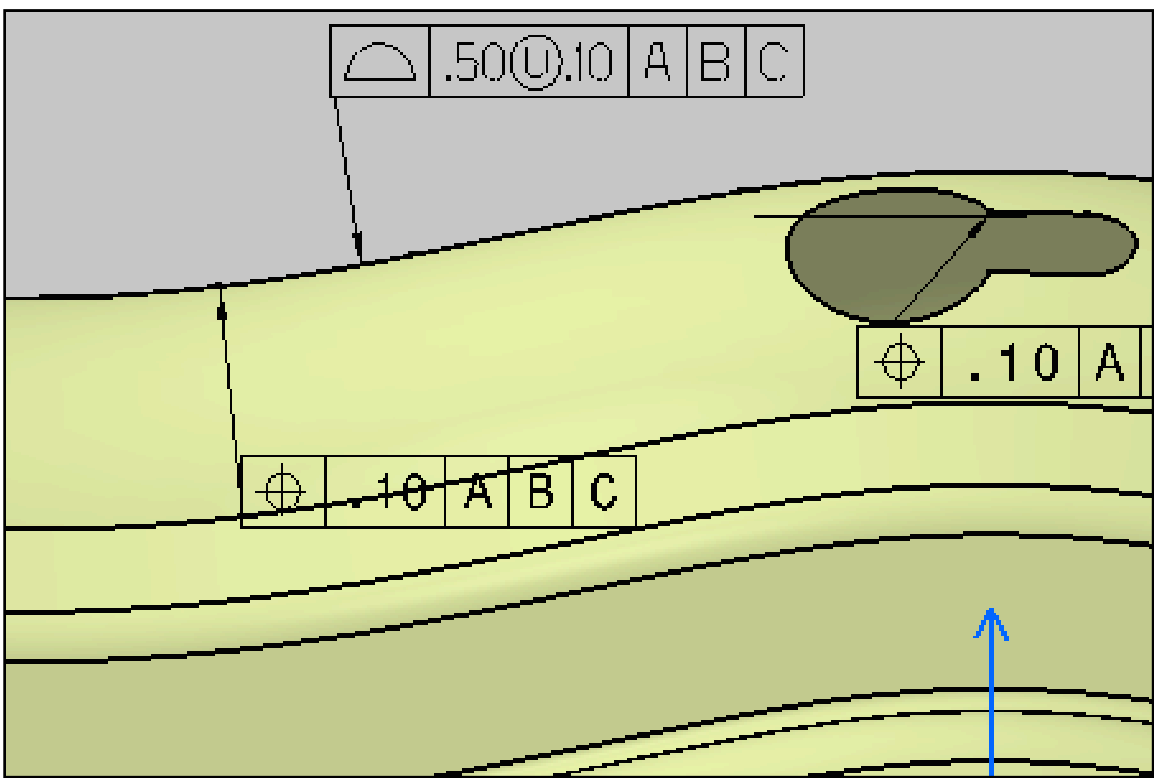
The **Geometrical Tolerance** dialog box appears.

5. Set the numerical value to 0.5 mm.

6. Select the **Tolerance Zone** tab and check the **Shifted Profile Tolerance** icon.



7. Click the **OK** in the dialog box.



The geometrical tolerance is created.

The geometrical tolerance is created in the specification tree.

8. Right-click the **Surface positioning** capture and select **Unset Current** form the contextual menu.

Creating Annotations

Creating Texts: click this icon, select a face and enter your text in the dialog box.

Creating Texts: click this icon, select a face and enter your text in the dialog box.

Creating Texts: click this icon, select a face and enter your text in the dialog box.

Creating Flag Notes: click this icon, select the object you want to represent the hyperlink, enter a name for the hyperlink and the path to the destination file.

Creating Flag Notes: click this icon, select the object you want to represent the hyperlink, enter a name for the hyperlink and the path to the destination file.

Add an Attribute Link: display the **Manage Hyperlink** or **Text Editor** dialog box, enter the text you need, select the annotation in the geometry area, right-click and select the **Attribute Link** contextual command. Select the appropriate geometry to access the parameter of interest, and select the parameter you wish to attach in the **Attribute Link** dialog box.

Create Datum Elements: click this icon, select the attachment surface and the anchor point of the datum feature, then enter the label in the dialog box .

Create Datum Targets: click this icon, select a face and enter your value and symbols in the dialog box.

Create Geometrical Tolerance: click this icon, select the element and define characteristics and values for Line 1 and/or Line2.

Create Roughness Symbol: click this icon, select the element and define roughness characteristics.

Creating Isolated Annotations: click an annotation creation icon, and click anywhere in the free space.

Create Dimensions: click this icon, select a geometrical element.

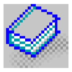
Create Basic Dimension: click this icon, select the context.

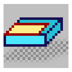
Create Coordinate Dimensions: click this icon, specify whether you want to create a 2D or a 3D coordinates dimension, and then select a vertex, a point (on a curve, on a plane, a coordinate), a line center or a point on a curve.

Create Stacked Dimensions: click this icon, and select the elements to include within the stacked dimensions system.

Create Cumulated Dimensions: click this icon, and select the elements to include within the cumulated dimensions system.

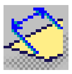
Creating Curvilinear Dimensions: click this icon, select a curve or an edge, and optionally choose a representation mode (offset, parallel, linear).

 Instantiate a Note Object Attribute: click this icon, select a Note Object Attribute, select a geometrical element.

 Create a Partial Surface: click this icon, select the restricted surface, select the restricting surface.

 Create a Deviation: click this icon, select the component, select the point to define an annotation.

 Create a Correlated Deviation: click this icon, select the component, select the set of point to define a correlated annotation.

 Create a Distance Between Two Points: click this icon, select the parent component, select the start and end points to define a distance between two points annotation.

Creating Texts

This task shows you how to create an annotation text.

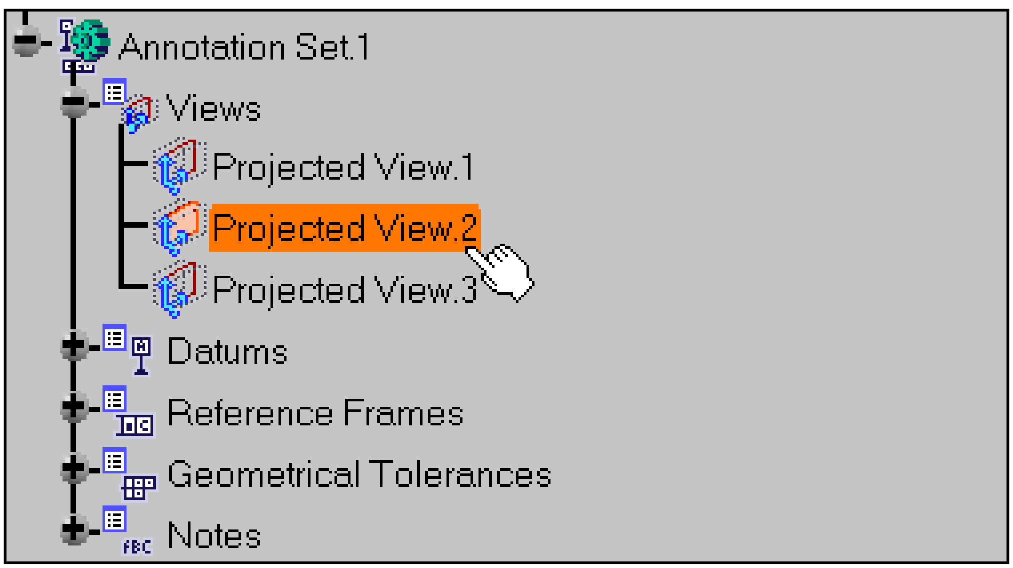
Three kinds of text may be created:

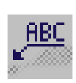
- Text with Leader
- Text
- Text parallel to screen

A text is assigned an unlimited width text frame.
You can set graphic properties (anchor point, text size and justification) either before or after you create the free text.
See Setting Basic Graphical Properties.
You can change any text to another kind at any time.

Open the Annotations_Part_04.CATPart document.

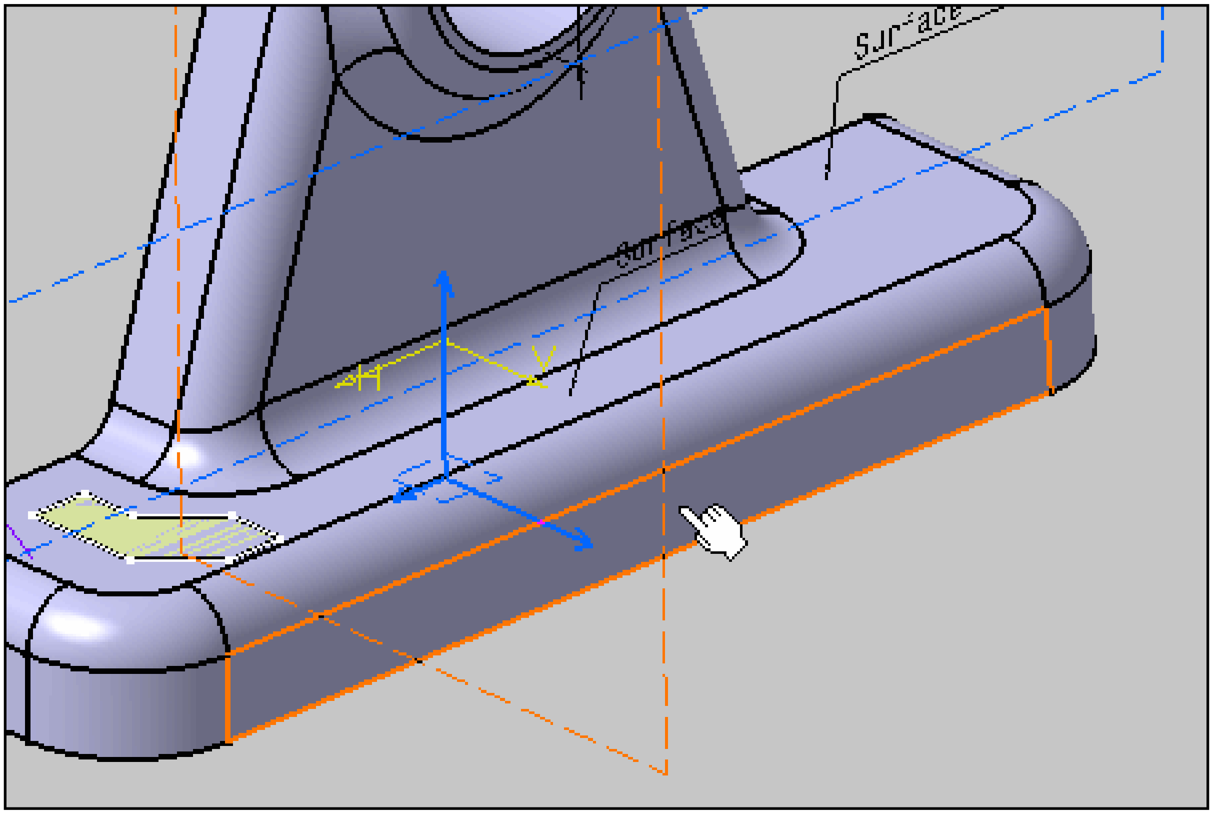
1. Activate the **Projected View.2** annotation plane.



2. Click the **Text with Leader** icon: 

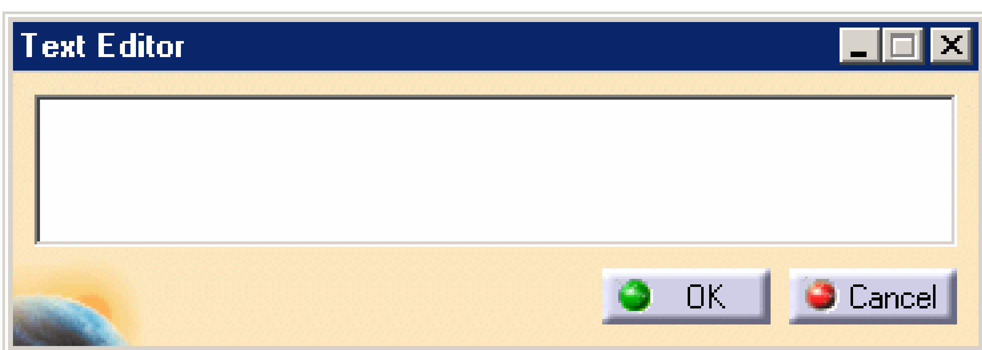
3. Select the face as shown to define a location for the arrow end of the leader.

This scenario illustrates the creation of a text by selecting geometry, but you can also select any Part Design or Generative Shape Design feature in the specification tree. In this case, the created annotation will not be attached to the selected feature, but to its geometrical elements at the highest level.

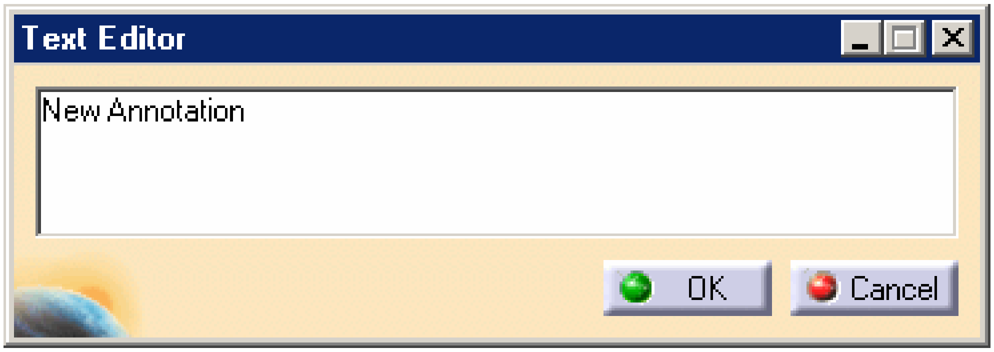


If the active view is not valid, a message appears informing you that you cannot use the active view.
This means that the annotation will be displayed in an annotation plane normal to the selected face.
For more information, see View/Annotation Planes.

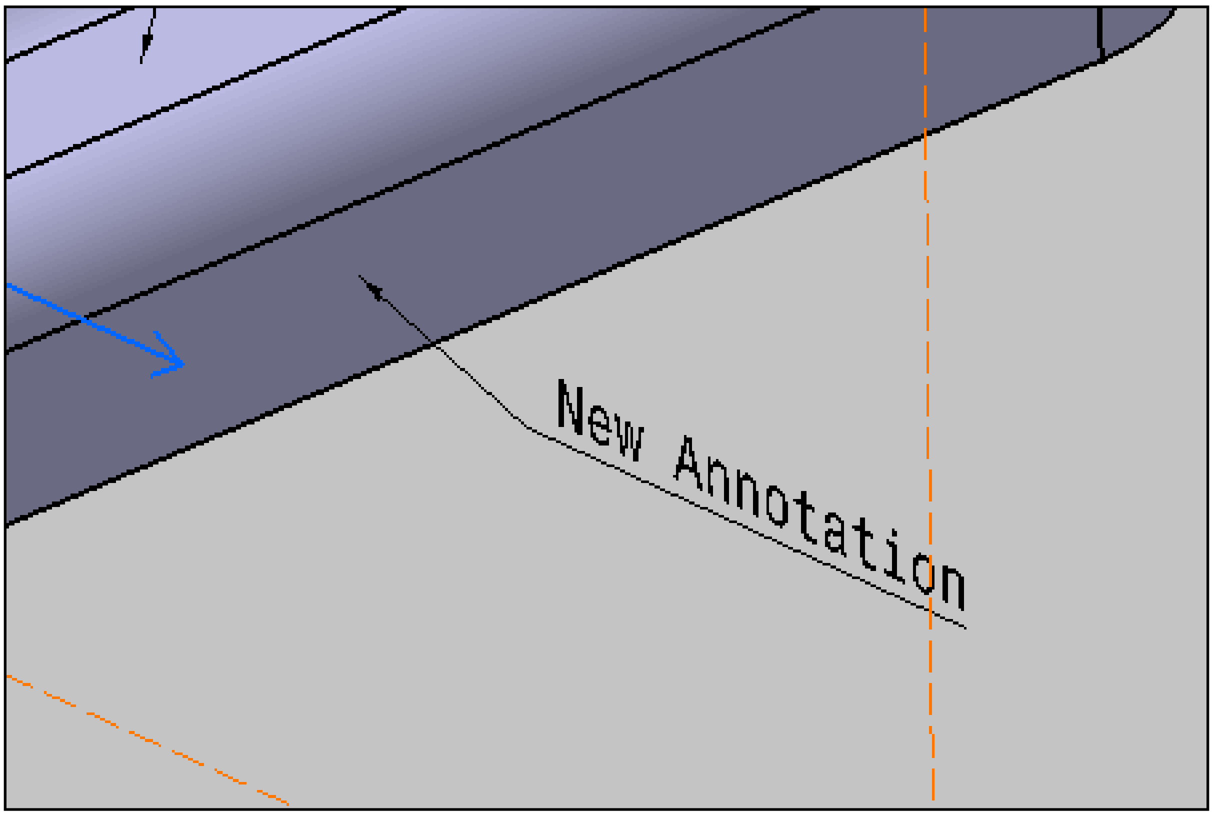
The **Text Editor** dialog box appears.



4. Enter your text, for example "New Annotation" in the dialog box.



5. Click **OK** to end the text creation. You can click anywhere in the geometry area too.



The text appears in the geometry.
The text (identified as Text.xxx) is added to the specification tree.

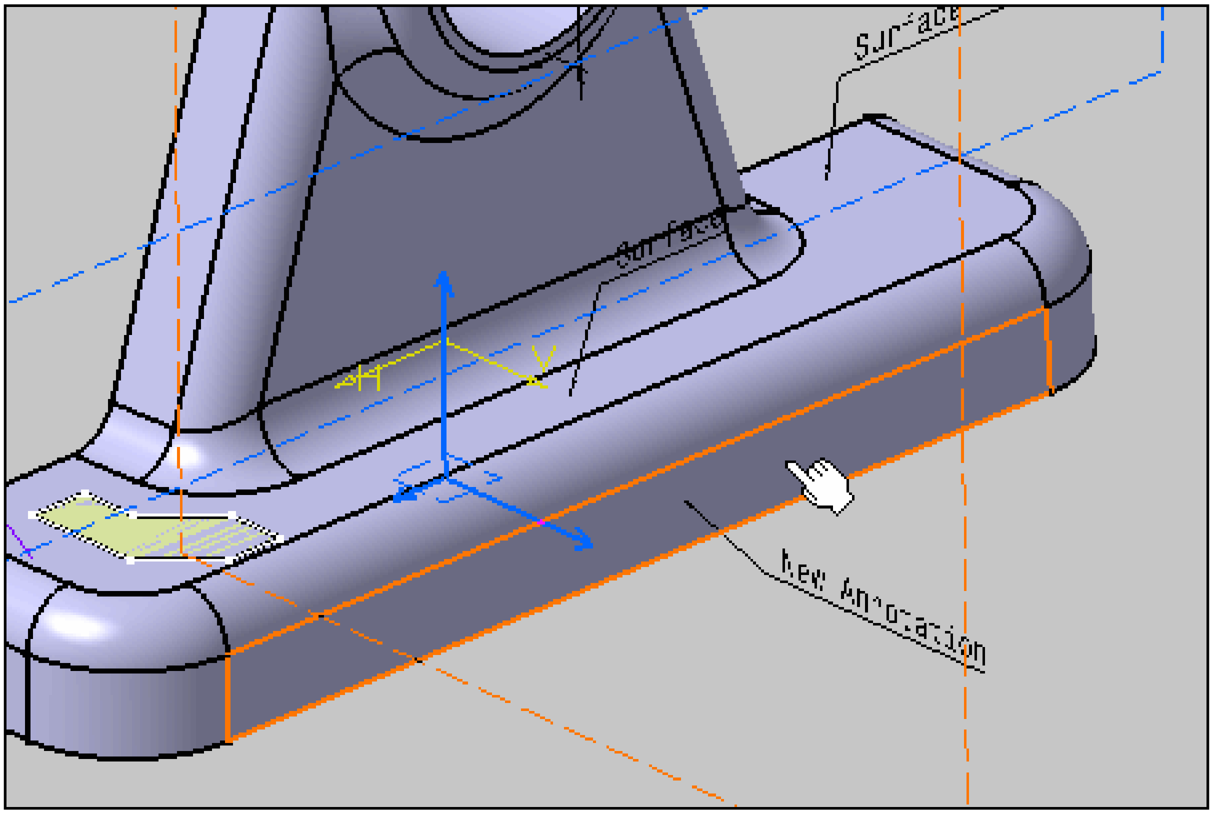
The leader is associated with the element you selected. If you move either the text or the element, the leader stretches to maintain its association with the element.

Moreover, if you change the element associated with the leader, application keeps the associativity between the element and the leader.

Note that using the Text Properties toolbar, you can define the anchor point, text size and justification.

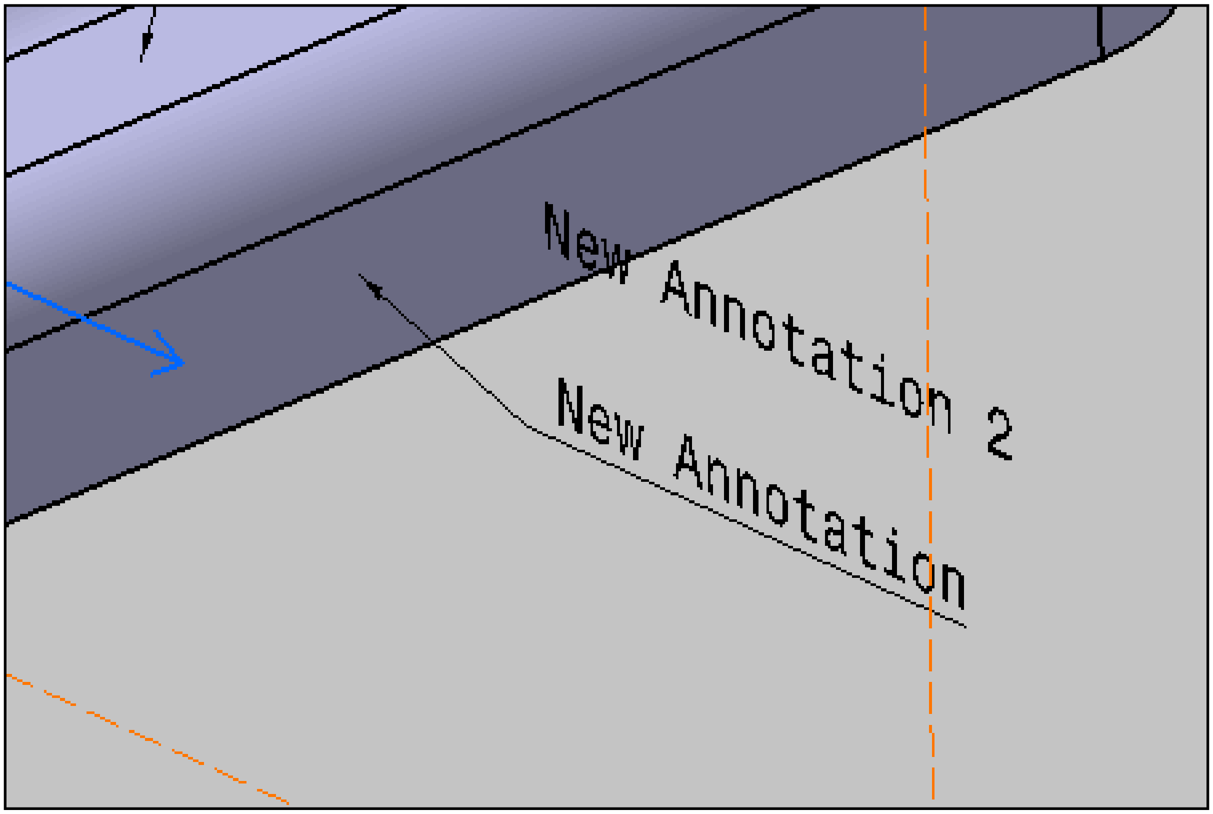
6. Click the **Text** icon: 

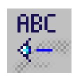
7. Select the face as shown.



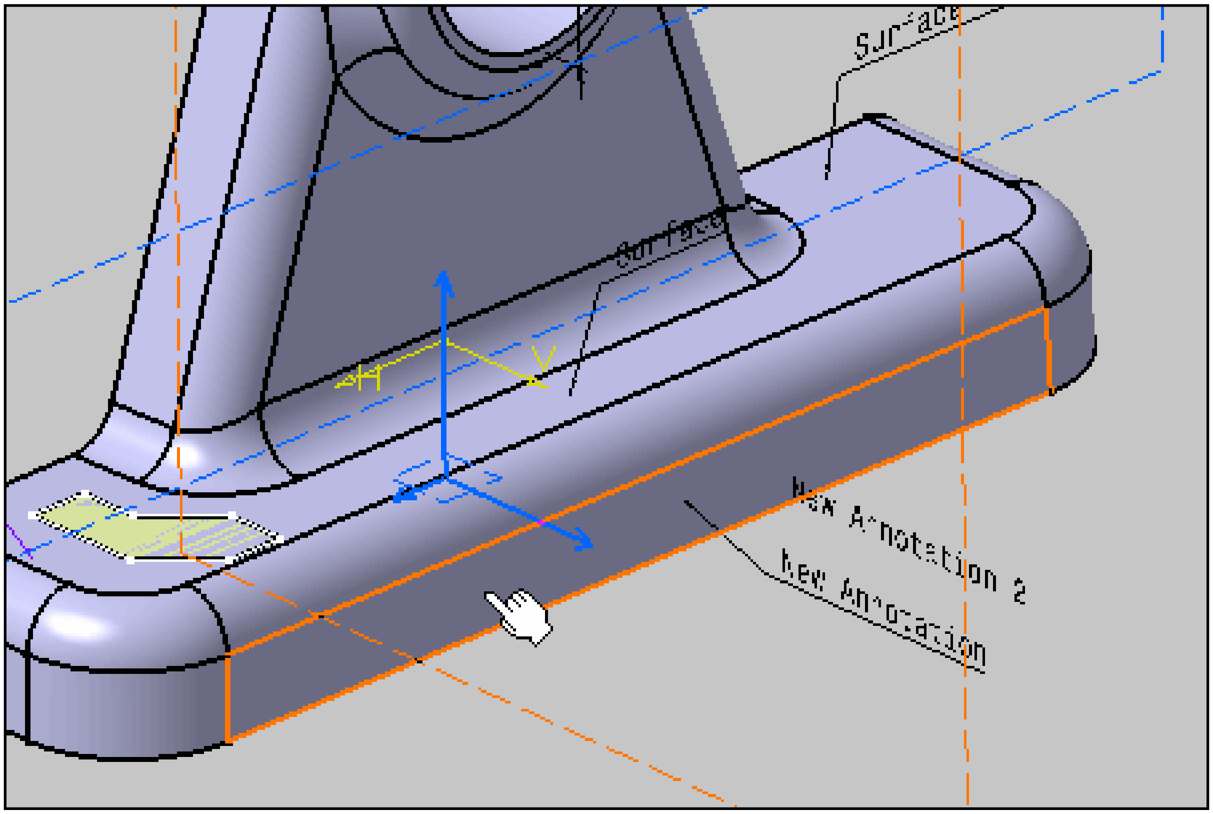
8. Enter your text, for example "New Annotation 2" in the dialog box and click **OK**.

The text appears in the geometry.
The text (identified as Text.xxx) is added to the specification tree.



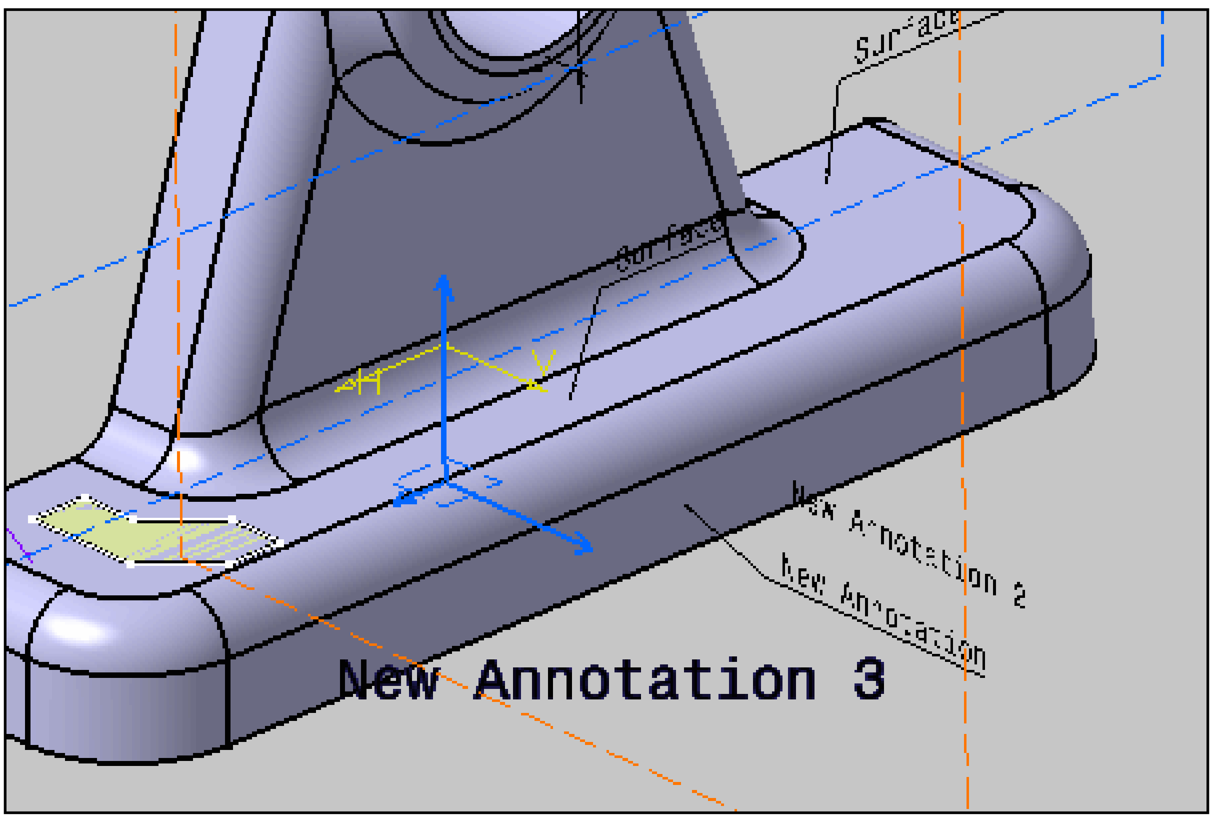
9. Click the **Text Parallel to Screen** icon: 

10. Select the face as shown.



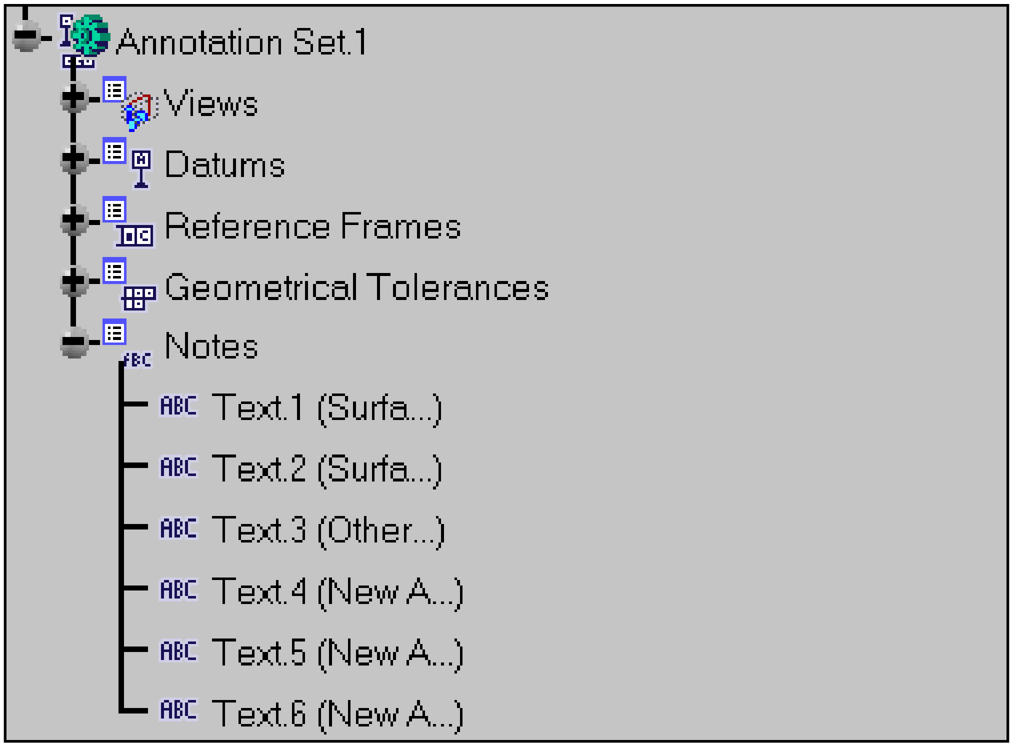
11. Enter your text, for example "New Annotation 3" in the dialog box and click **OK**.

The text appears in the geometry.



See also Setting Annotation Parallel to Screen.

The texts (identified as Text.xxx and its text between brackets) are added to the specification tree in the **Notes** group.



You can move a text using either the drag capability. See Moving Annotations.

Note also that you can resize the manipulators.

For more information, refer to Customizing for 3D Functional Tolerancing & Annotations.

Creating Flag Notes

This task shows you how to create an annotation flag note.

A flag note allows you to add links to your document and then use them to jump to a variety of locations, for example to a marketing presentation, a text document or a HTML page on the intranet. You can add links to models, products and parts as well as to any constituent elements.

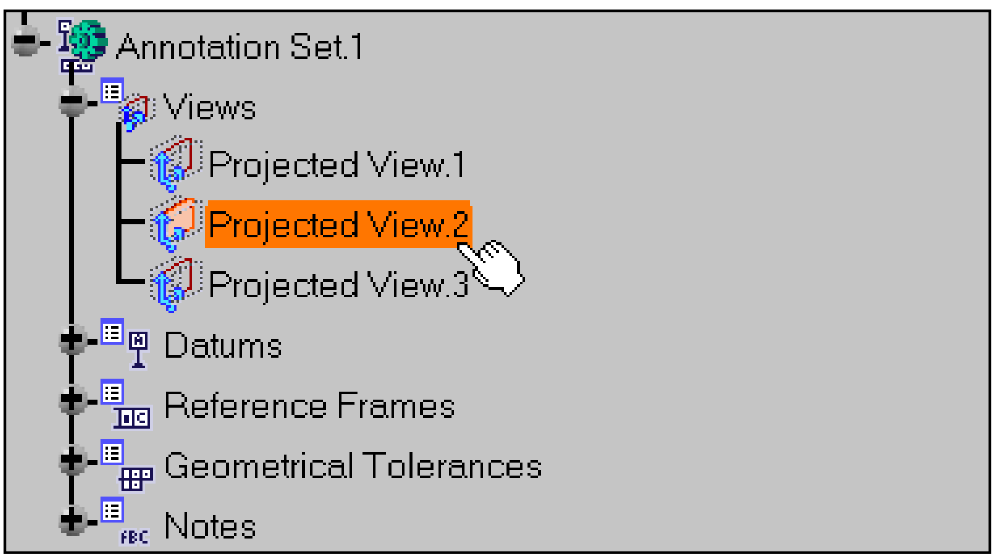
Two kinds of flag note may be created:

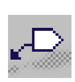
- Flag note with Leader
- Flag note

A flag note is assigned an unlimited width text frame.
You can set graphic properties (anchor point, text size and justification) either before or after you create the free text.
See Setting Basic Graphical Properties.
You can change any flag note to another kind at any time.
You can specify a hidden text to the flag note.

Open the Annotations_Part_04.CATPart document.

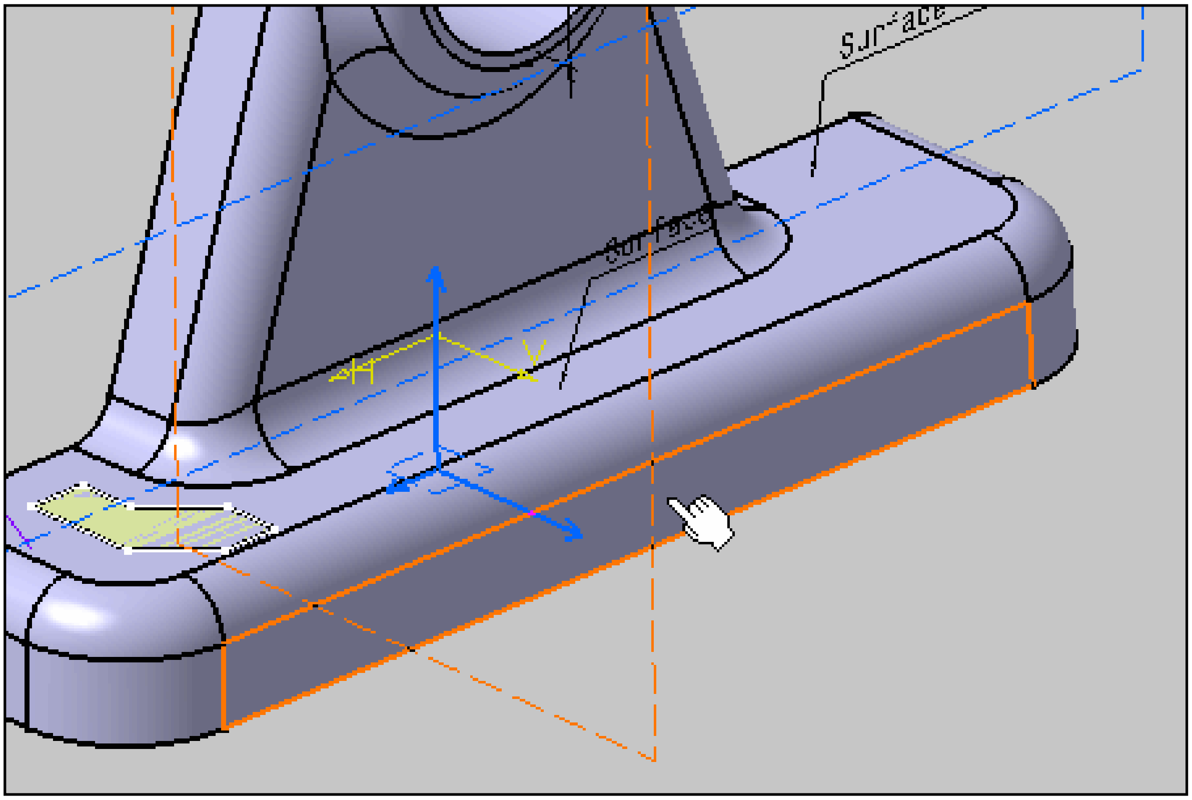
1. Activate the **Projected View.2** annotation plane.



2. Click the **Flag Note with Leader** icon: 

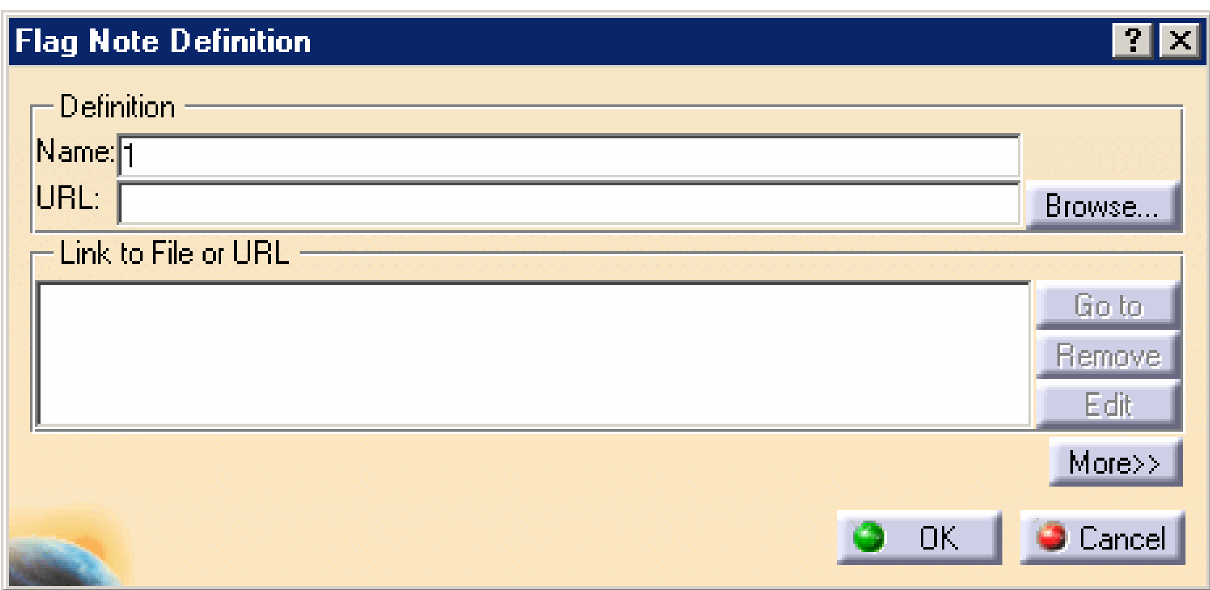
3. Select the face as shown to define a location for the arrow end of the leader.

This scenario illustrates the creation of a flag note by selecting geometry, but you can also select any Part Design or Generative Shape Design feature in the specification tree. In this case, the created annotation will not be attached to the selected feature, but to its geometrical elements at the highest level.

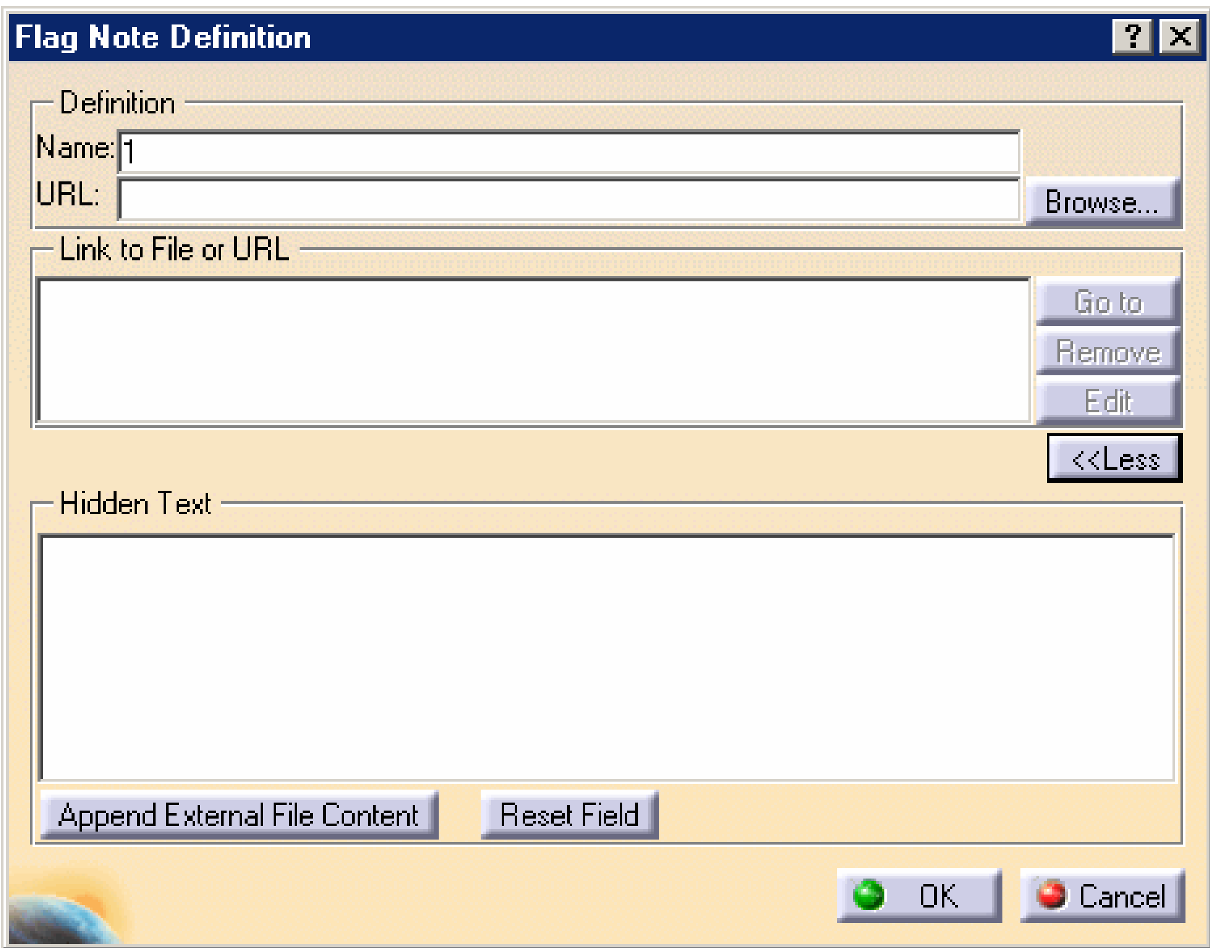


If the active view is not valid, a message appears informing you that you cannot use the active view.
Therefore, the application is going to display the annotation in an annotation plane normal to the selected face.
For more information, see View/Annotation Planes.

The **Flag Note Definition** dialog box appears.



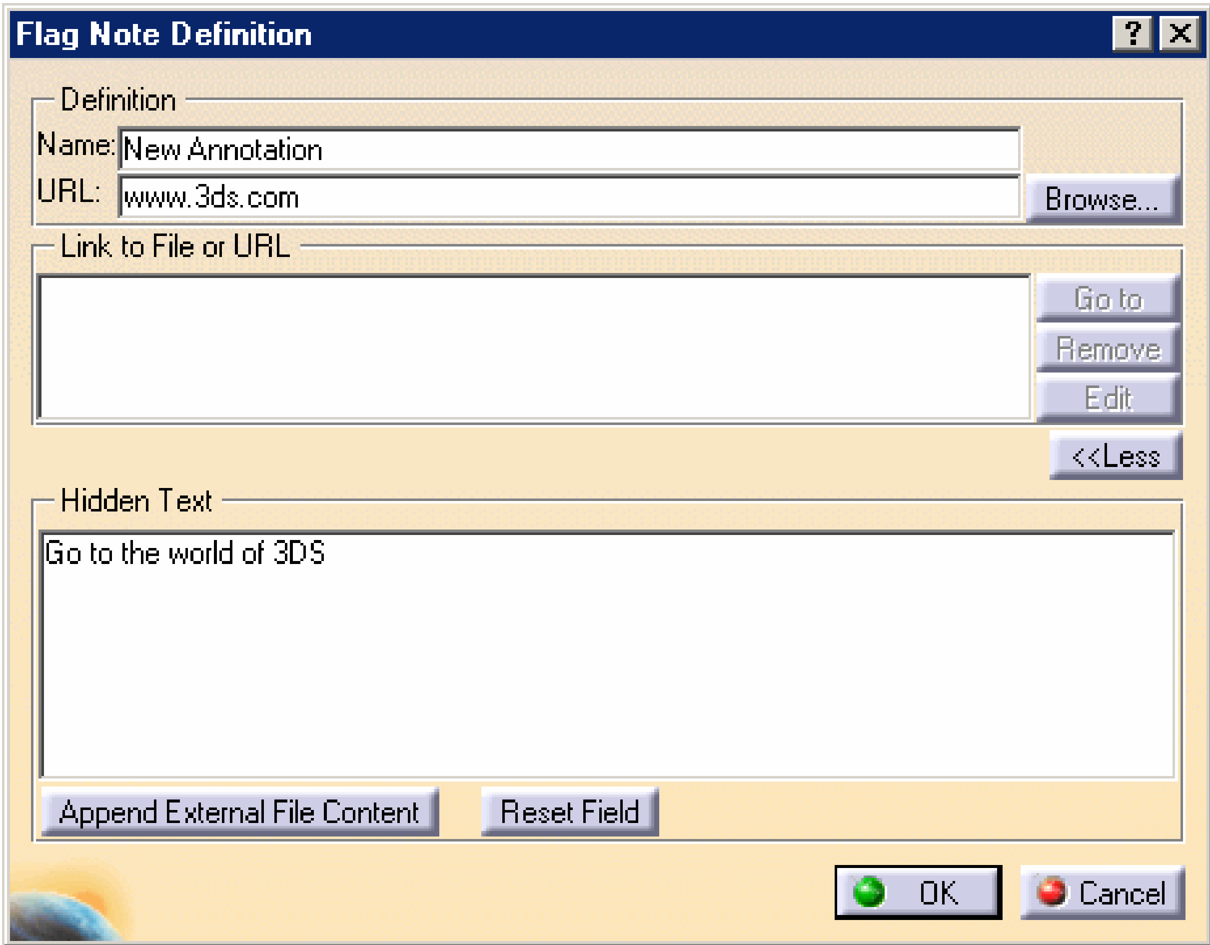
- You may specify the flag note's name link in the Name field.
- You may specify one or several links associated with the flag note in the URL field clicking the **Browse...** button.
- In the **Link to File or URL** list you can see the list of links:
  - To activate one of them, select it and click the **Go to** button.
  - To remove one of them, select it and click the **Remove** button.
  - To edit one of them, select it and click the **Edit** button.
- Clicking the **More>>** button lets you define a hidden text which is displayed with the list of links in a tooltip when the mouse pointer stays on the flagnote
  - You can write your text or import it from a text file.
  - You can leave the field blank.



Links and Hidden text in a Flagnote are not extracted in drawing view.

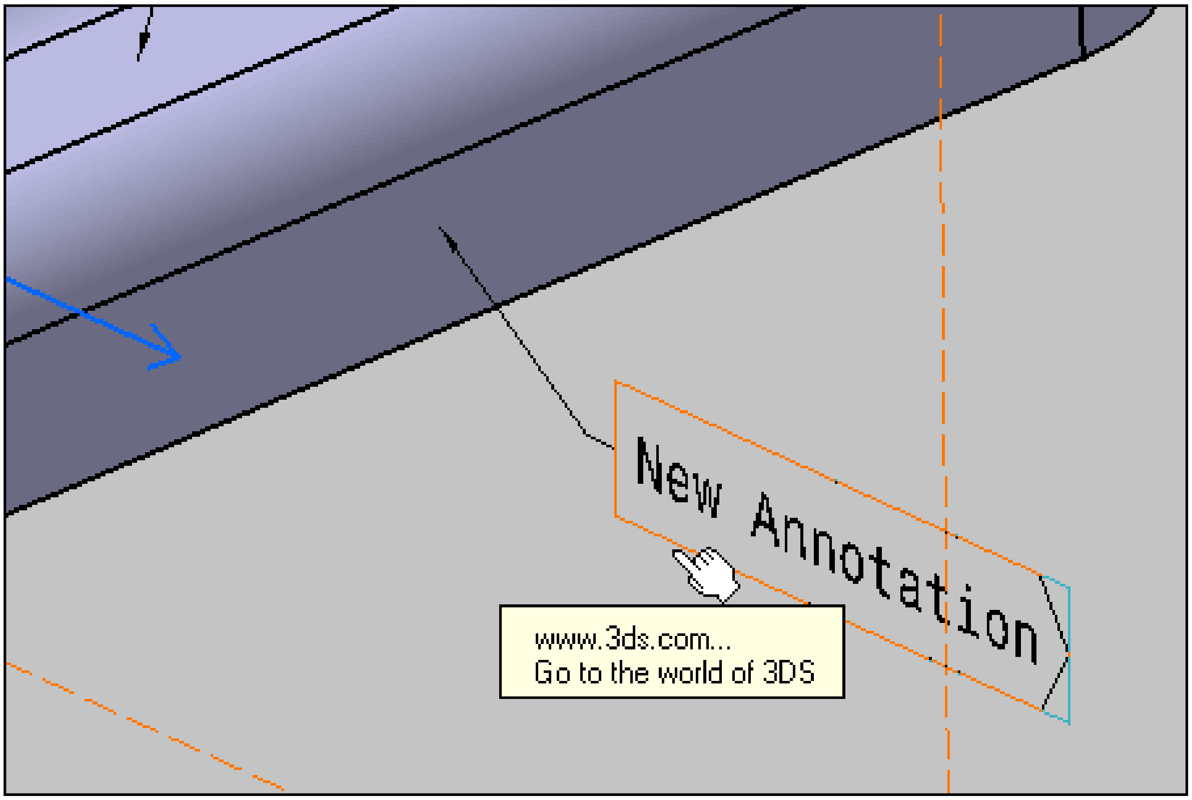
4. Define the flag note:

- the name: New Annotation
- a link: www.3ds.com
- a hidden text: Go to the world of 3DS



5. Click **OK** to end the flag note creation. You can click anywhere in the geometry area too.

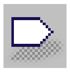
The flag note appears in the geometry, the tooltip containing the URL and the hidden text is displayed when the mouse pointer stays on the flag note.



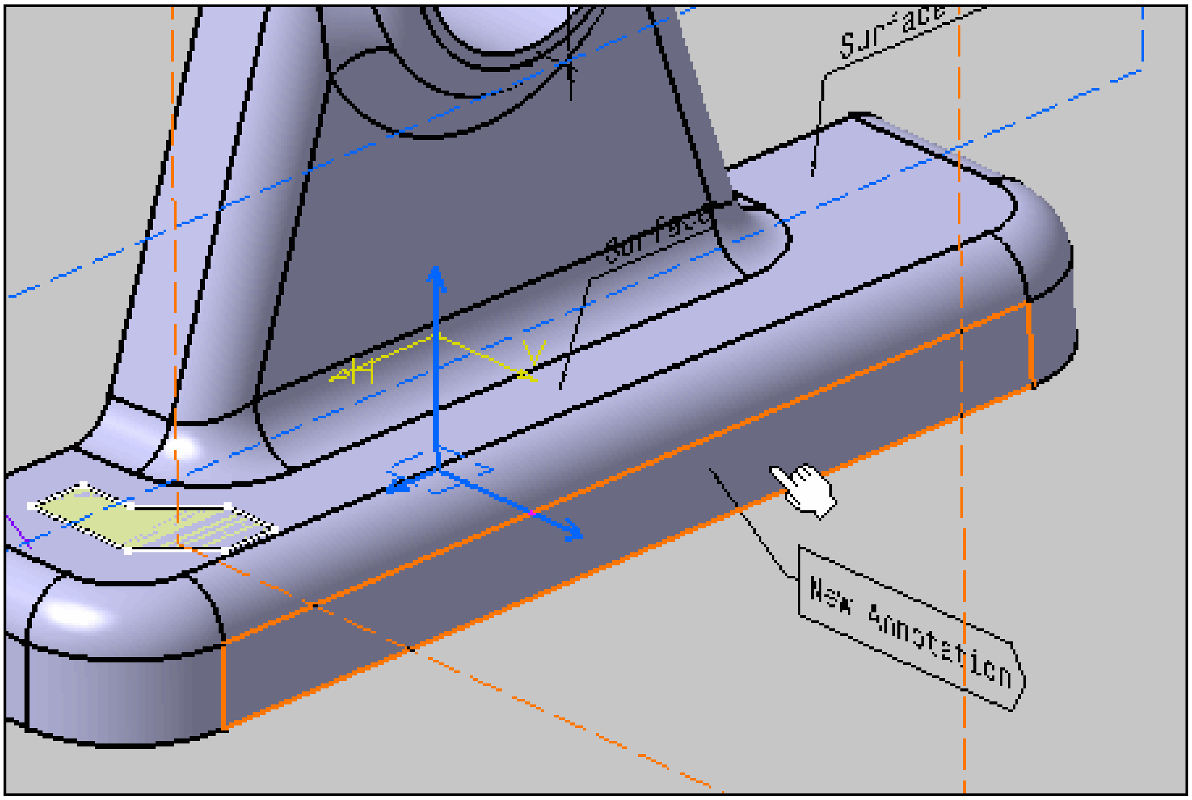
The leader is associated with the element you selected. If you move either the text or the element, the leader stretches to maintain its association with the element.

Moreover, if you change the element associated with the leader, the associativity between the element and the leader is kept.

Note that using the Text Properties toolbar, you can define the anchor point, text size and justification.

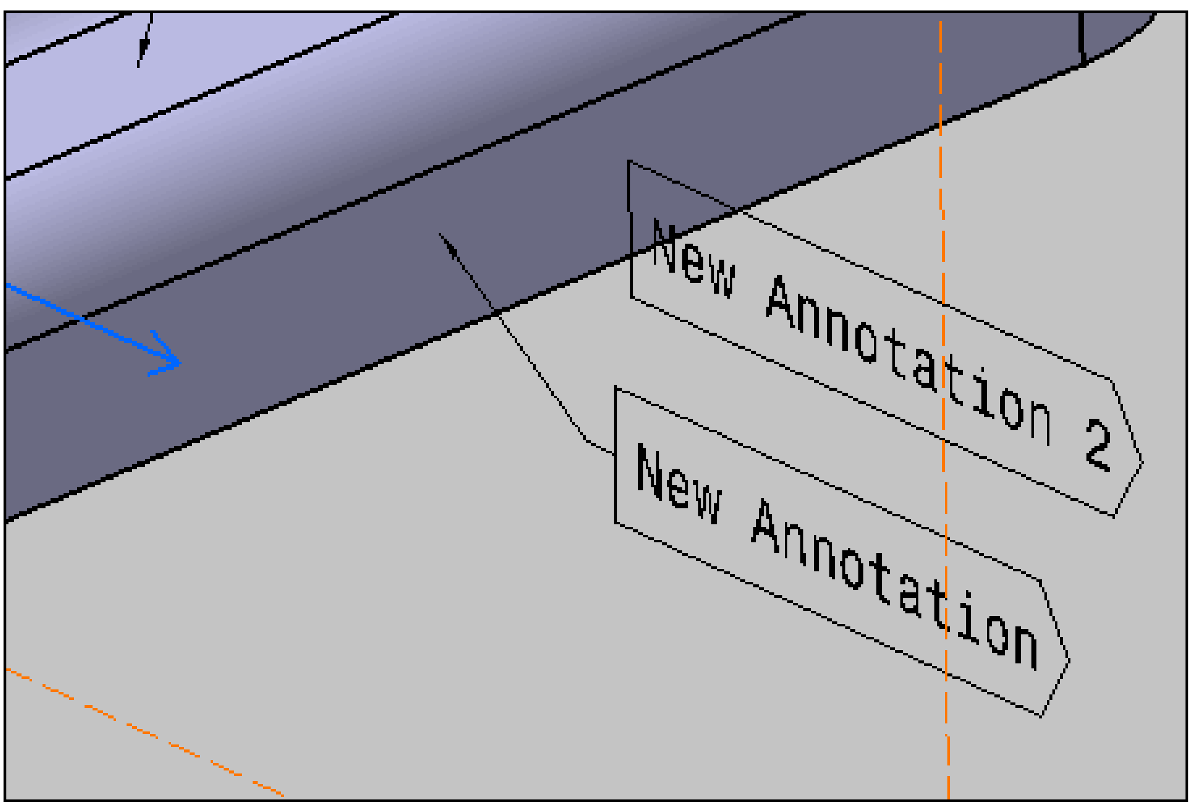
6. Click the **Flag Note** icon: 

7. Select the face as shown.

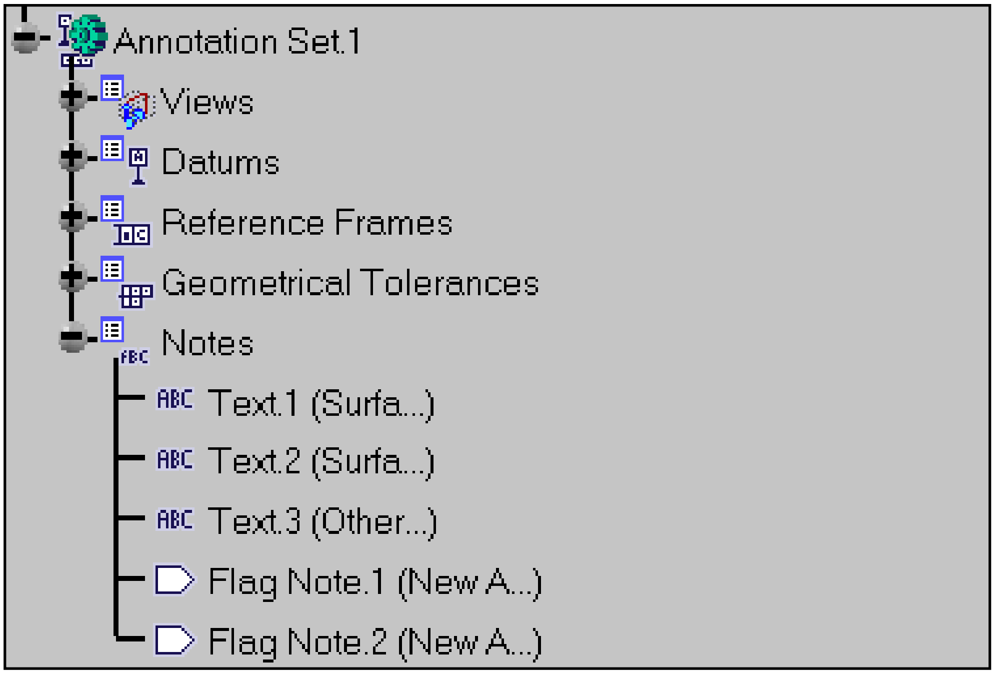


8. Enter your flag note name, for example "New Annotation 2" in the dialog box, specify a link and click **OK**.

The flag note appears in the geometry.



The flag notes (identified as Flag Note.xxx and its name between brackets) are added to the specification tree in the **Notes** group.



You can move a flag note using the drag capability. See Moving Annotations.
Note also that you can resize the manipulators. For more information, refer to Customizing for 3D Functional Tolerancing & Annotations.

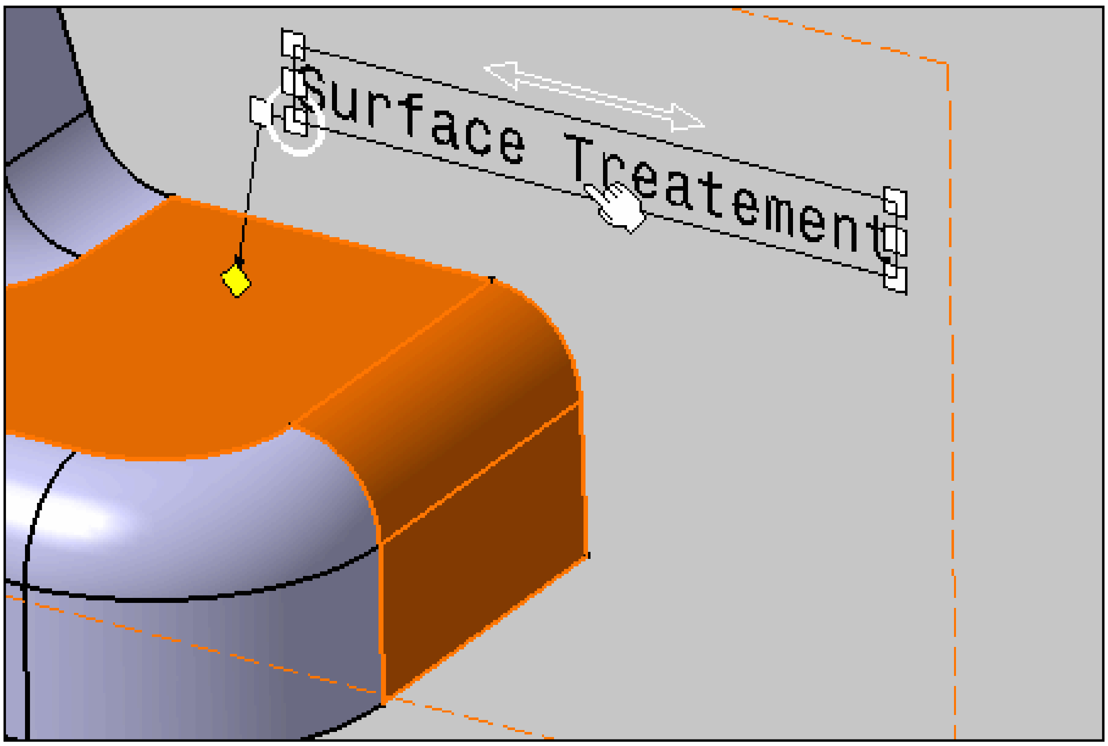
Adding an Attribute Link

This task shows you how to add an attribute link parameter to a text while you are creating this annotation. Note that the operating mode described here is valid for Text or a Flag Note.

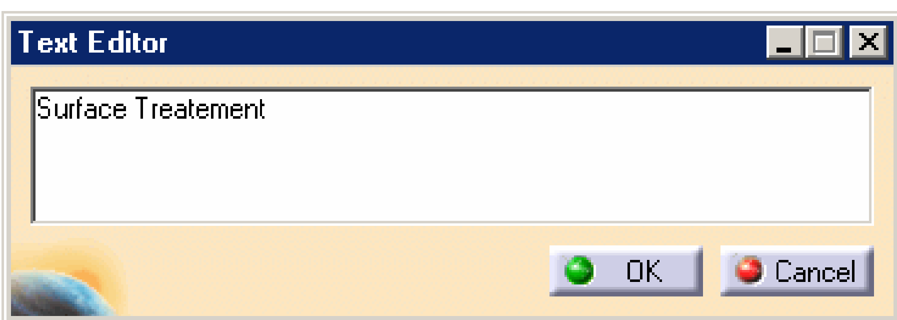
Open the Annotations_Part_04.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

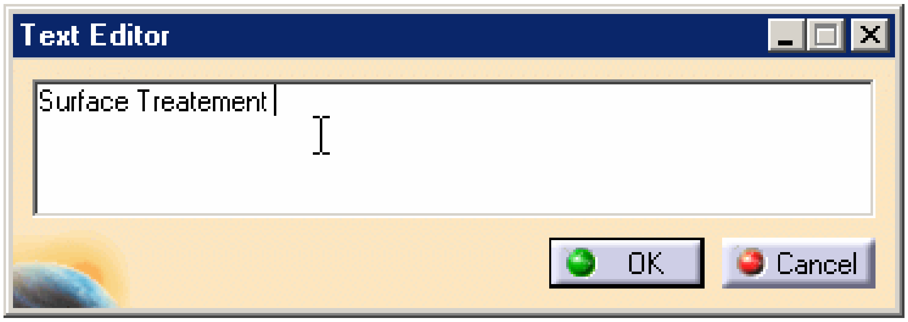
1. Double-click the annotation text to edit it.



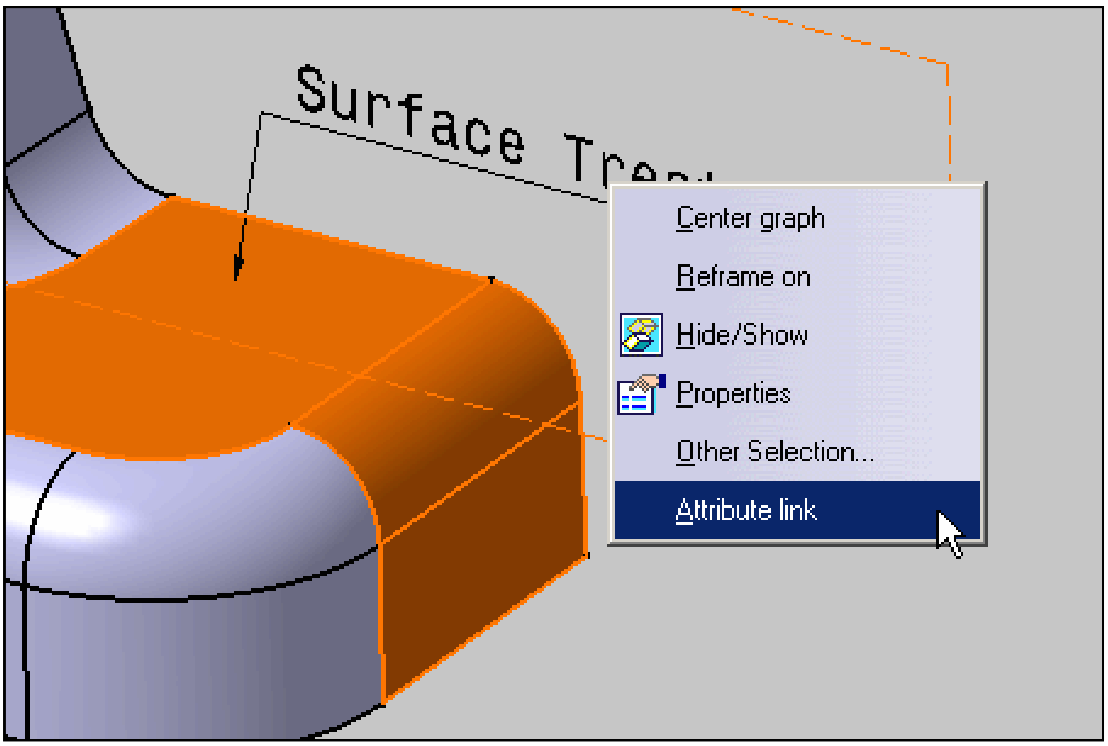
The **Text Editor** dialog box appears.



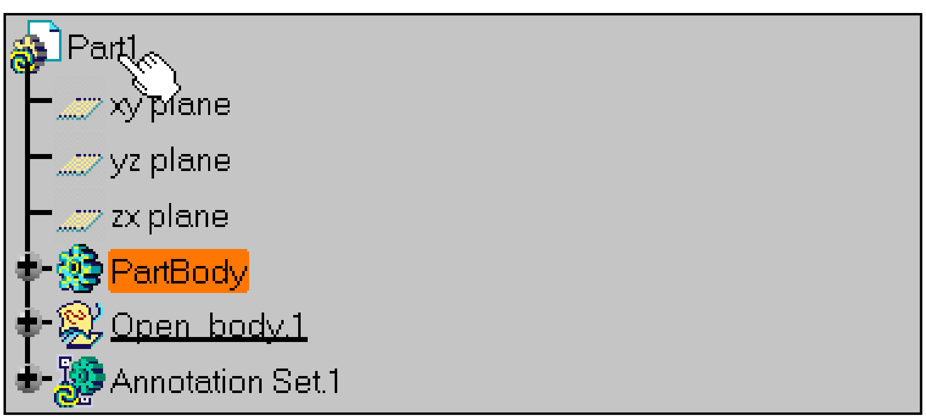
2. Take care to un-select the text into the **Text Editor** dialog box and put the cursor after the white space at the end of the text.



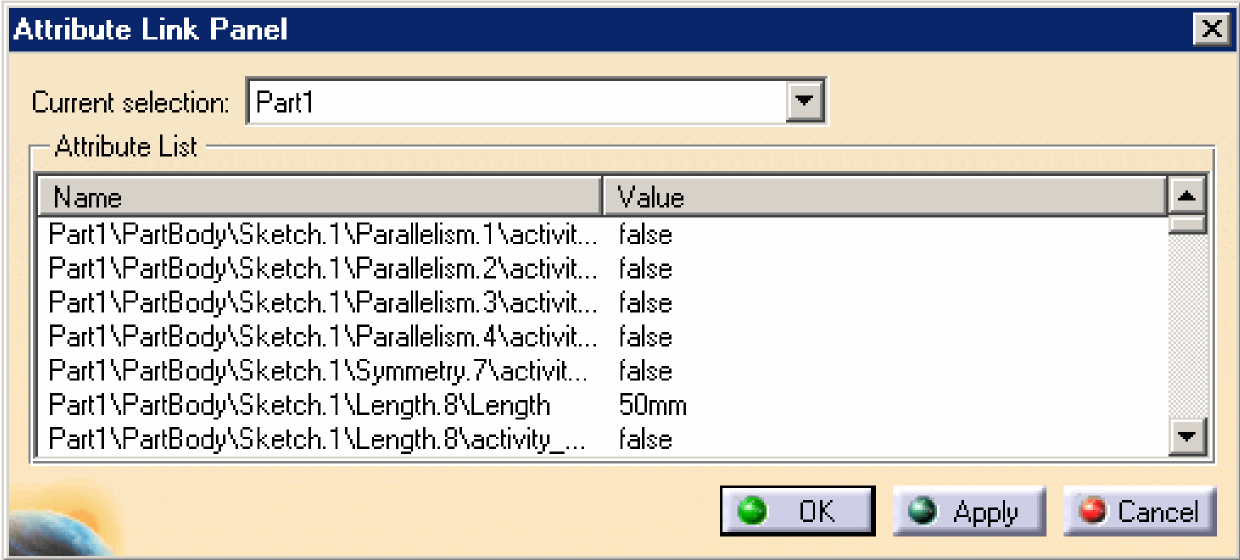
3. Right-click the annotation in the geometry area and select the **Attribute Link** contextual command (the annotation is not highlighted!).



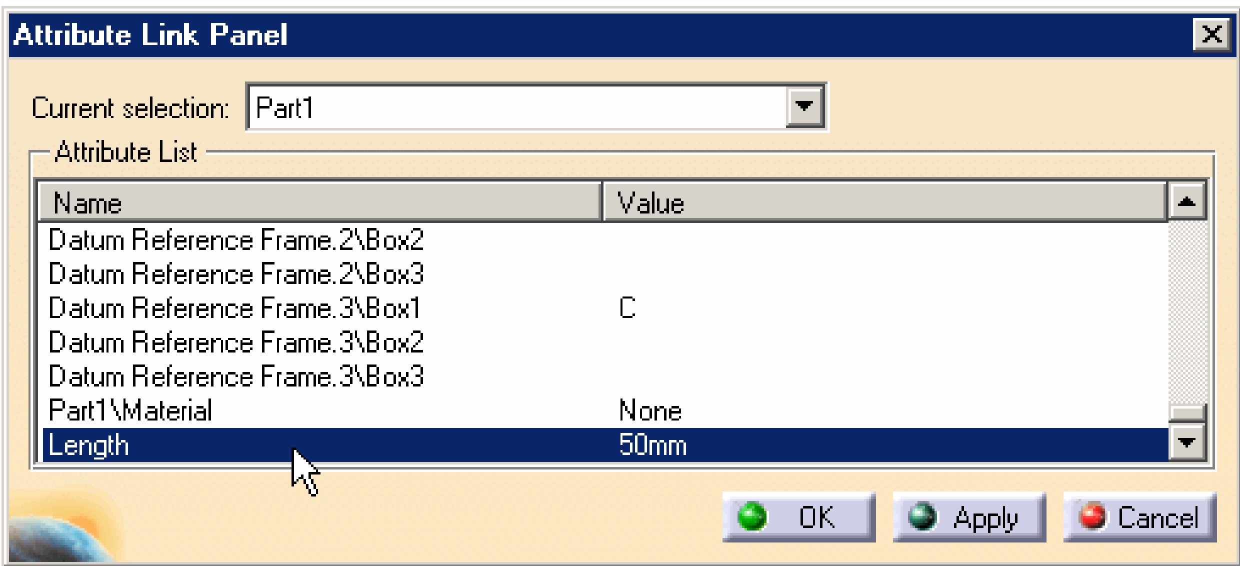
4. Select **Part 1** in the Specification Tree to access all parameters defined for the part.



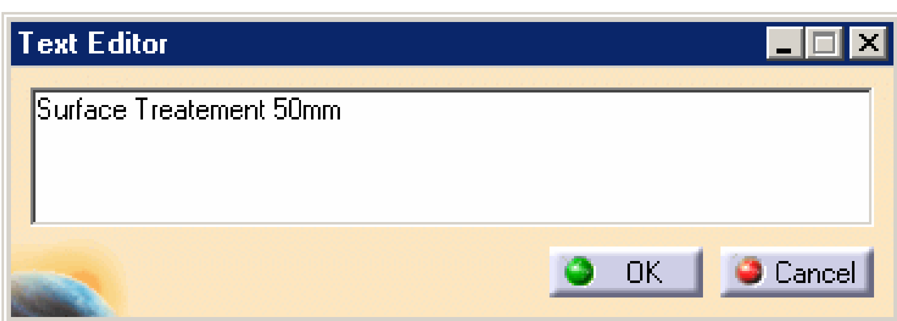
The **Attribute Link** dialog box appears.



5. Select **Length** as the parameter you wish to attach.



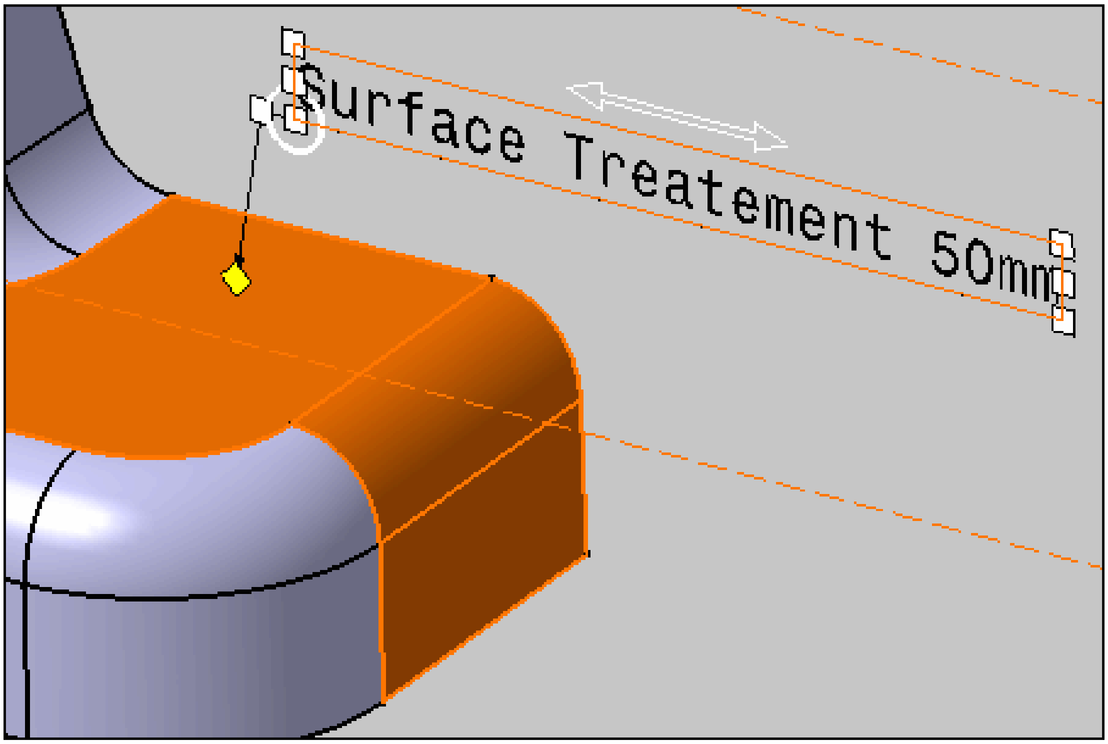
6. Click **OK** to close the **Attribute Link** dialog box.



The value "50mm" now appears both in the **Text Editor** dialog box and in the annotation. You cannot edit this value in this dialog box but you can delete it.

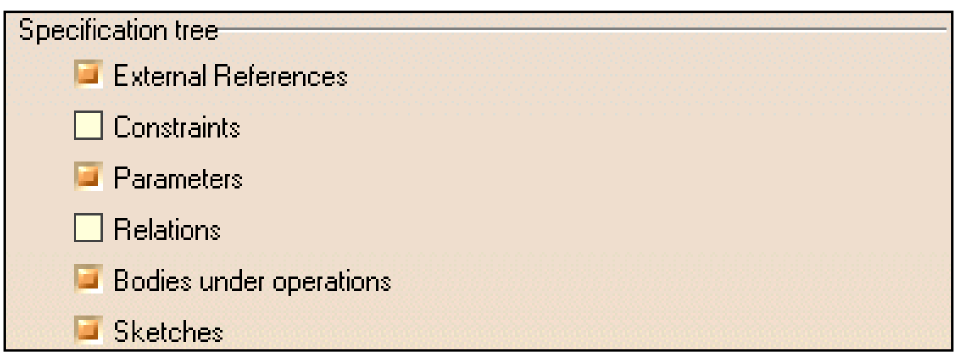
7. Click **OK** to confirm the operation and close the dialog box.

The parameter is attached to the textual annotation.

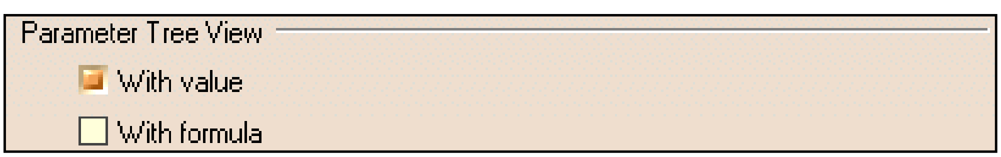


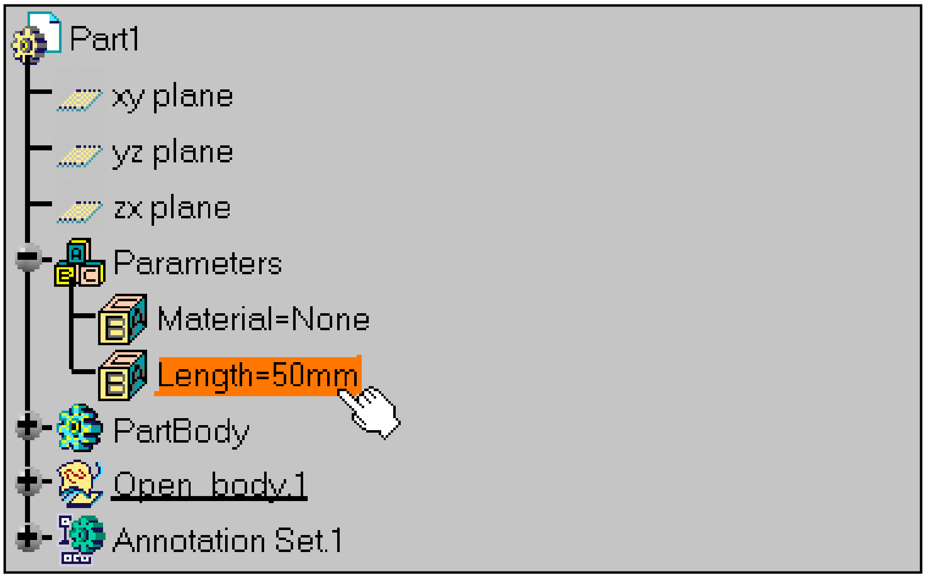
8. Double-click **Length** to edit the parameter: enter 70 mm in the **Edit Parameter** dialog box which appears and click **OK**.

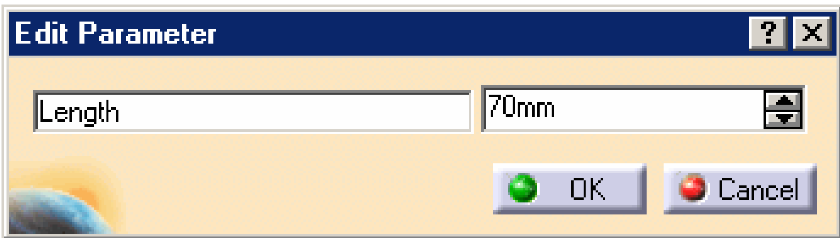
If you wish to display the Parameters node, select the **Tools->Options...** command.
In the **Infrastructure** category, select the **Part Infrastructure** sub-category then the **Display** tab and check the option **Parameters** .



If you wish to display the parameter' value, select the **Tools->Options...** command.
In the **General** category, select the **Parameters and Measure** sub-category then the **Knowledge** tab and check the option **With value**.

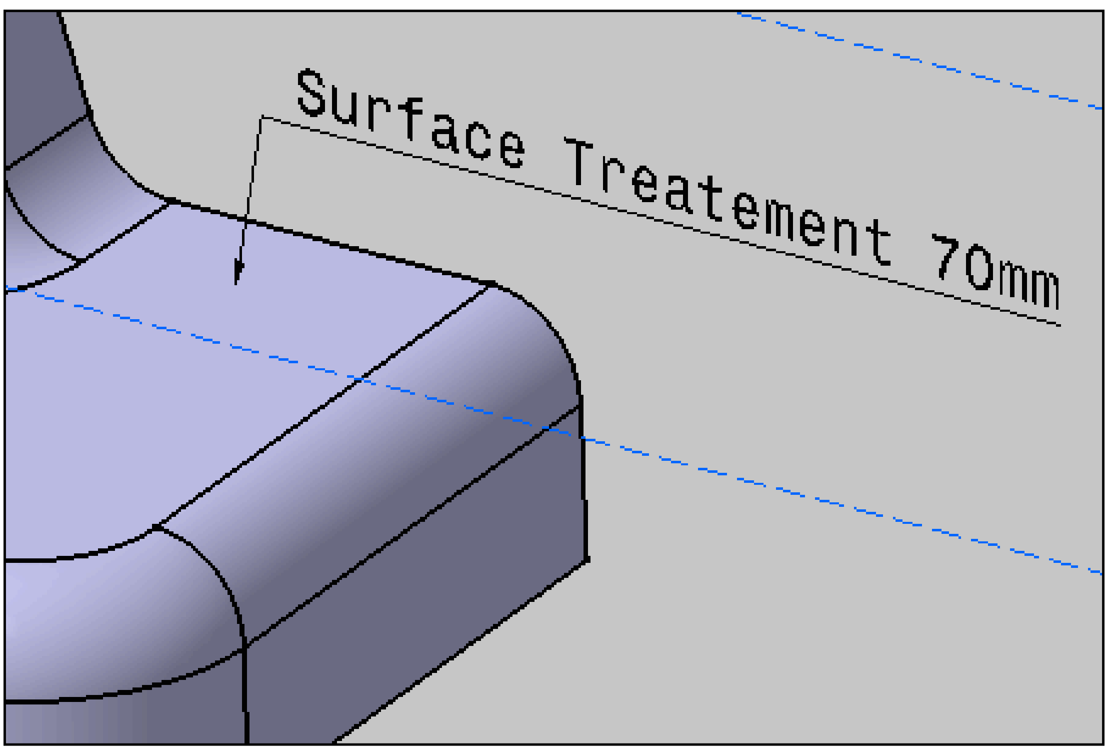






9. Un-select the annotation.

The new value is displayed in the annotation text.



8. If you need to cut the relationship between "70mm" as displayed in the **Text.1** and **Length**, right-click the annotation and select the **Isolate Text** contextual command.
   You can then edit "70mm".

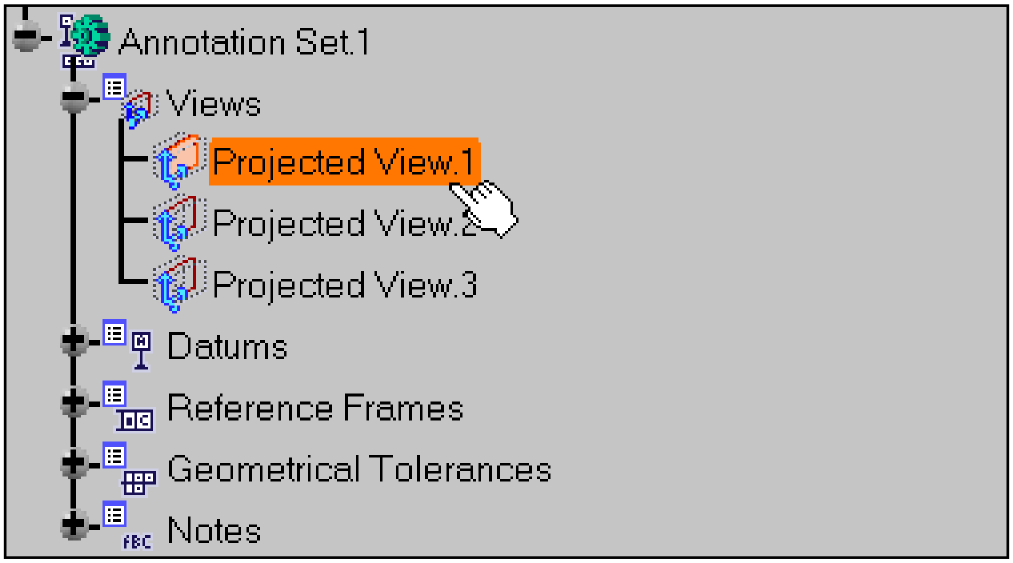
Attaching an attribute to a textual annotation is possible when editing this annotation.

Creating Datums

This task shows you how to specify a datum element.
See Datum Principles for more information.

Open the Annotations_Part_04.CATPart document.

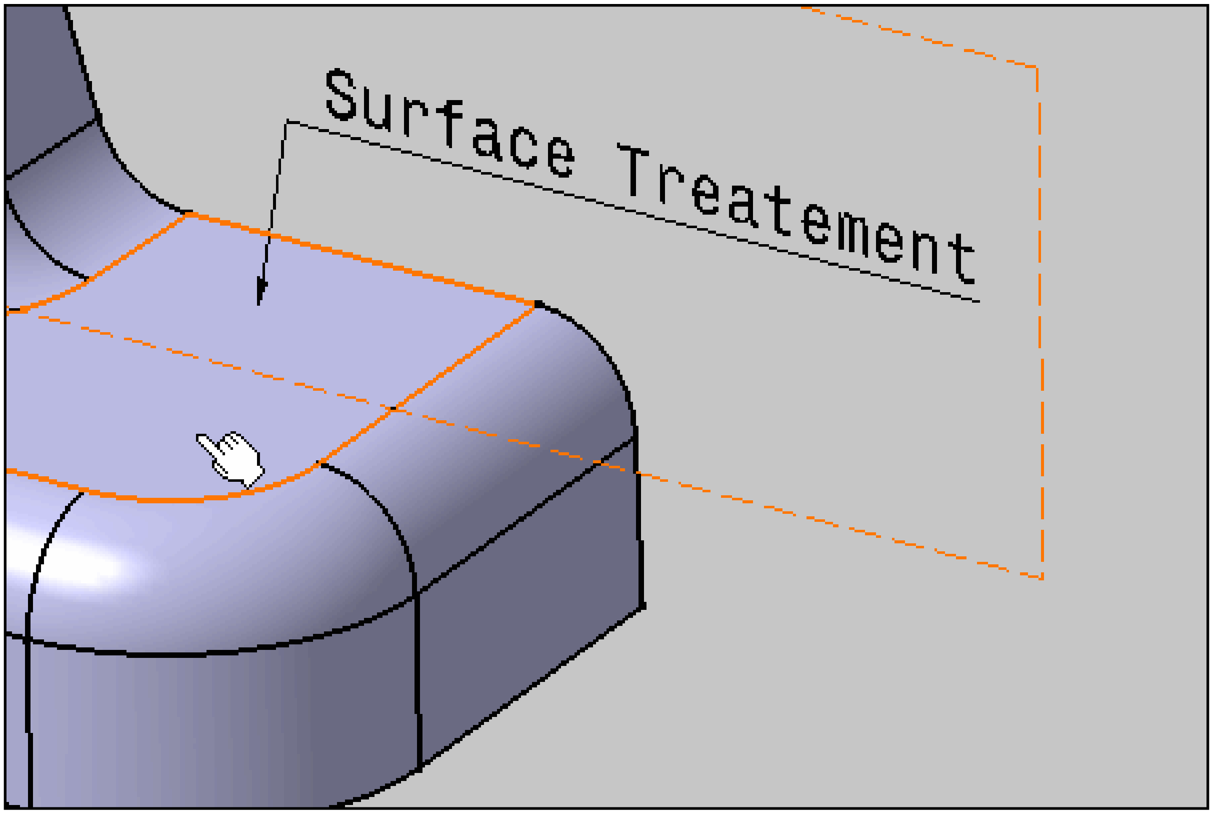
1. Activate the **Projected View.1** annotation plane.



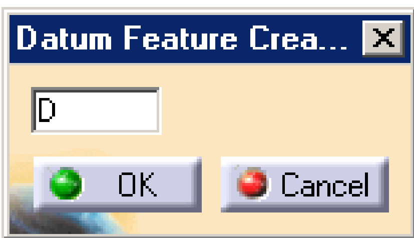
2. Click the **Datum Element** icon: 

3. Select the attachment surface of the datum feature.

This scenario illustrates the creation of a datum element by selecting geometry, but you can also select any Part Design or Generative Shape Design feature in the specification tree. In this case, the created annotation will not be attached to the selected feature, but to its geometrical elements at the highest level.

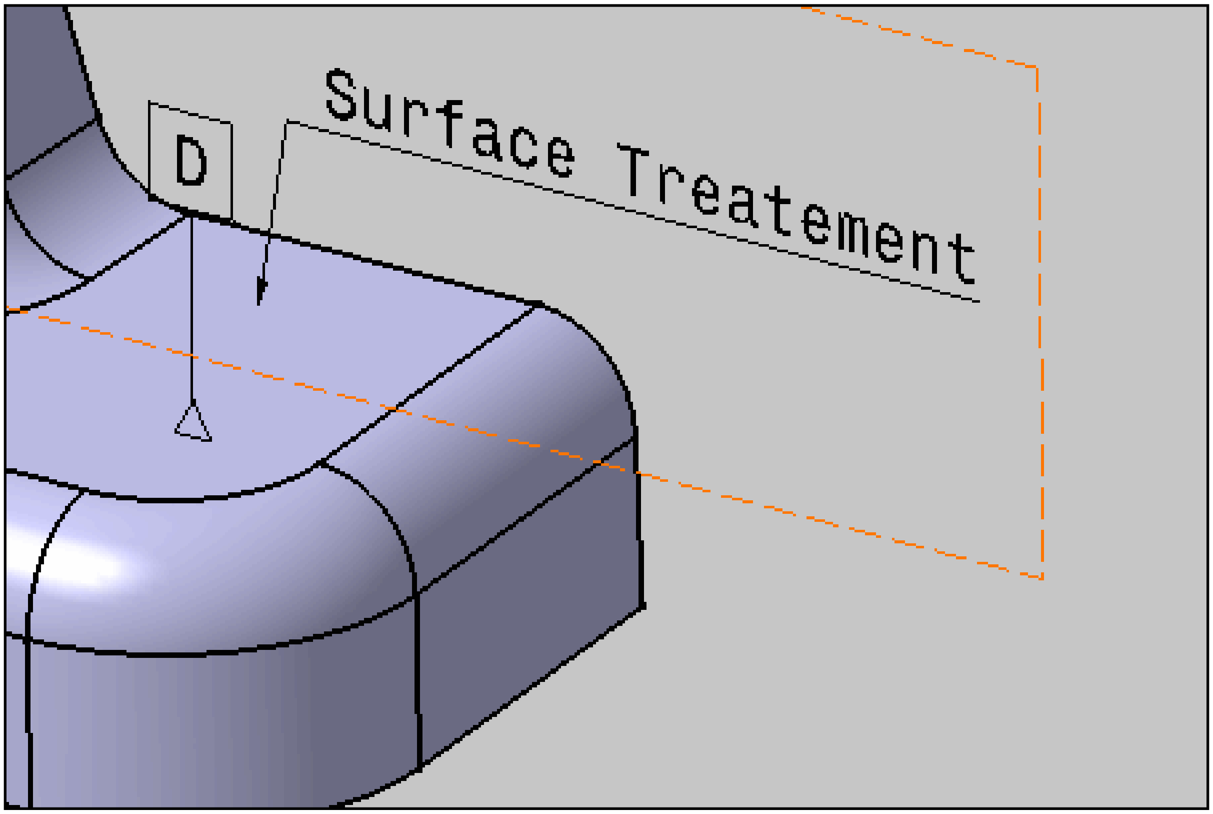


The **Datum Feature** dialog box displays with D as default identifier.



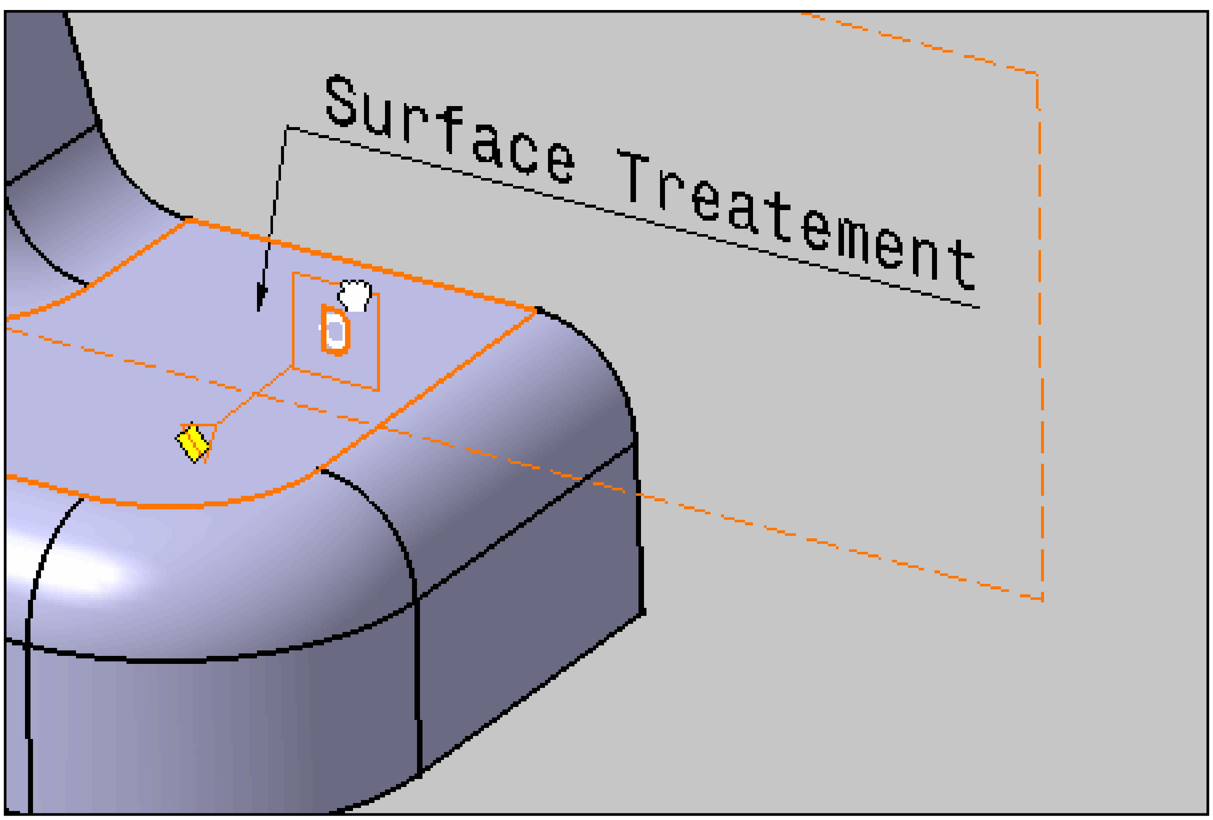
4. Click **OK** to create the datum if the identifier corresponds to your choice.

The datum feature is created in a specific annotation plane.
The "Datum" entity (identified as Simple Datum.xxx) is added to the specification tree.

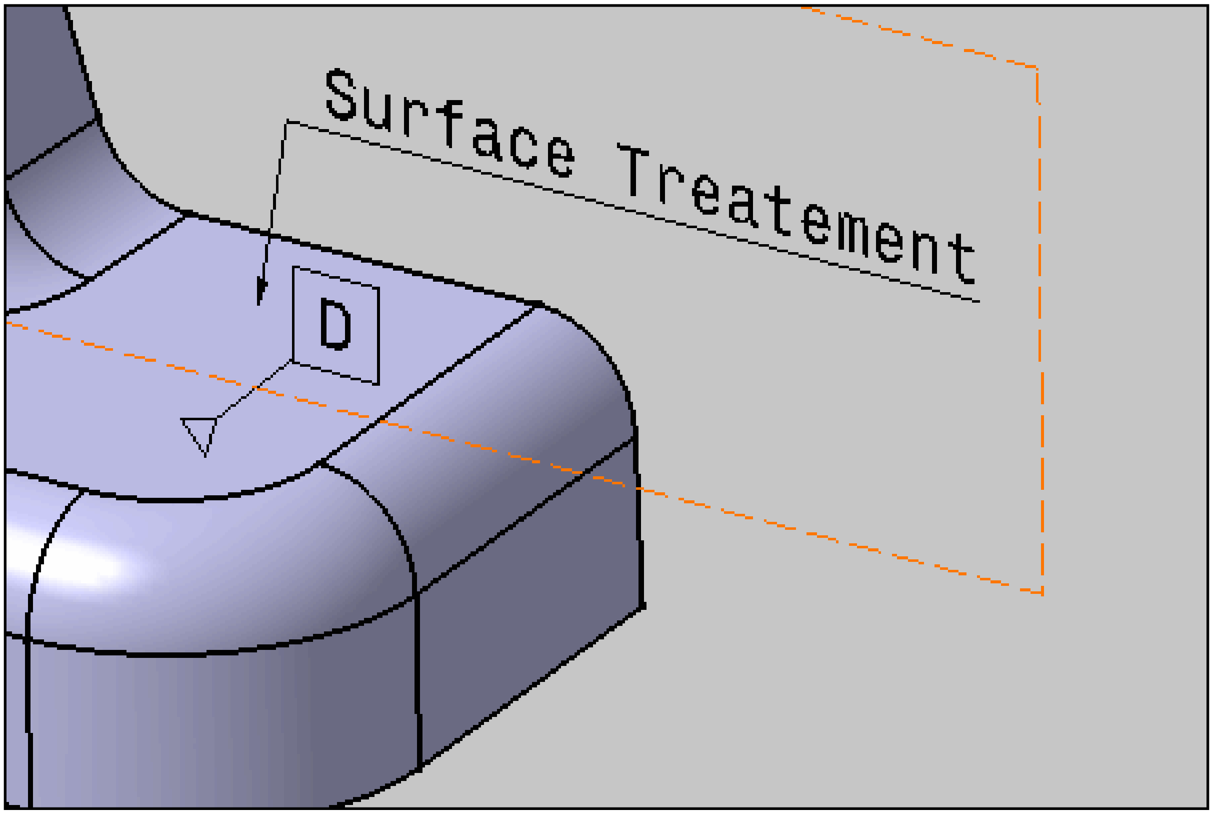


The datum is only a 3D annotation without any semantic link to the geometrical tolerancing.

5. Select the datum and drag it anywhere. You can notice that it remains in the annotation plane.



6. Release the datum.



To edit a datum, double-click the datum, enter the new label in the Datum Feature Modification dialog box that is displayed, and click OK. The modification is simultaneously taken into account.

Two datum elements must not have the same label. A datum label must be unique to ensure that tolerance specifications are consistent.

Creating Datum Targets

This tasks shows you how to specify datum targets on datum elements.

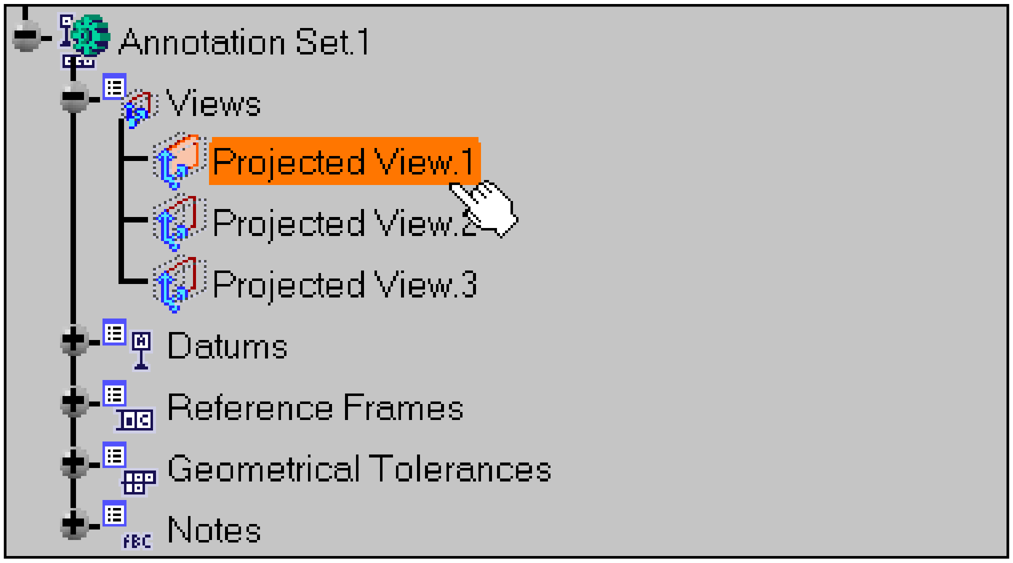
Before performing the task, here are a few principles you should be familiar with:

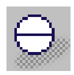
- When defining a datum on planar or cylindrical surfaces the use of datum targets is optional.
- A target element can be a point, line, circular or rectangular surface lying on the datum element:
  - When the datum target is a point, then the circular frame is linked to a cross-placed on the surface. Framed dimensions shall define the location of the point.
  - When the datum target is a line, then the circular frame is linked to a line placed on the surface. Framed dimensions shall define the length and the location of the line.
  - When the datum target area is square or circular, the area dimensions are indicated in the upper compartment of the circular frame, or placed outside and connected to the appropriate compartment by a leader line (when there is no sufficient space within the compartment).
- The minimum number of targets is defined by the datum depending on whether it is used as primary, secondary or tertiary datum in a reference frame.
- For instance, if the datum feature is a cylinder, the targets may be two non-parallel lines, tangent to the cylinder and perpendicular to its centerline, in order to define "equalizing" datum ("V-type-equalizers").
- If the datum element is prismatic or complex, the use of datum targets is mandatory. In this case when selecting targets a message indicates the current step of the datum definition.
- When the datum is established from datum targets, then the letter identifying the surface is repeated on the right side of the datum indicator followed by the list of numbers identifying the targets (separated by comas).
- When there is no sufficient space within the compartment, the dimensions of the datum target area are placed outside the circular frame and connected to the appropriate compartment by a leader line terminated by a dot.

Open the Annotations_Part_04.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

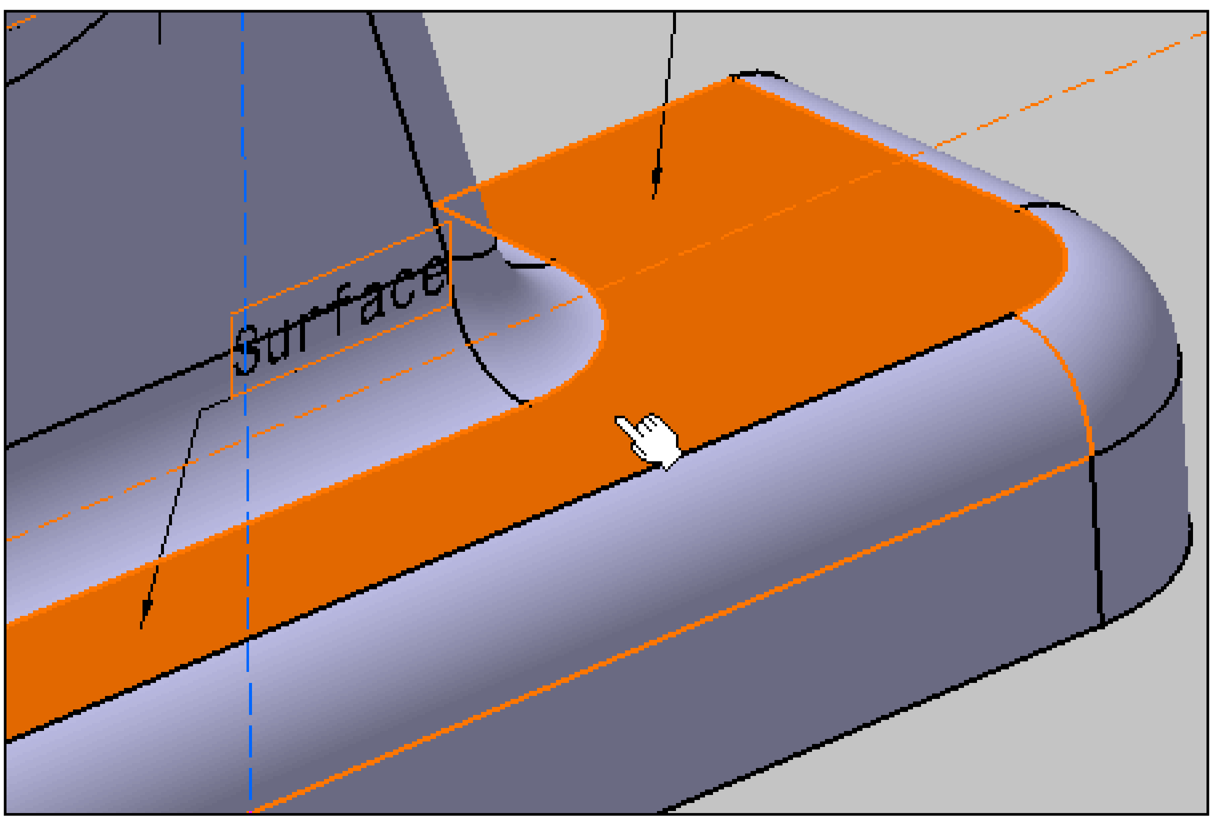
1. Activate the **Projected View.1** annotation plane.



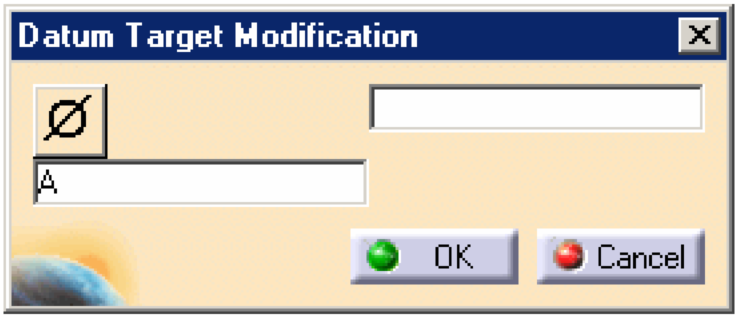
2. Click the **Datum Target** icon: 

3. Select the face as shown.

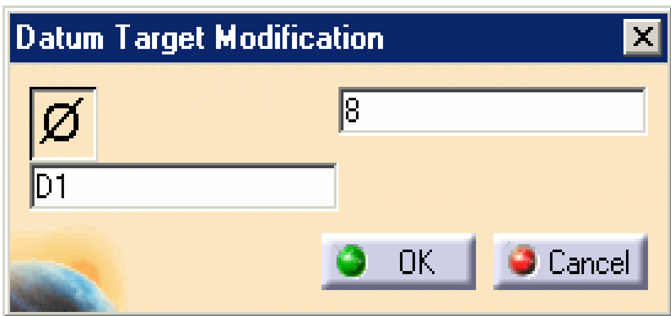
This scenario illustrates the creation of a datum target by selecting geometry, but you can also select any Part Design or Generative Shape Design feature in the specification tree. In this case, the created annotation will not be attached to the selected feature, but to its geometrical elements at the highest level.



The **Datum Target Modification** dialog box appears.

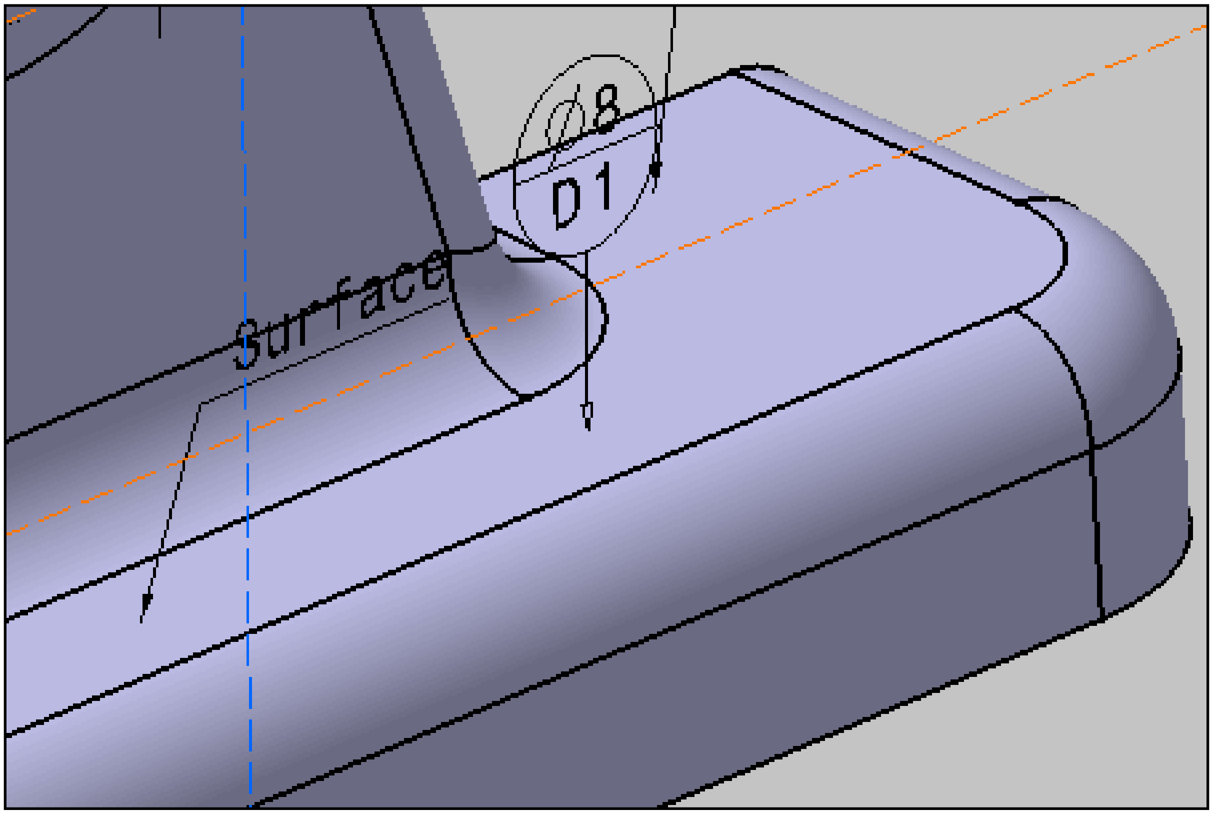


4. Click the diameter icon in the **Datum Target Modification** dialog box, enter 8 in the field opposite, enter D1 in the field to the left.



5. Click **OK** to validate.

You have created a datum target on datum plane D. The datum target corresponds to a 8mm-diameter surface. Its name is "D1" and it is identified as "Target..xxx" in the specification tree.


Ø8
D1
Surface

Creating Geometrical Tolerances

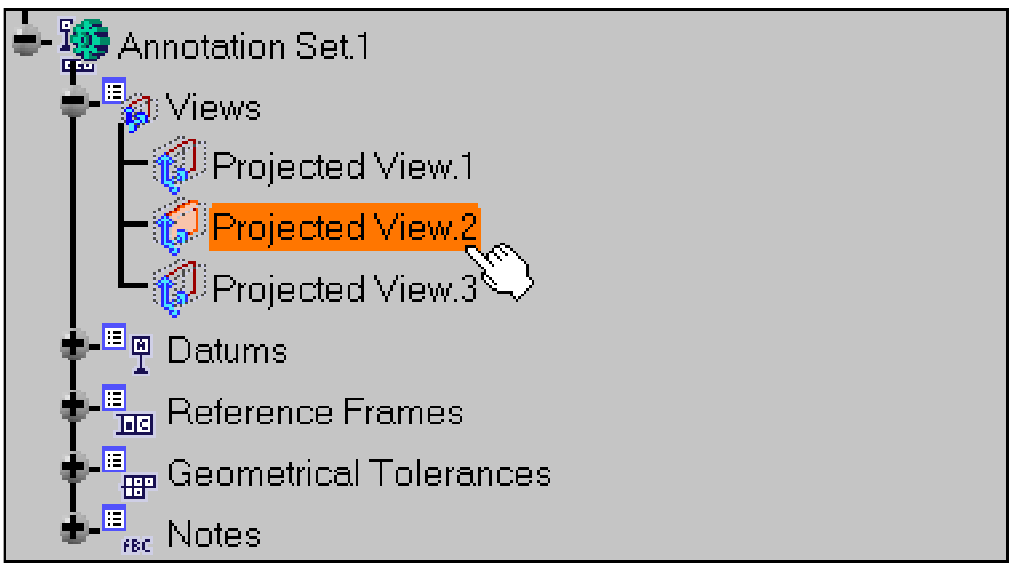
This task will show you how to create a geometrical tolerance annotation.

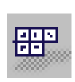
Before performing the task, here are a few principles you should be familiar with:

- Principles and Fundamental Rules for Geometrical Tolerancing
- Geometric Tolerancing
- Symbols for Geometrical Tolerances
- Symbols for Modifiers

Open the Annotations_Part_04.CATPart document.

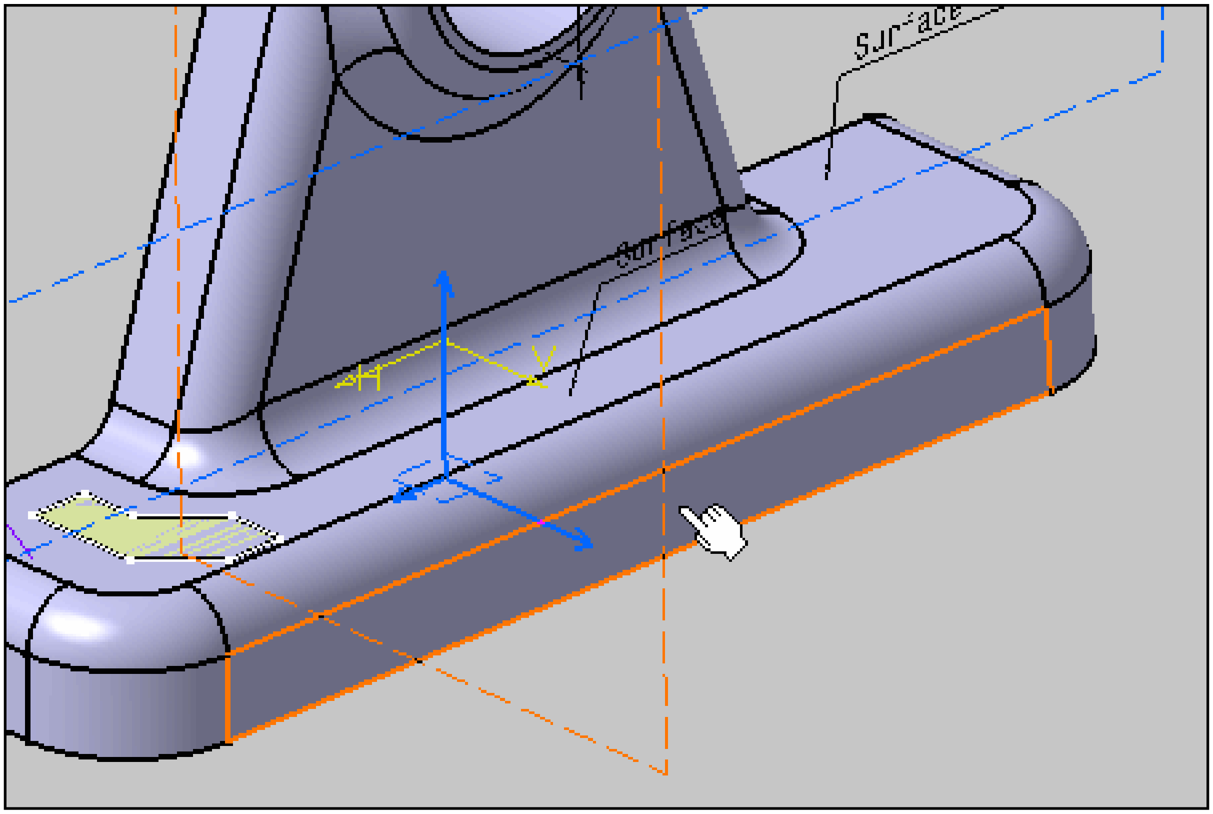
1. Select the **Projected View.2** annotation plane.



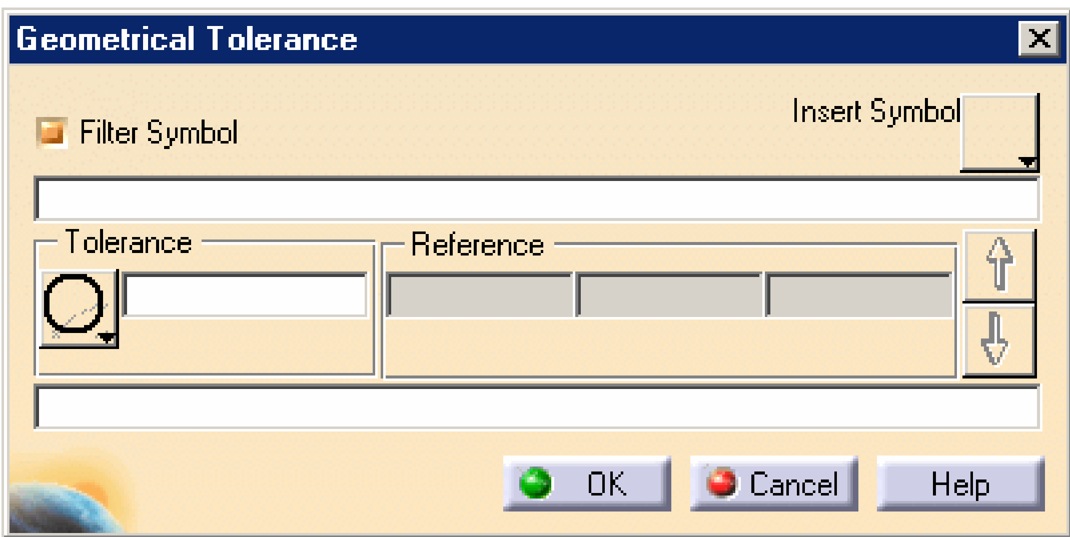
2. Click the **Geometrical Tolerance** icon: 

3. Select the face as shown.

This scenario illustrates the creation of a geometrical tolerance by selecting geometry, but you can also select any Part Design or Generative Shape Design feature in the specification tree. In this case, the created annotation will not be attached to the selected feature, but to its geometrical elements at the highest level.



The **Geometrical Tolerance** dialog box appears.



This dialog box allows you to:

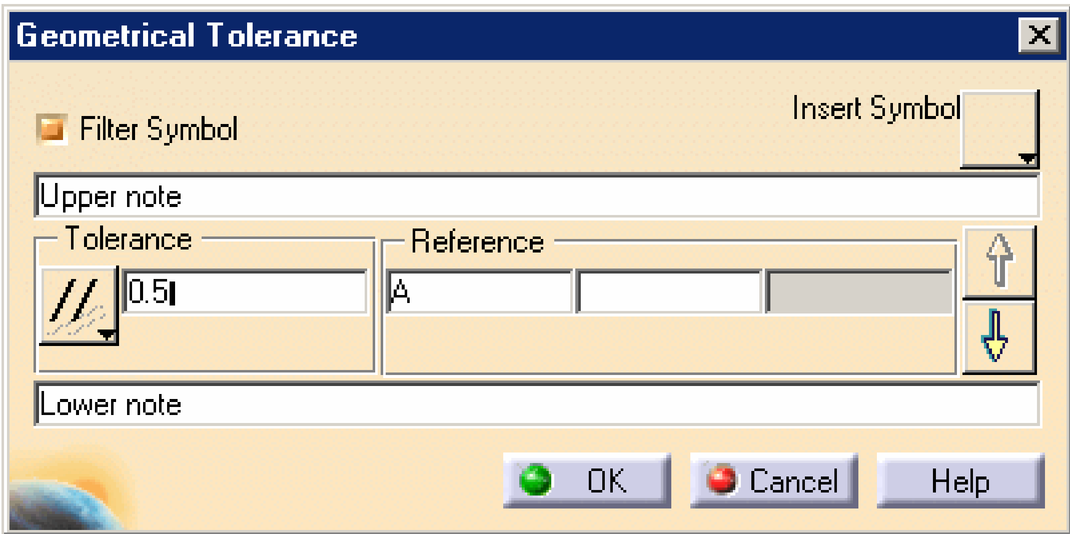
- Specify as many specification lines as you want (with the Up and Down arrows).
- Insert several modifiers anywhere in a tolerance or a reference.
- Add notes upper and lower the set of specification.

3. Set the parallelism symbol to define the tolerance.

4. Enter the value of the tolerance: **0.5** and insert the **Least Material Condition** symbol modifier.

5. Enter **A** as reference.

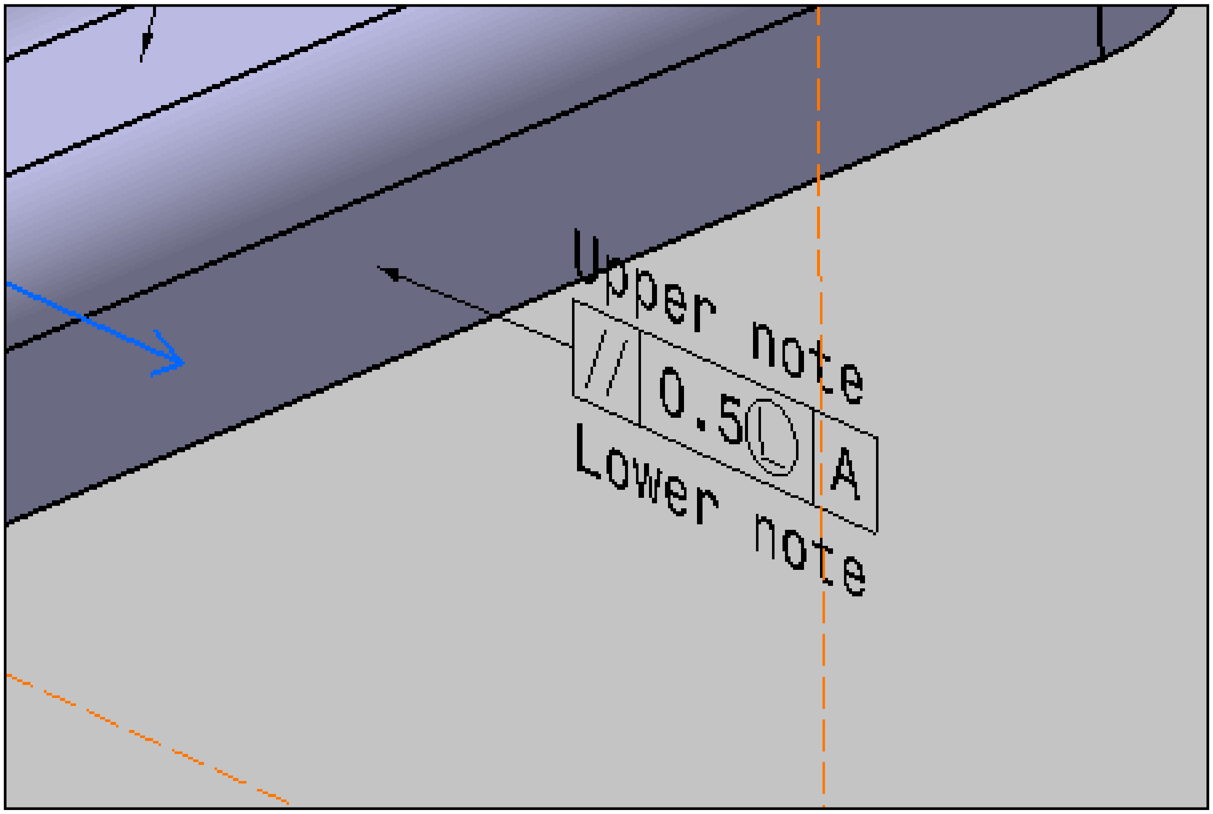
6. Specify the upper and lower notes.



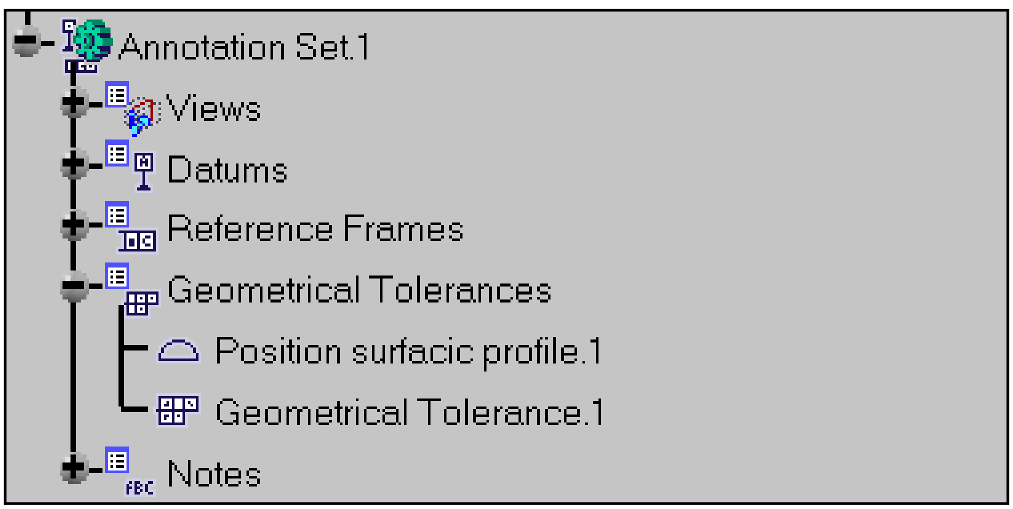
Modifiers are not displaying in tolerance and reference fields and appear with a "|" character.

7. Click **OK** to confirm the operation and close the dialog box.

The geometrical tolerancing annotation is attached to the part.



The geometrical tolerance entity (identified as Geometrical Tolerance.xxx) is added to the specification tree in the **Geometrical Tolerances** group.

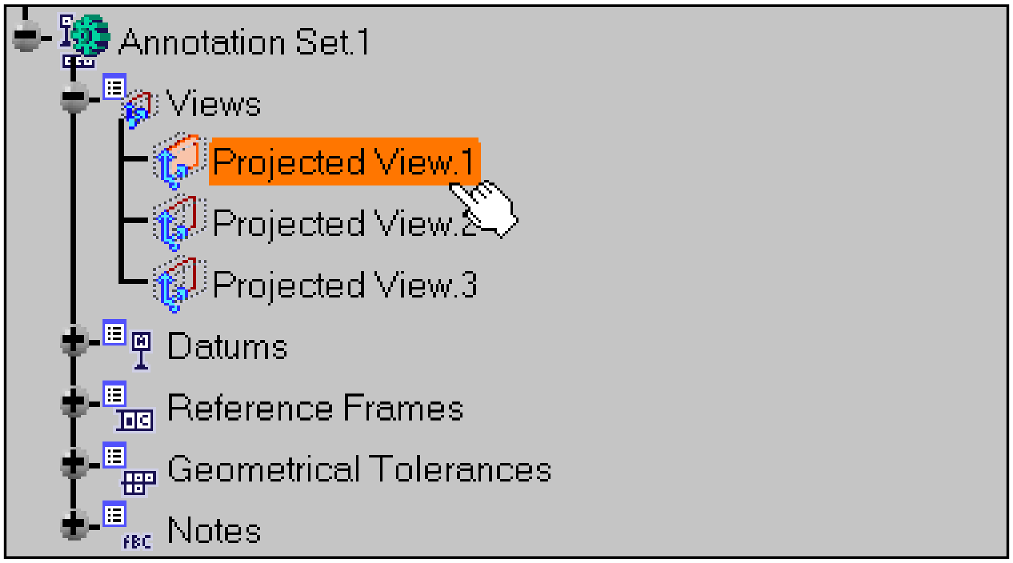


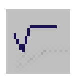
Creating Roughness Symbols

This task shows you how to create a roughness symbol annotation.

Open the Annotations_Part_04.CATPart document.

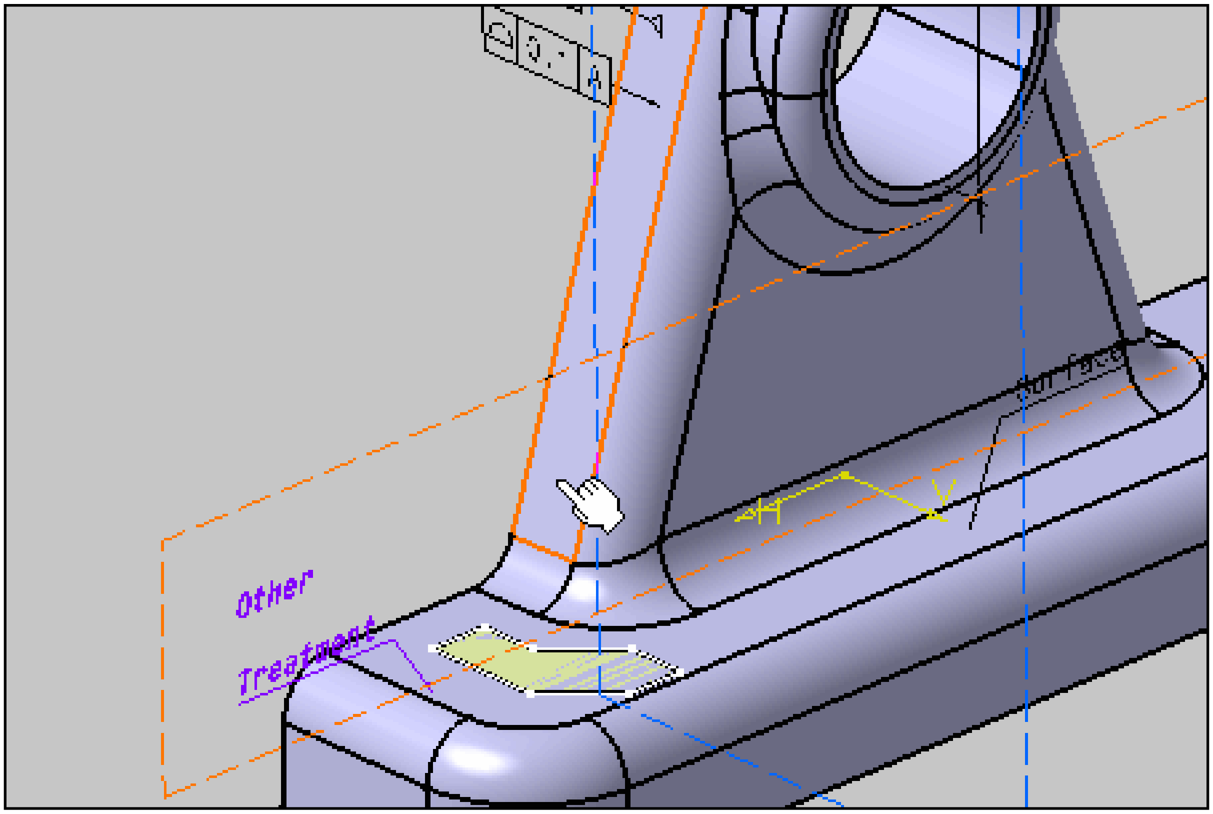
1. Select the **Projected View.1** annotation plane.



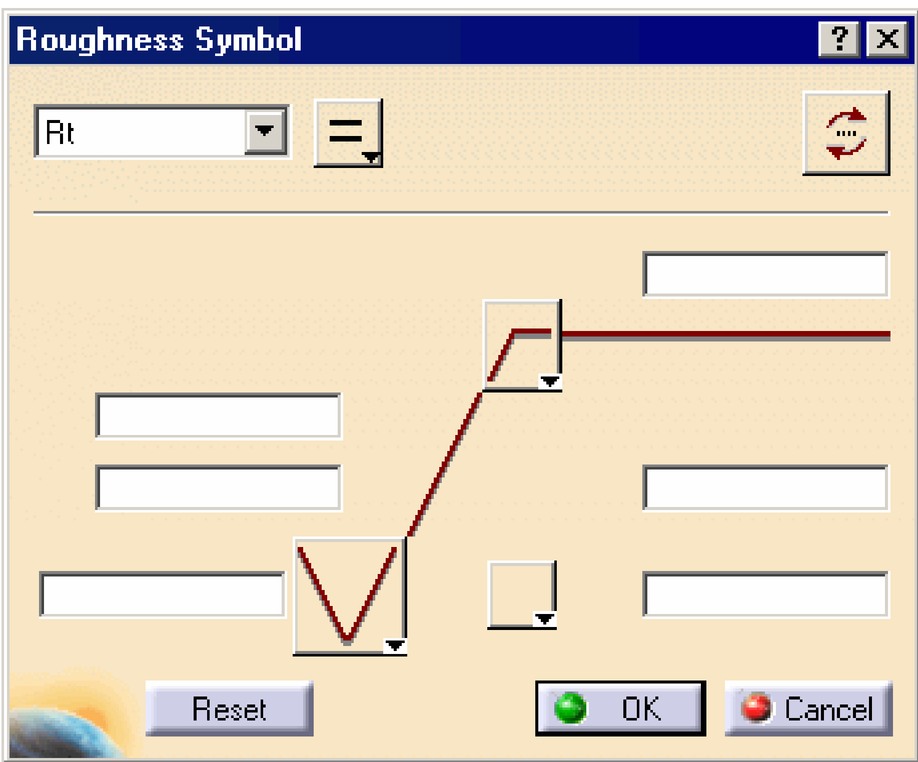
2. Click the **Roughness** icon: 

3. Select the surface as shown on the part.

This scenario illustrates the creation of a roughness symbol by selecting geometry, but you can also select any Part Design or Generative Shape Design feature in the specification tree. In this case, the created annotation will not be attached to the selected feature, but to its geometrical elements at the highest level.



The **Roughness Symbol Editor** dialog box appears.



The new roughness symbol allows you to respect the following standard parameters:

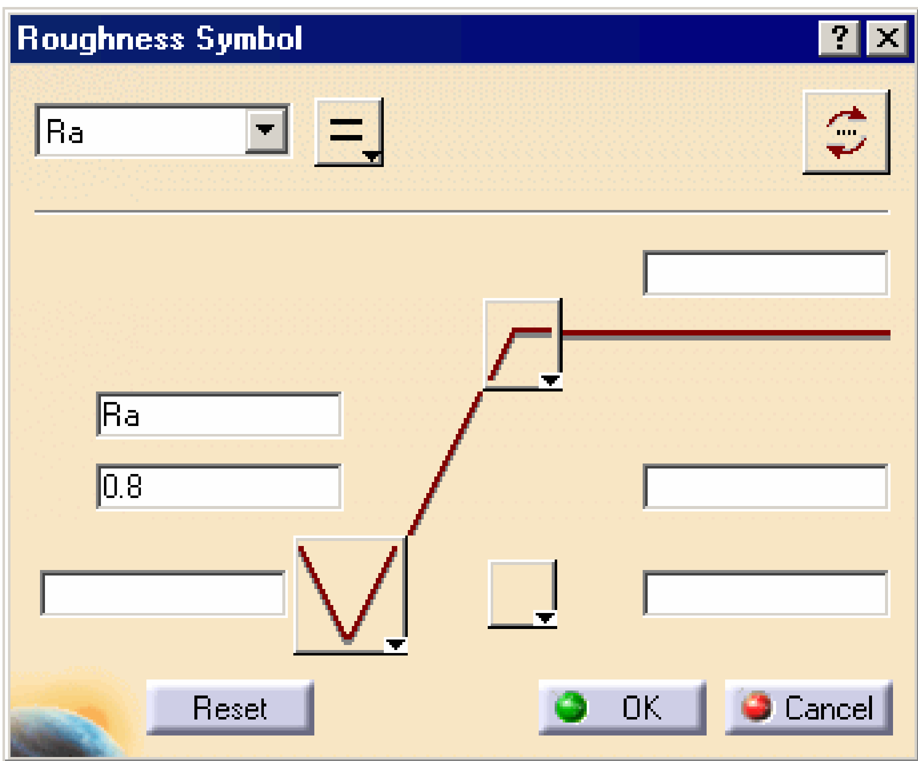
- ISO
- ASME
- ANSI
- JIS

You can modify these the standard parameters using the **Tools -> Standards...** command, then selecting the **Drafting** category.

The old roughness symbols created in previous document are migrated at edition.

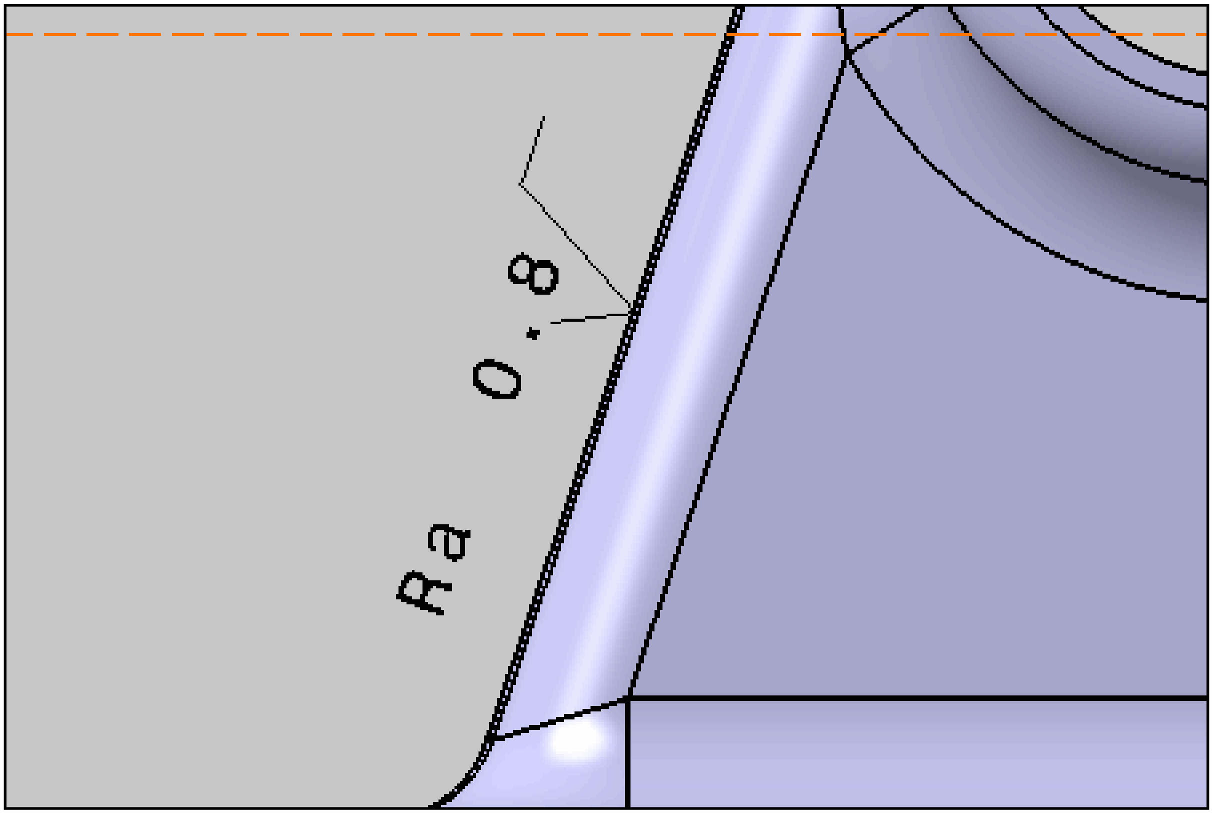
If you slide a new roughness symbol outside its reference, an extension line is added. Its overrun is fixed to 2mm and cannot be modified.

4. Select **Ra** symbol and enter a value: 0.8



5. Click **OK**.

The roughness symbol is created without a leader. To add one, see Adding Leaders and Using Breakpoints.


Ra 0.8

Creating Isolated Annotations

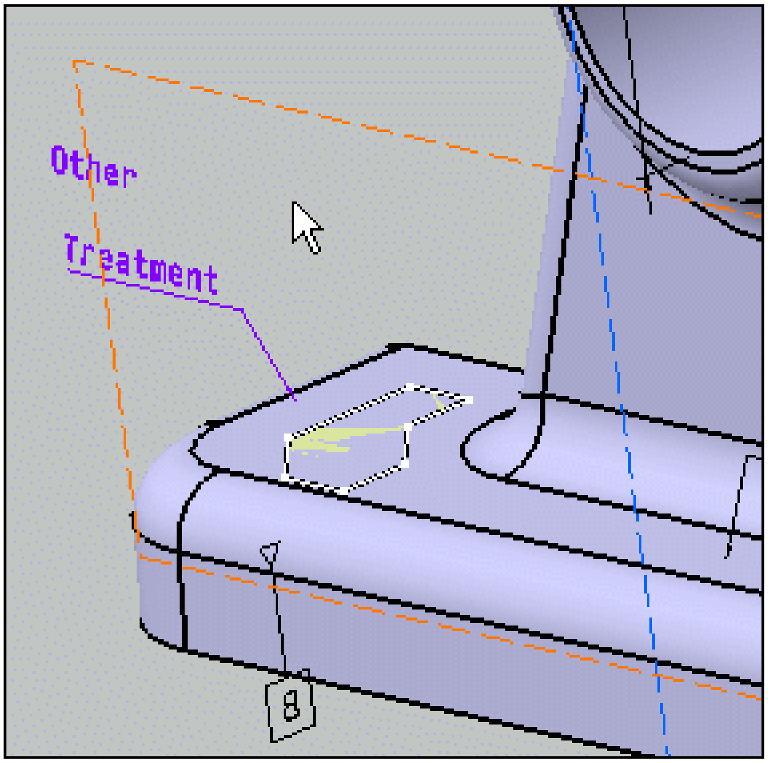
This task shows you how to create isolated annotations, i.e. annotations that are not linked to any geometry.

You can create the following types of isolated annotations:

- Text
- Text with Leader
- Text Parallel to Screen
- Flag Note
- Flag Note with Leader
- Datum Element
- Datum Target
- Geometrical Tolerance
- Roughness
- Note Object Attribute (Instantiated from Catalog Browser)

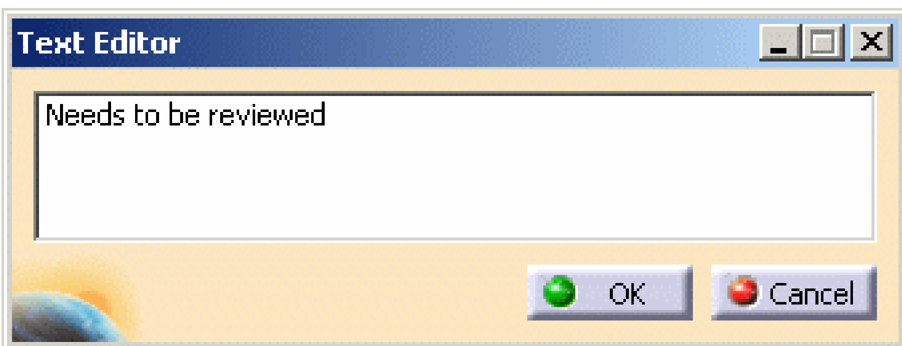
Open the Annotations_Part_04.CATPart document.

1. Activate the **Projected View.1** annotation plane.
2. Click an annotation creation icon, for example the **Text** icon:
3. Click anywhere in the free space (not on the geometry).

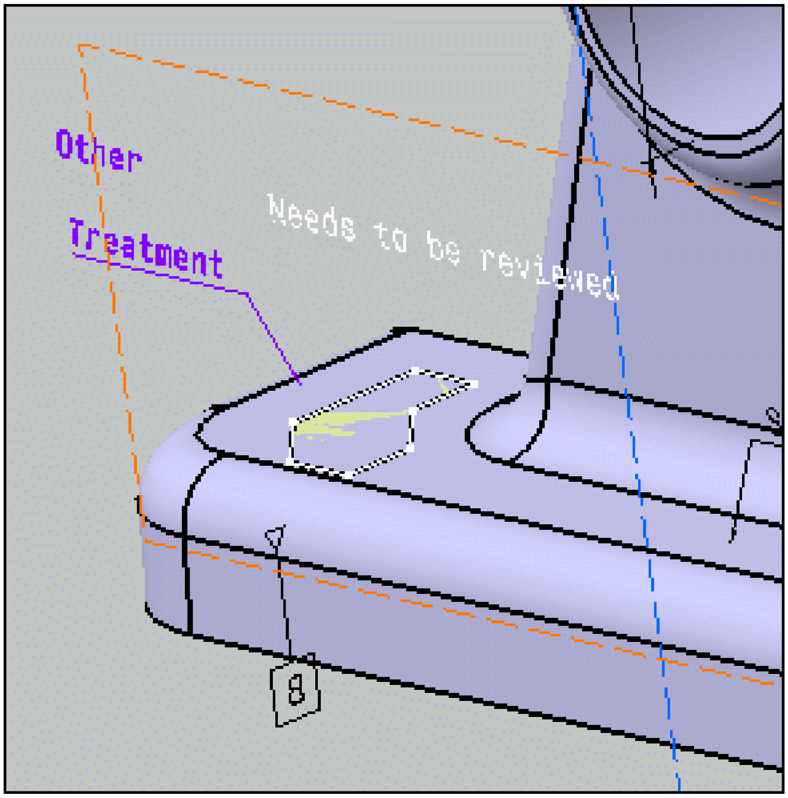


The **Text Editor** dialog box is displayed.

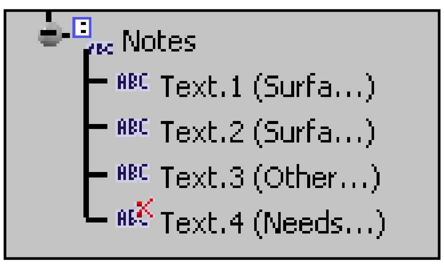
4. Type your text and then click **OK**.



The text is created in the specified annotation plane. You can now select it and drag it anywhere: it will remain in the annotation plane.



The note entity, Text.4, is added to the specification tree. A specific mask identifies this annotation as being isolated.



- Creating isolated dimensions is not possible.
- Isolated annotations are not semantic. You cannot convert them to semantic annotations.
- If you add a leader to an isolated annotation, or if you create an isolated annotation with leader (such as a text with leader or a flag note with leader), this leader will not be associative.

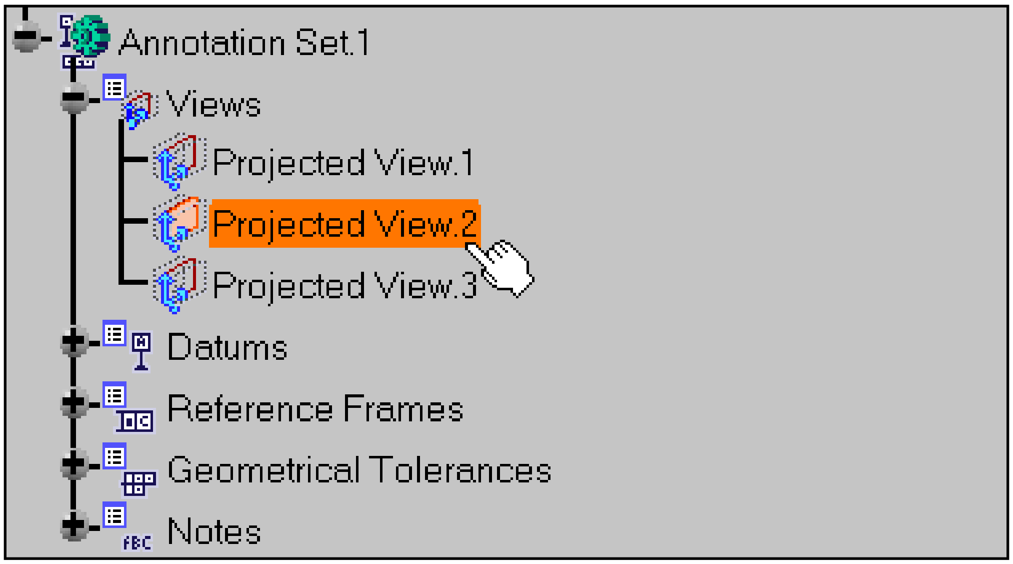
Creating Dimensions

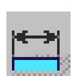
This task shows you how to create a dimension annotation. See Dimension Units reference for dimension's unit display.

Open the Annotations_Part_04.CATPart document:

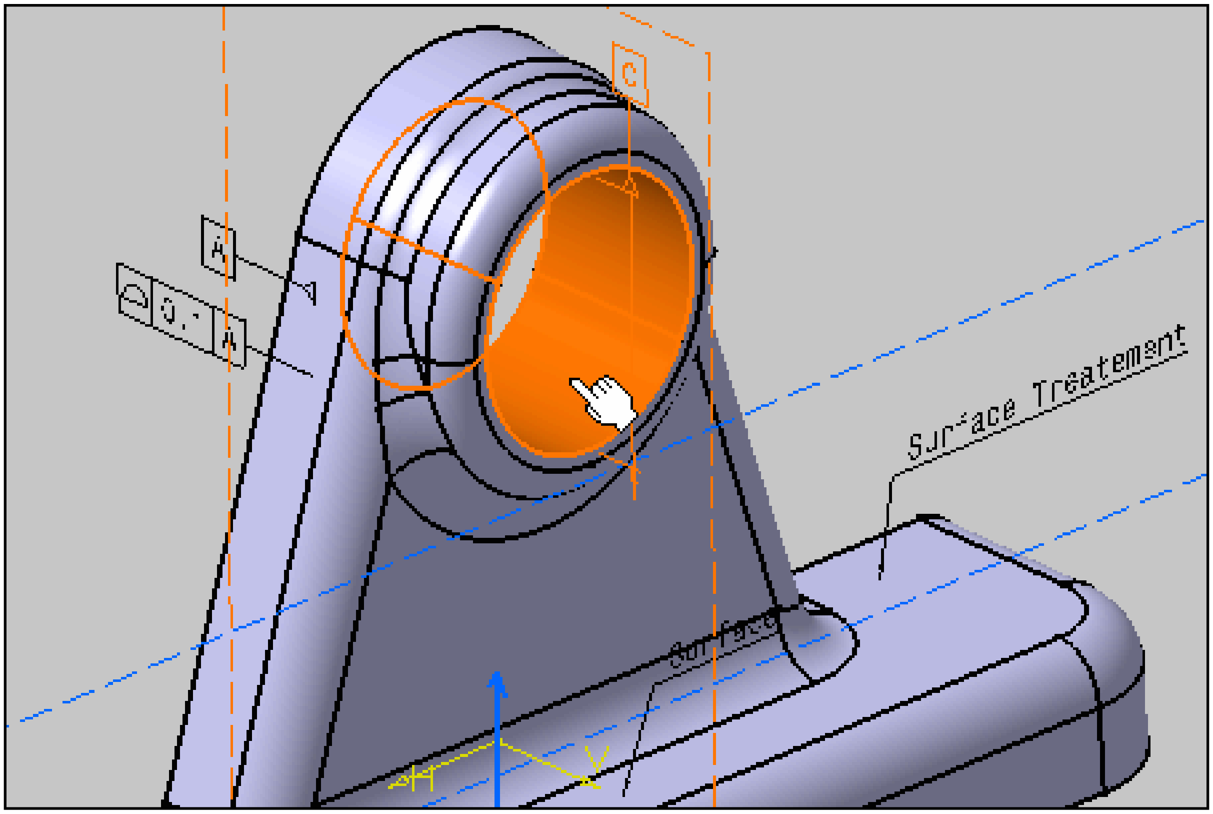
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Activate the **Projected View.2** annotation plane.



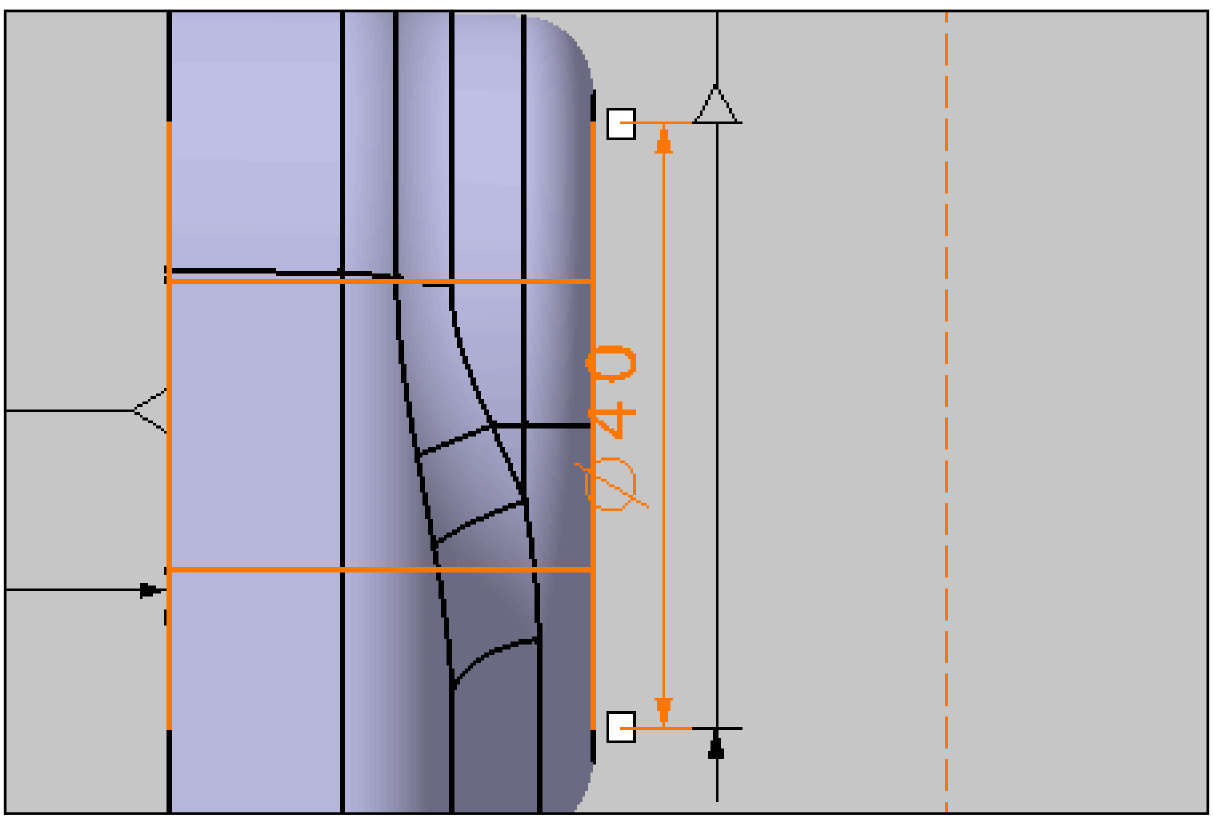
2. Click the **Dimensions** icon: 

3. Select the surface as shown on the part.

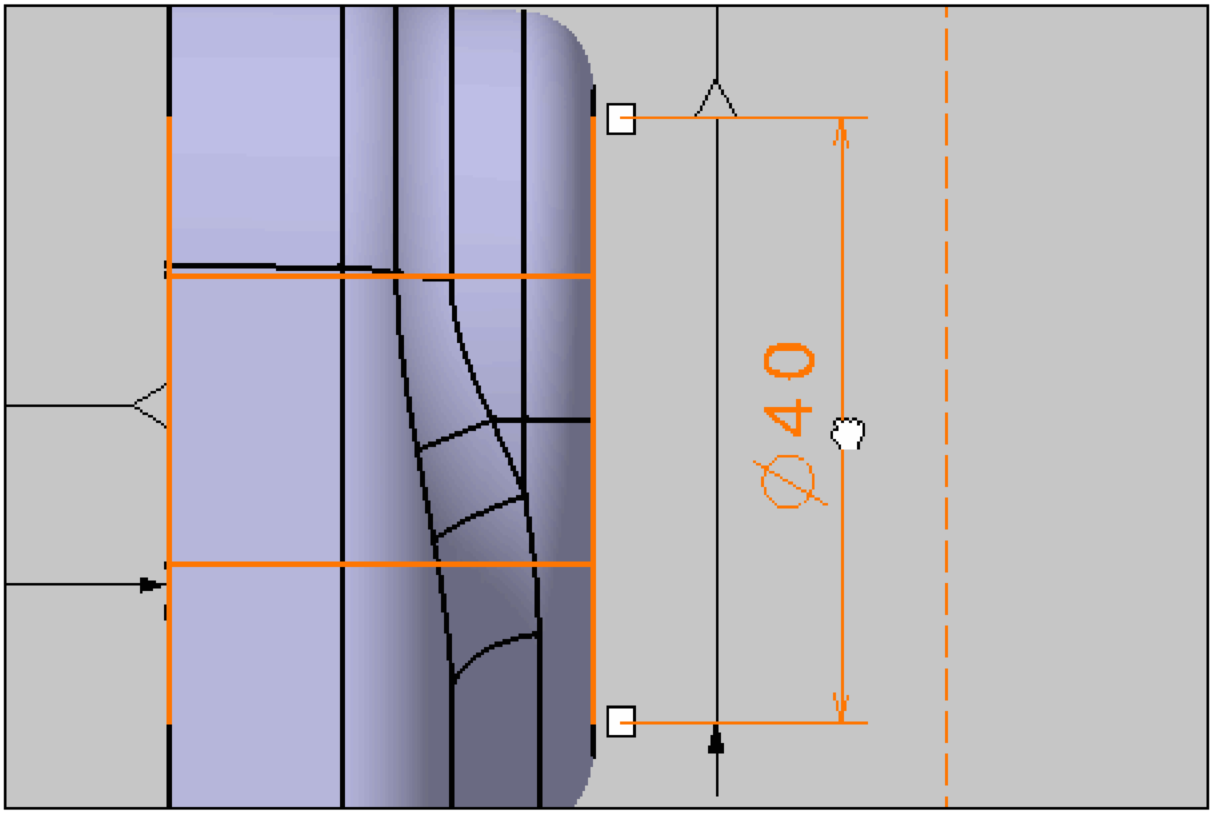


4. Click anywhere to create it.

The dimension is created.



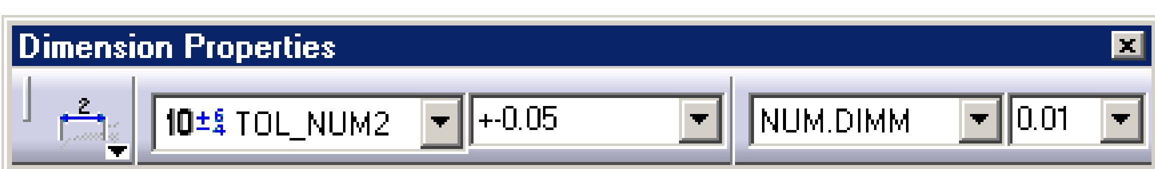
5. Drag the dimension.



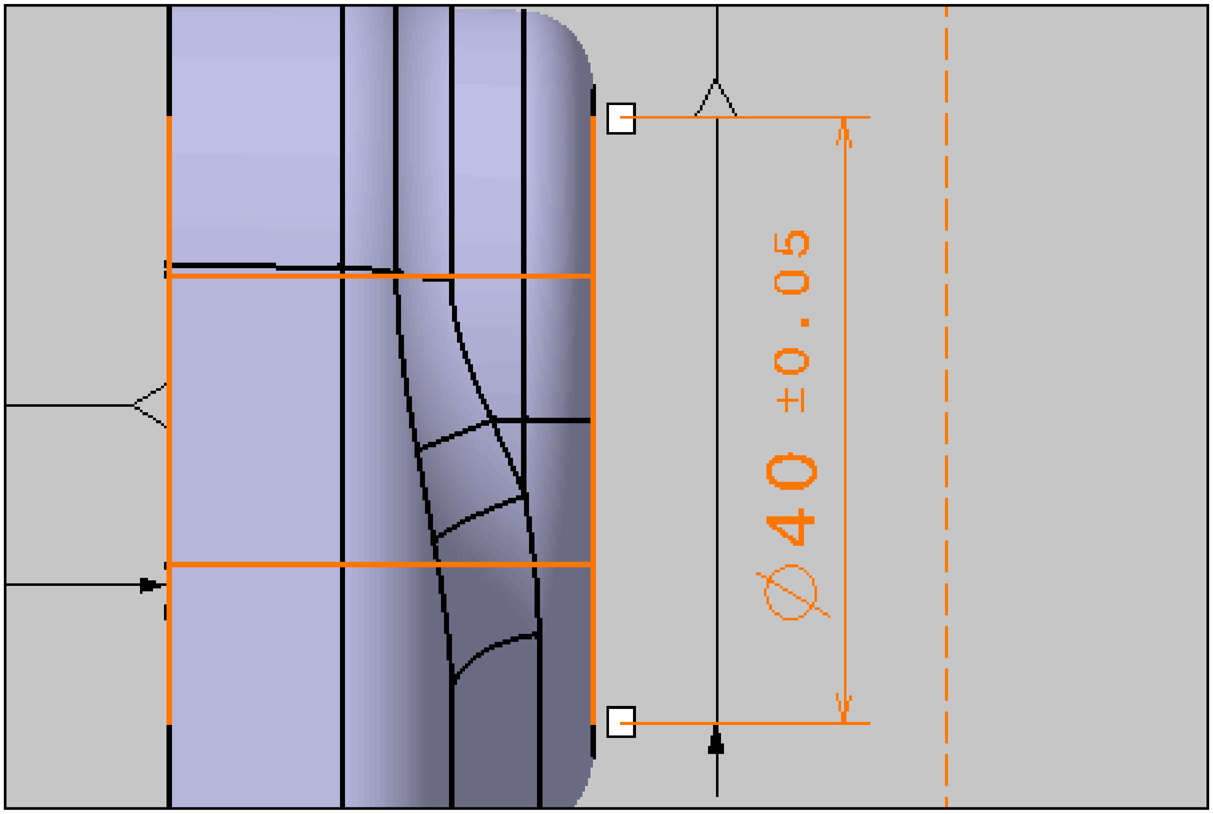
The **Dimension Properties** toolbar displays five combo boxes:

- **Line Type** to select the dimension line attachment.
- Tolerance Description to display the tolerance according to a standard.
- Tolerance to valuate the tolerance dimension.
- Numerical Display Description to set the numerical tolerance display.
- Precision to set the tolerance precision.

6. In the **Dimension Properties** toolbar, select **TOL_NUM2** in the **Tolerance Description** combo box, and **+-0.5** in the **Tolerance** combo box (the dimension is still selected).



Tolerances are displayed with the dimension.


∅40 ±0.05

Setting Dimension Representations

This task shows you how to set dimension annotation representations.

There are five representation mode:

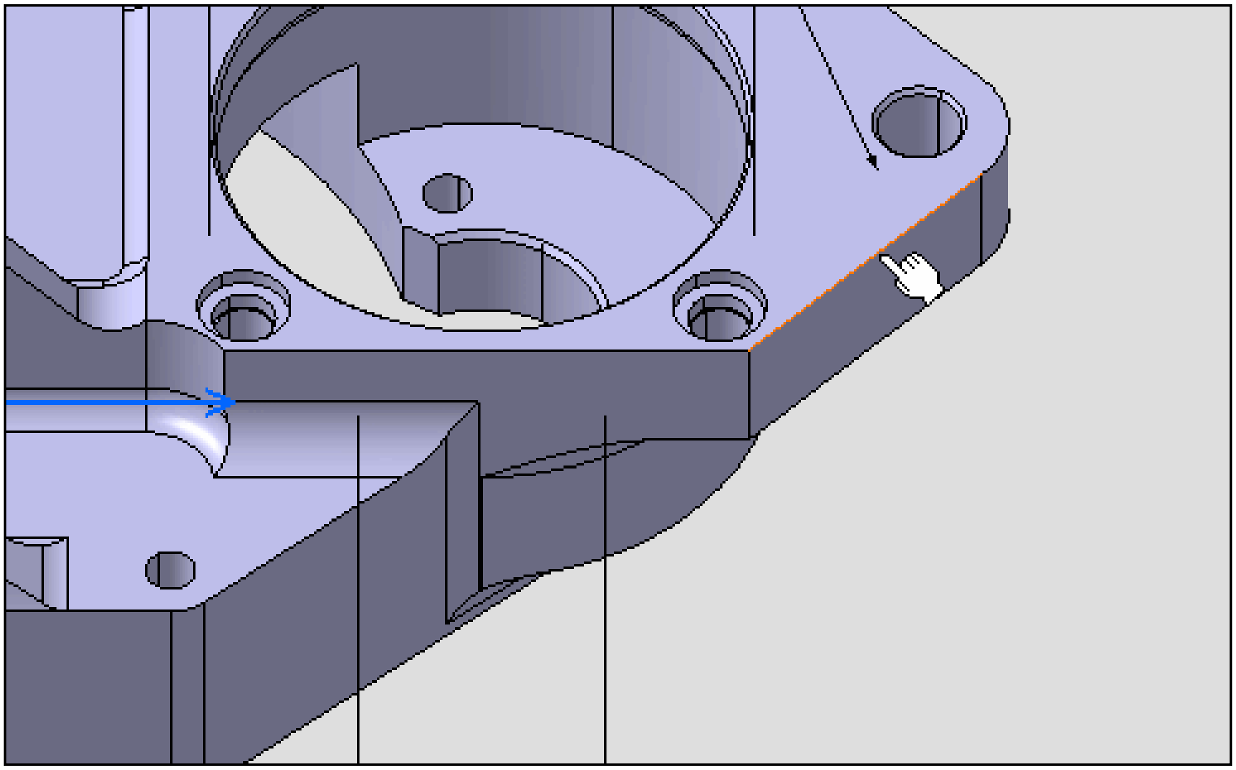
- **Projected Dimension**: project the dimension (on element, horizontal or vertical) according to the cursor position, see Dimension following the cursor (CTRL toggles).
- **Force Dimension on Element**: project the dimension on element.
- **Force Horizontal Dimension in view**: project the dimension horizontally, according to the annotation plane reference axes.
- **Force Vertical Dimension in view**: project the dimension vertically, according to the annotation plane reference axes.
- **True length dimension**: useless.

Open the Annotations_Part_02.CATPart document:

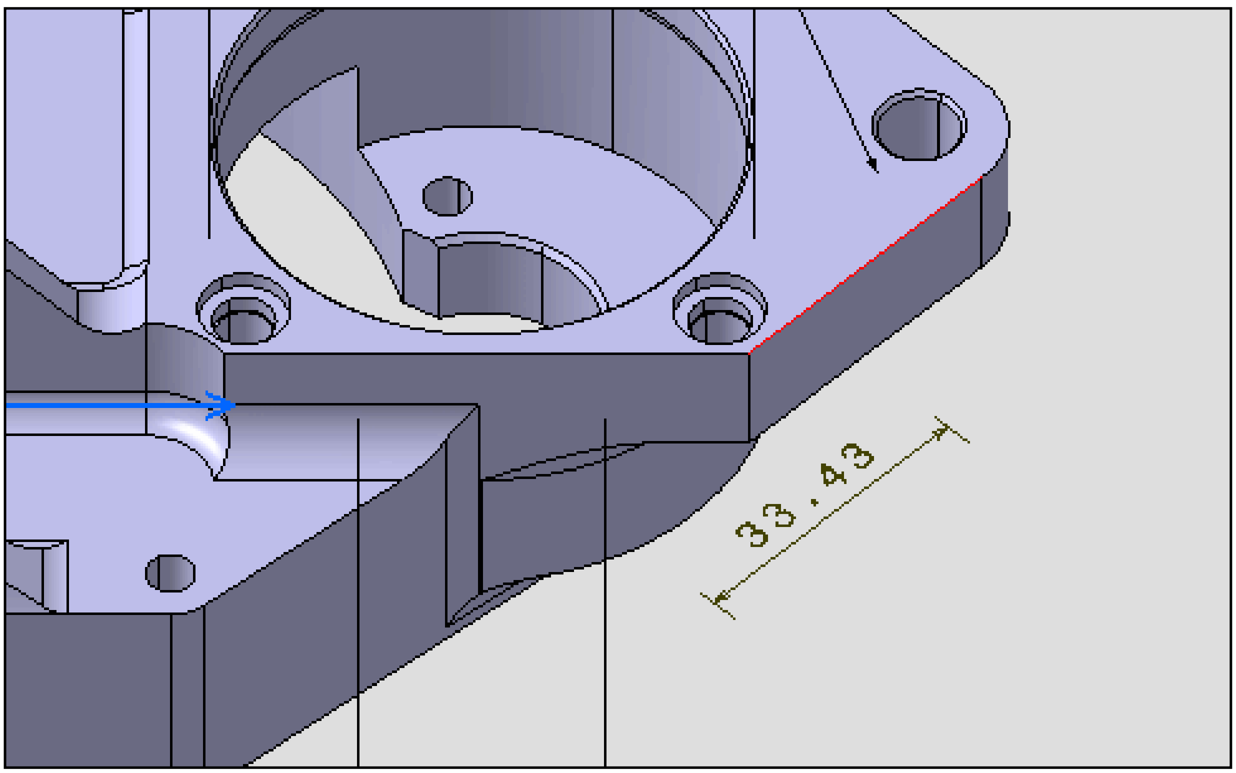
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Click the **Dimensions** icon:

2. Select the edge as shown on the part.

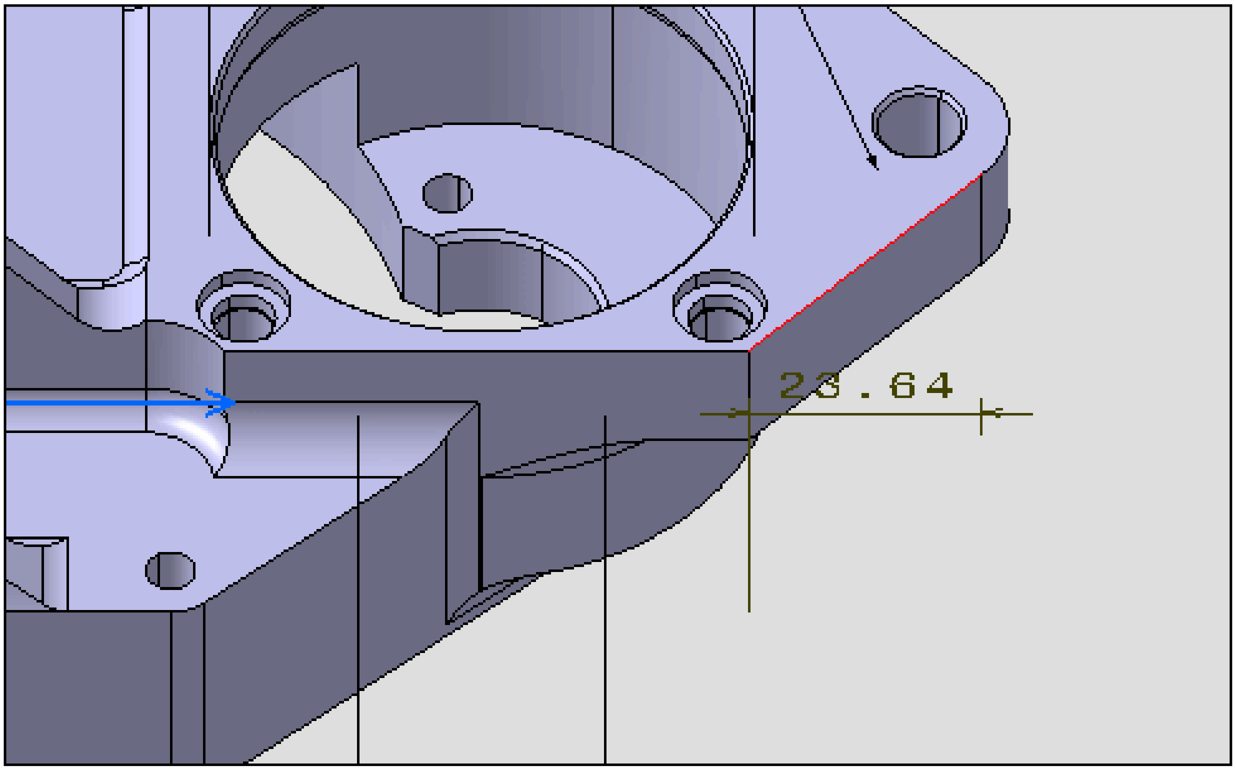


A linear dimension appears during the creation process.



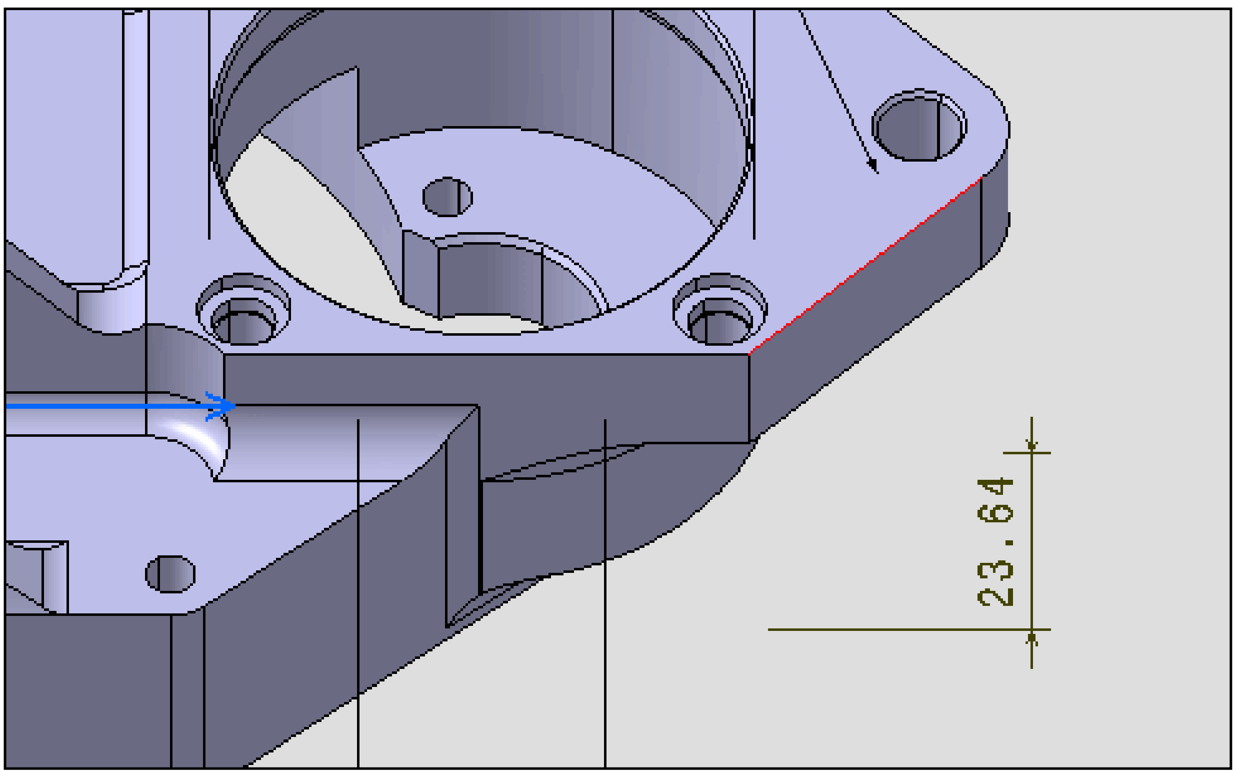
3. Right-click the dimension and select **Dimension Representation -> Force Horizontal Dimension in view** from the contextual menu.

The dimension representation is modified.



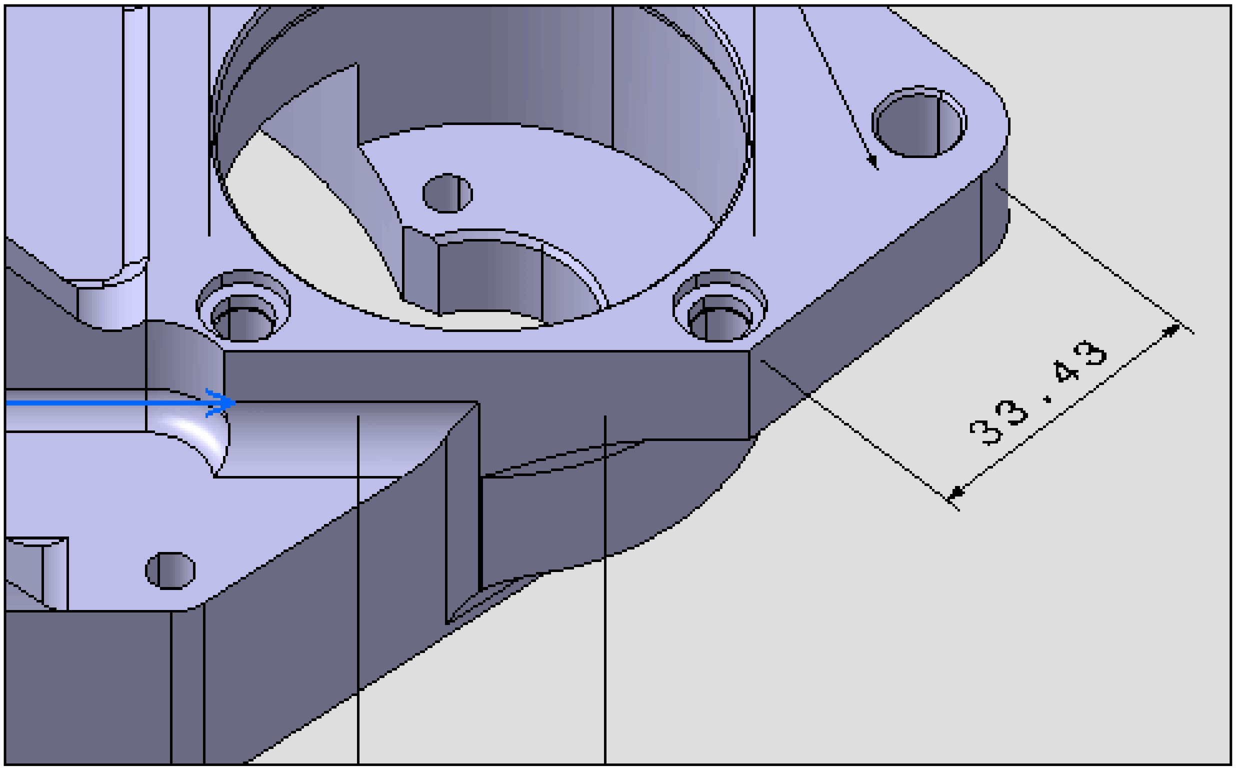
4. Right-click the dimension and select **Dimension Representation -> Force Vertical Dimension in view** from the contextual menu.

The dimension representation is modified.



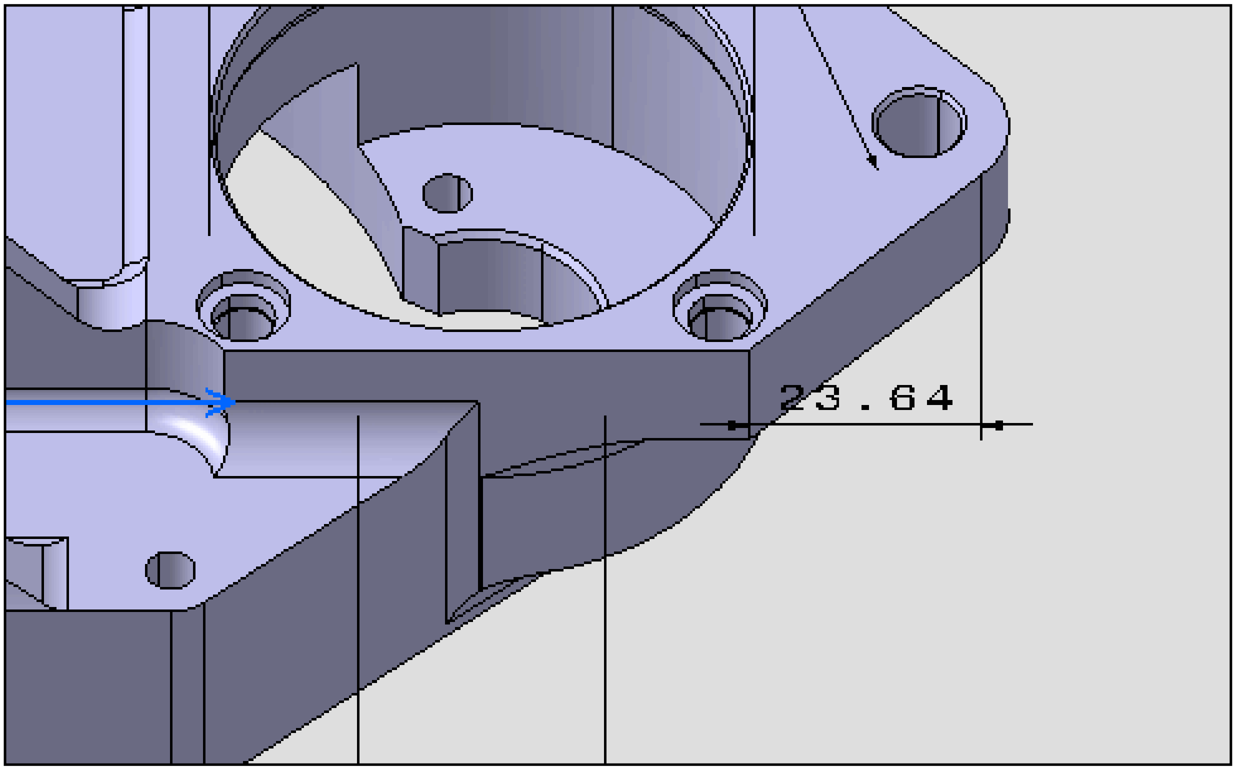
5. Right-click the dimension and select **Dimension Representation -> Force Dimension on Element** from the contextual menu.

6. Click outside any geometry to end the dimension creation process, move the dimension if needed.



7. Right-click the dimension and select **Dimension.1 object -> Dimension Representation -> Force Horizontal Dimension in view** from the contextual menu.

The dimension representation is modified.

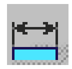

23.64

Creating Angular Dimensions

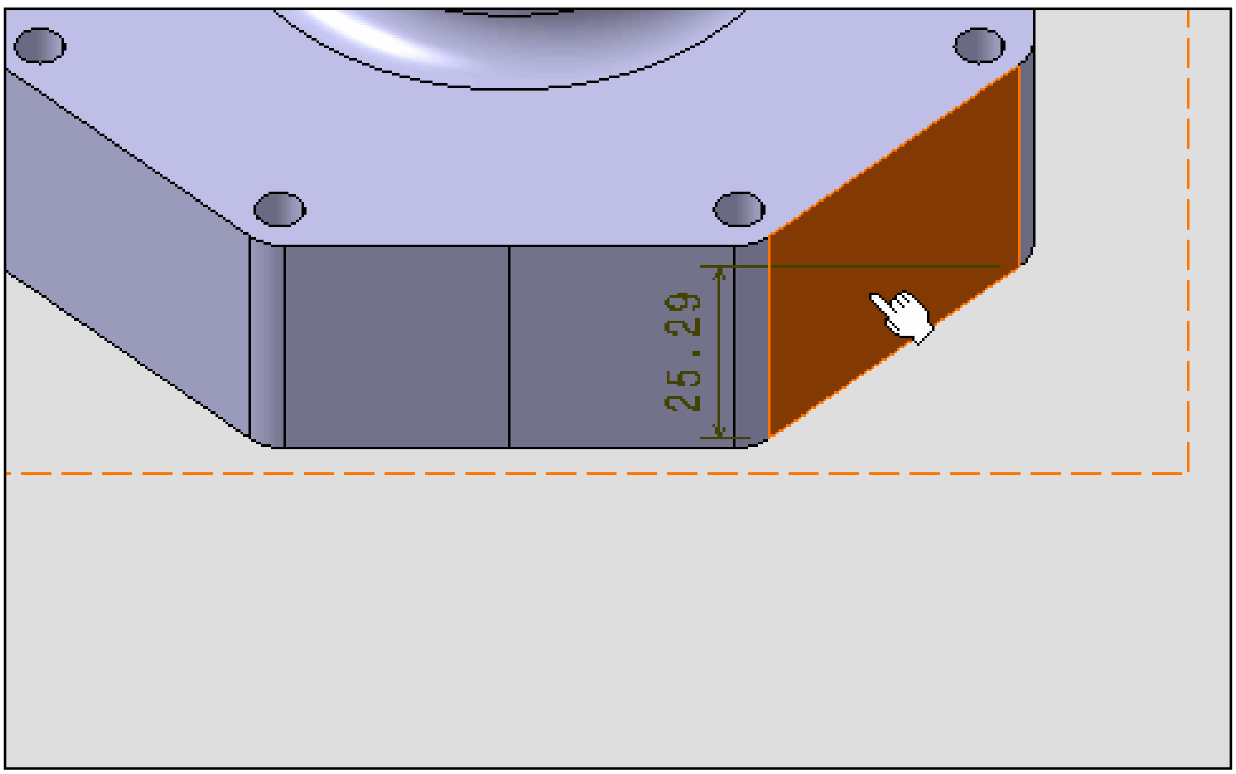
This task shows you how to create angular dimension annotations.

Open the Annotations_Part_02.CATPart document:

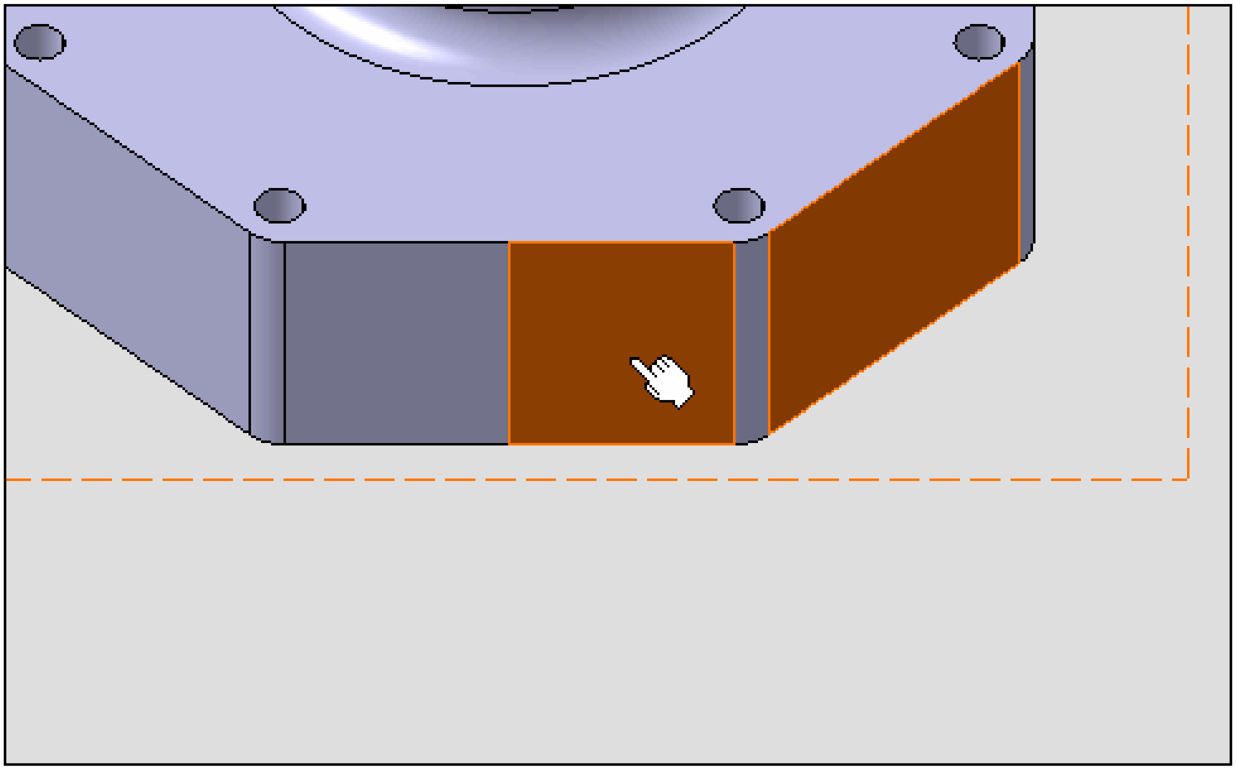
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Click the **Dimensions** icon: 

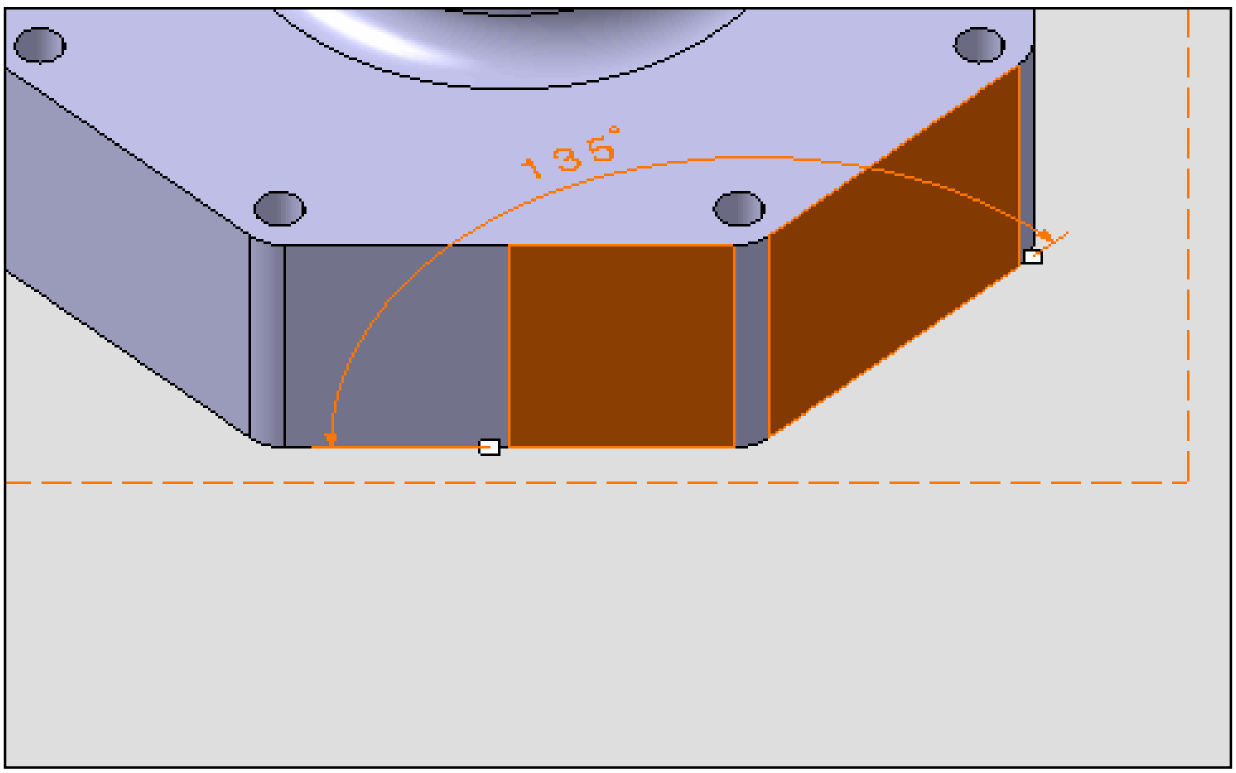
2. Select the surfaces as shown on the part.

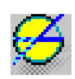


A linear dimension appears during the creation process before you select the second surface.

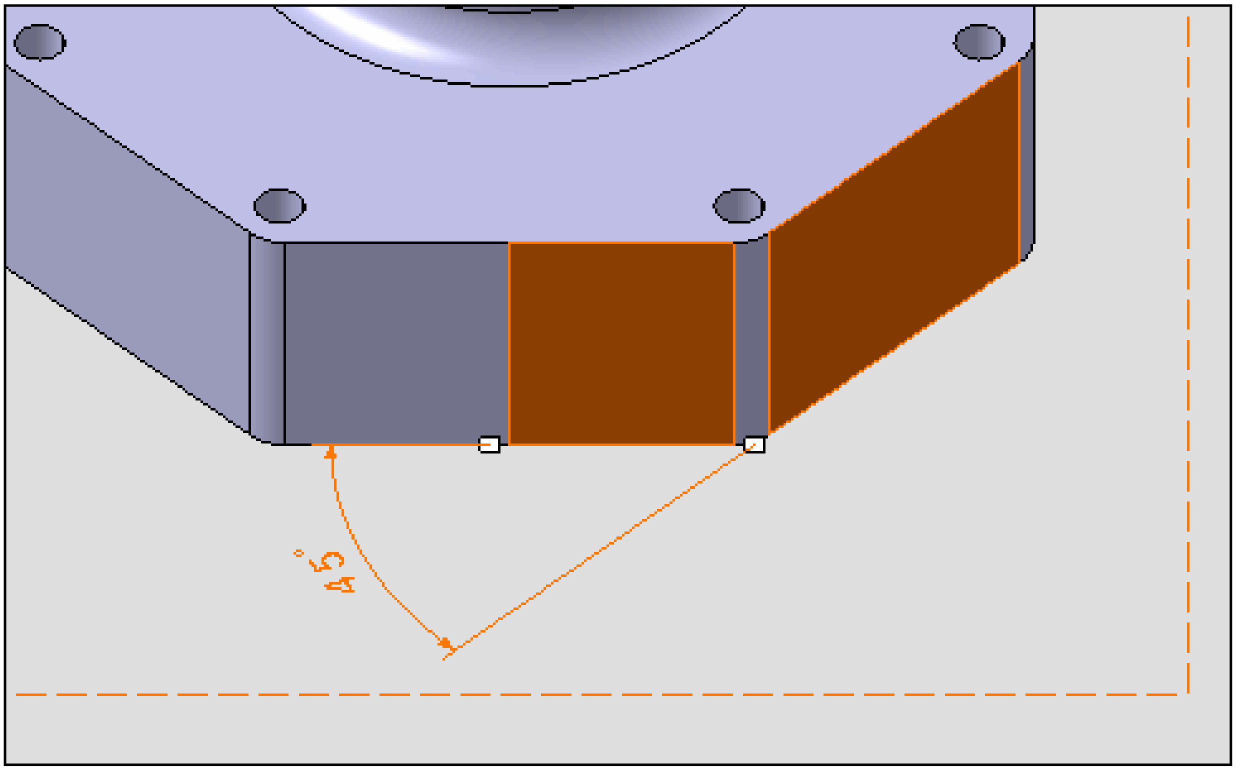


The angular dimension appears.



3. Right-click the angular dimension and select **Angle Sector -> Sector 4**  from the contextual menu.

The angular dimension is modified.

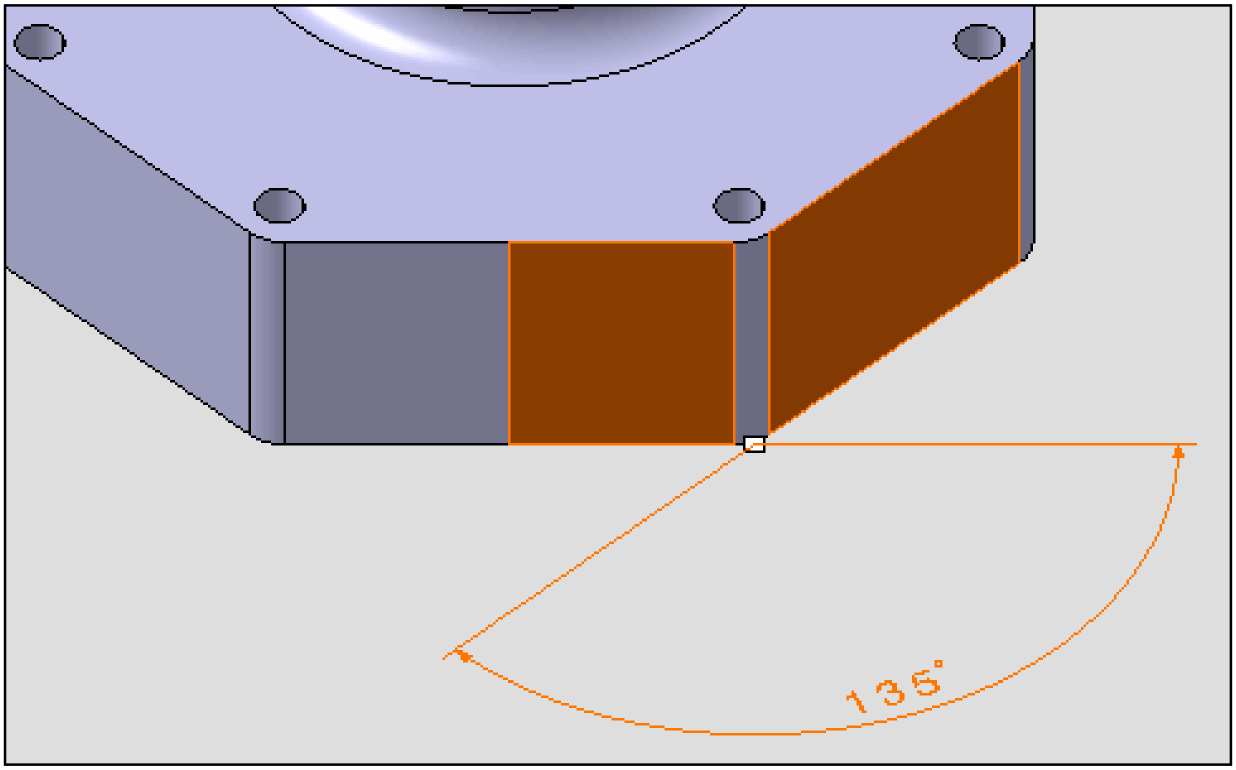


4. Click outside any geometry to end the dimension creation process.

Because the Always try to create semantic tolerances and dimensions option has been selected the **Angular Size** icon is displayed in the specification tree and the angular dimension named **Angular Size.1** during the dimension creation process.
But at the end of the dimension creation process, this dimension has not been set as semantic.
This is why the **Linear Dimension** icon appears in the specification tree and the angular dimension renamed as **Dimension.1**.

5. Right-click the angular dimension and select **Dimension.1 object -> Angle Sector - > Sector 1**  from the contextual menu.

The angular dimension sector is modified.


135°

Creating Basic Dimensions

This task shows you how to create basic dimensions. These reference dimensions are used to define the location or the size of a geometrical element, from existing or new dimensions, and related to a specific context.

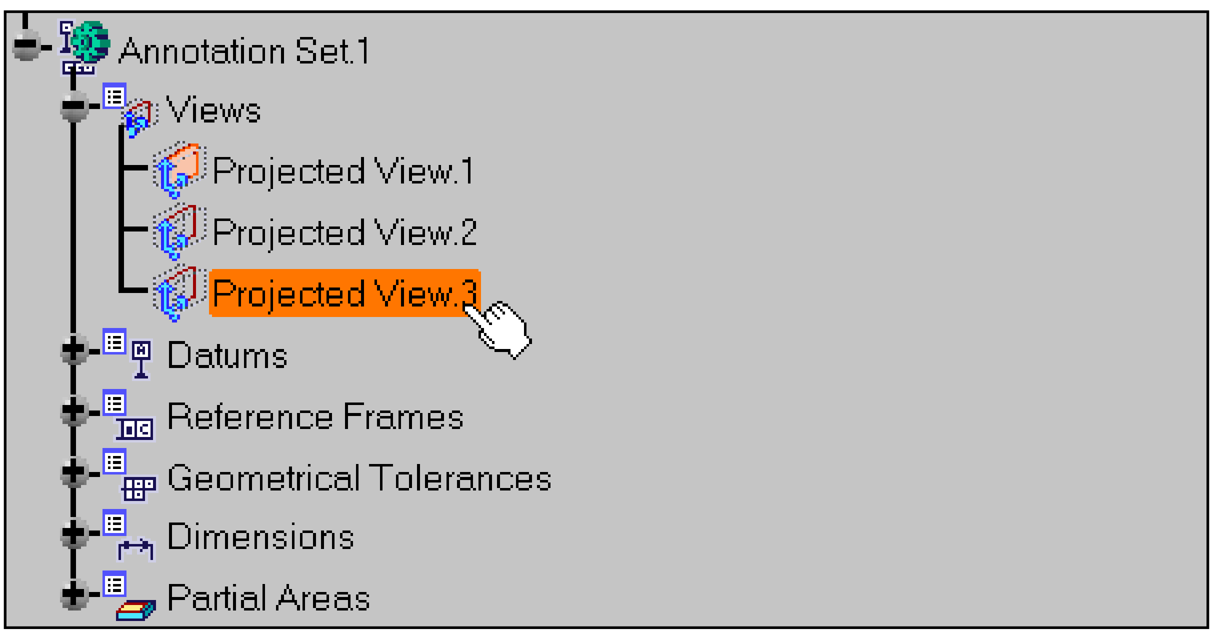
This command allows you to create basic dimensions from the four following contexts:

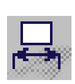
- Restricted areas, where basic dimensions define the restricted area dimensions and location.
- Datum targets, where basic dimensions define the datum target location from another datum target or geometrical elements.
- Datum reference frames, where basic dimensions define the datum reference frame location from reference elements or representing constructed geometry.
- Geometrical tolerances, where basic dimensions define the toleranced element location from references in the reference frame.

Open the Tolerancing_Annotations_03 CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

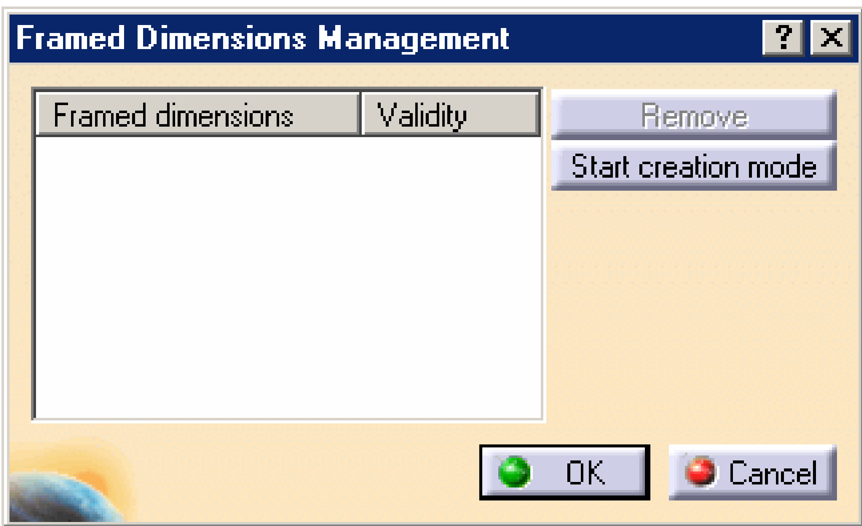
1. Double-click the **Projected View.3** to activate it.



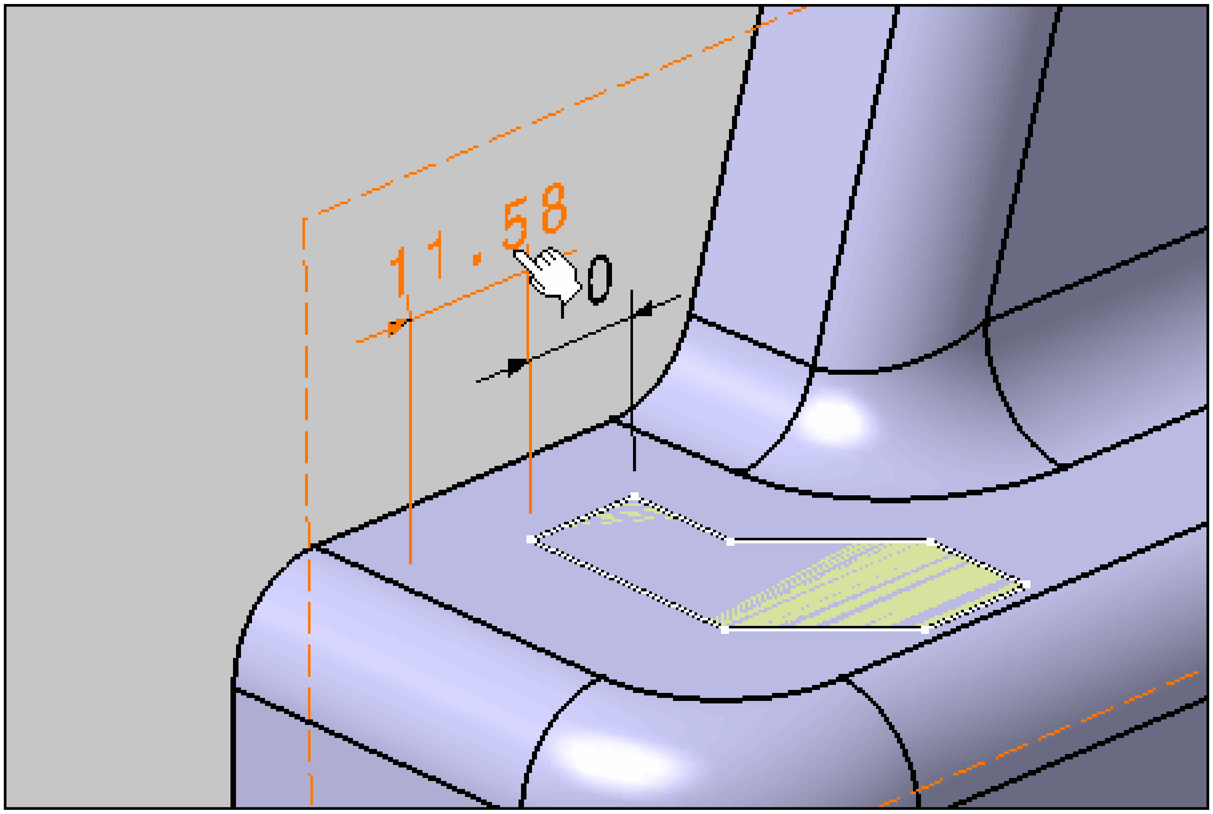
2. Click the **Basic Dimension** icon: 

3. Select the **Restricted Area.1** partial area.

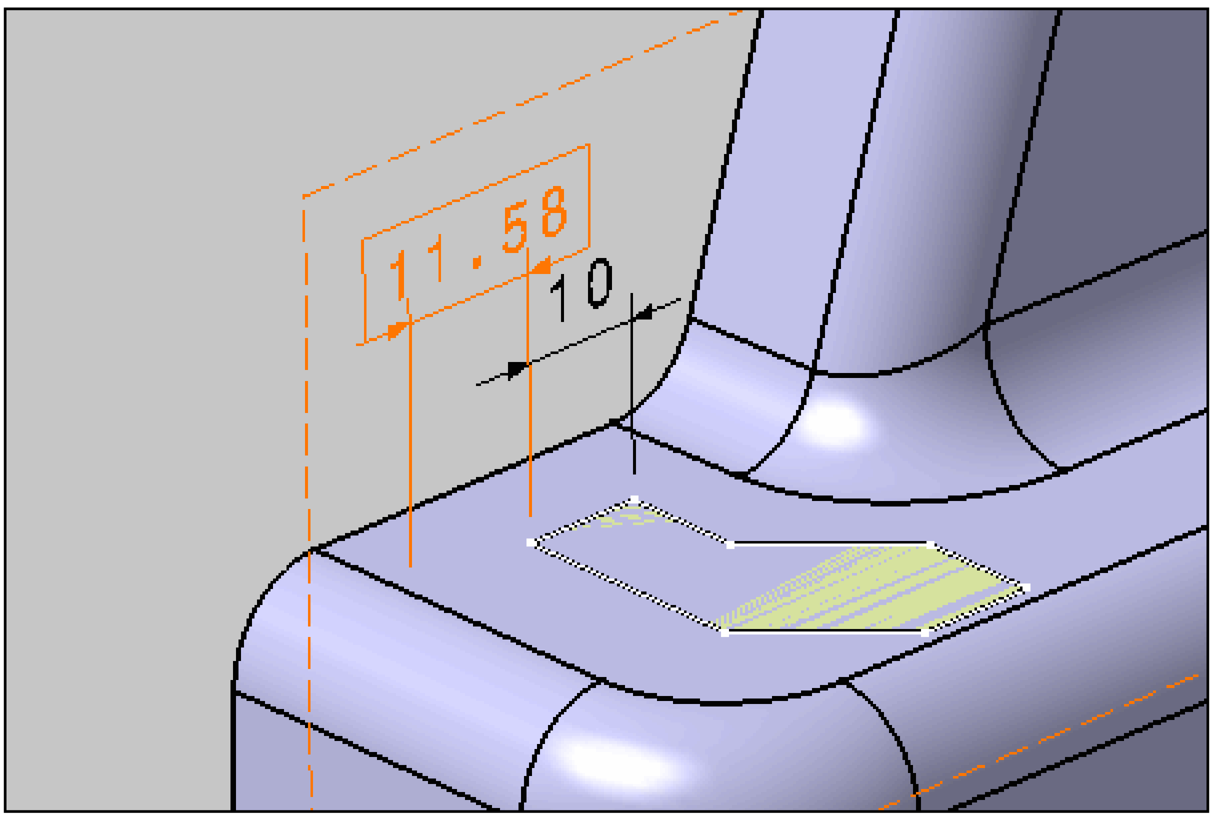
The **Framed Dimensions Management** dialog box appears.

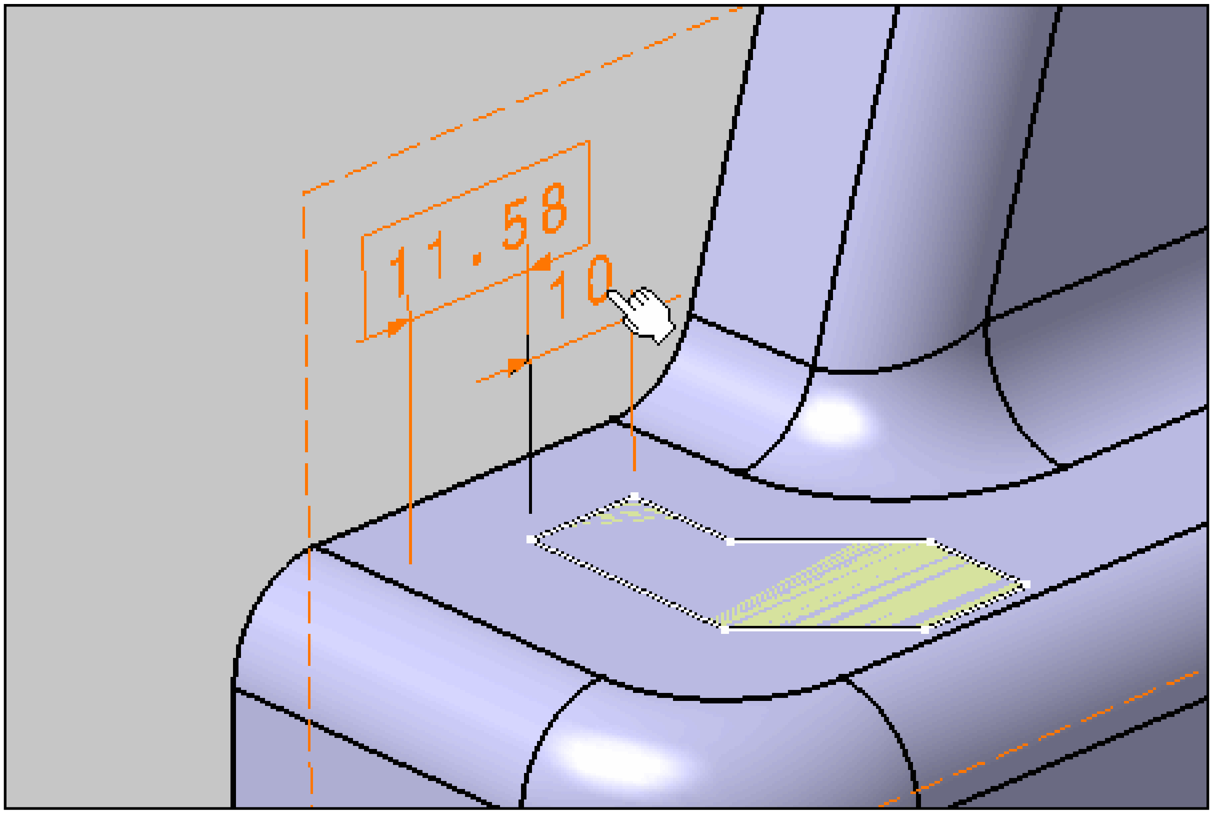


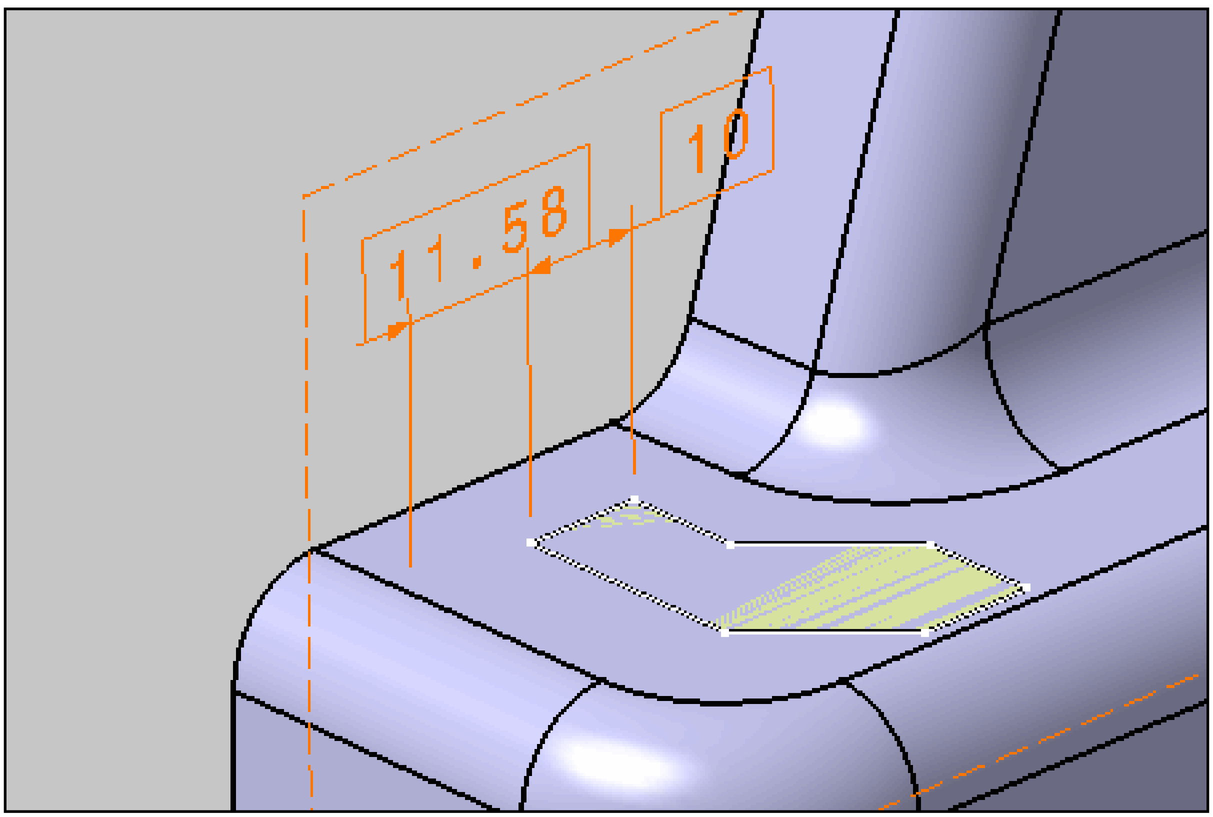
4. Select dimensions as shown on the part.



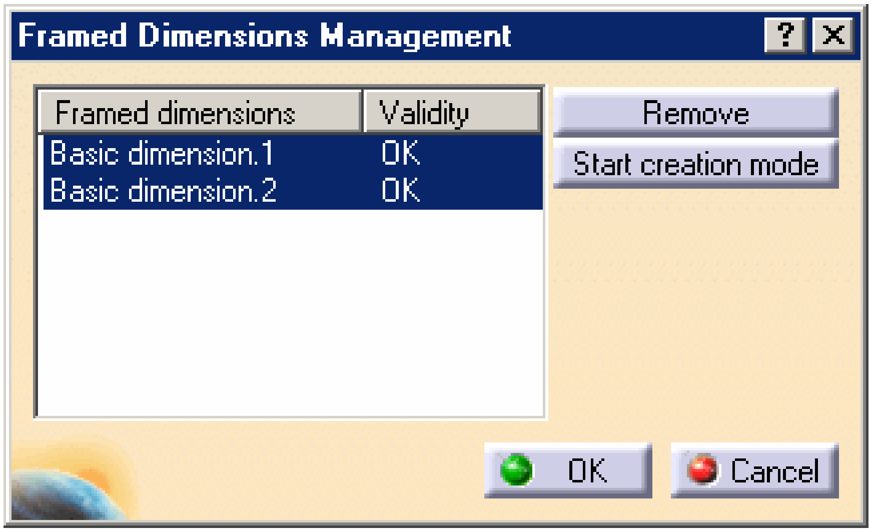
The dimension is converted into a basic dimension when clicking.



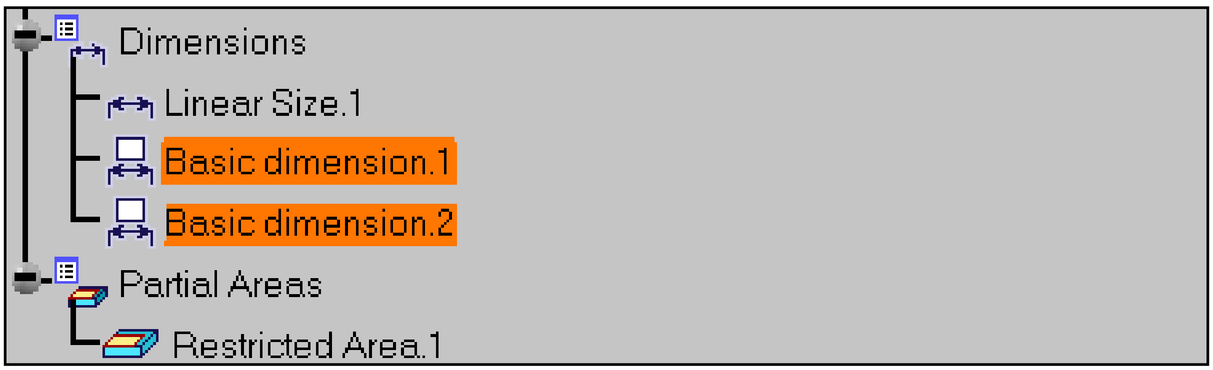




The **Framed Dimensions Management** dialog box is updated. The validity of each converted annotation is checked.

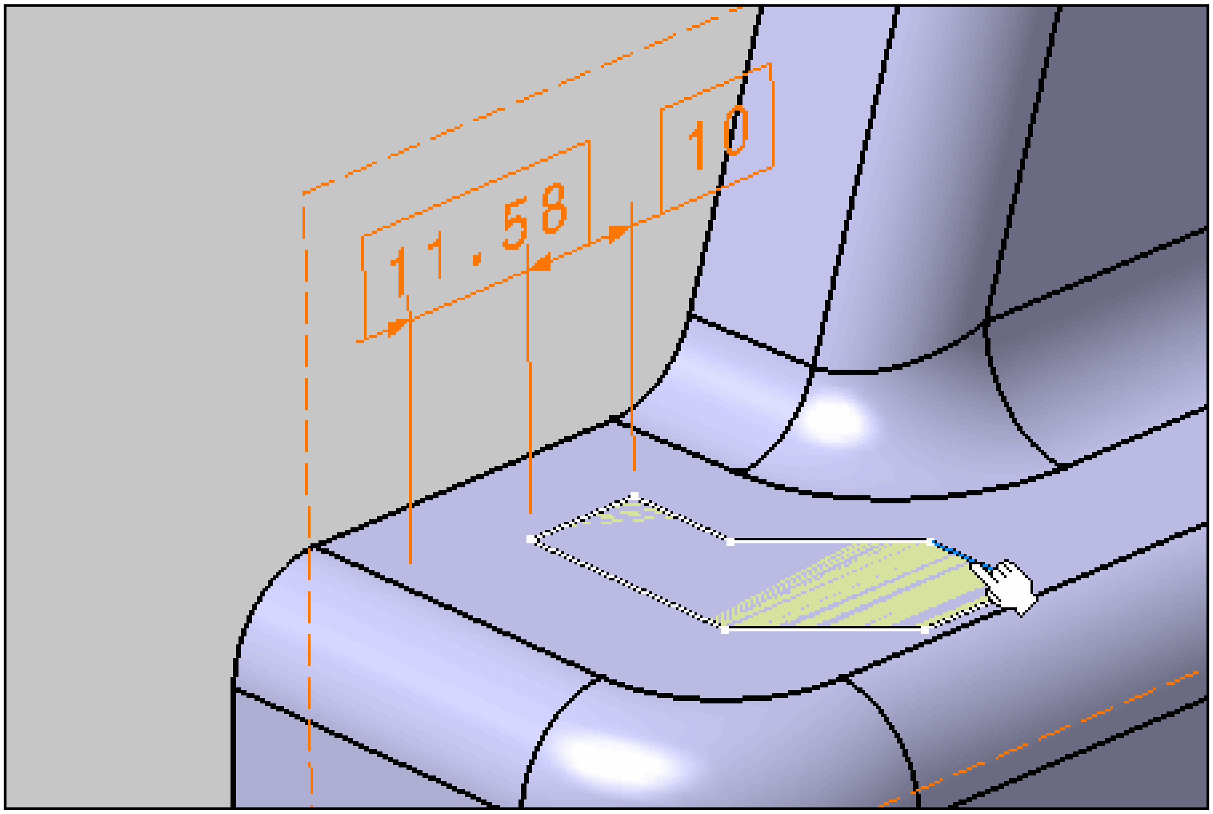


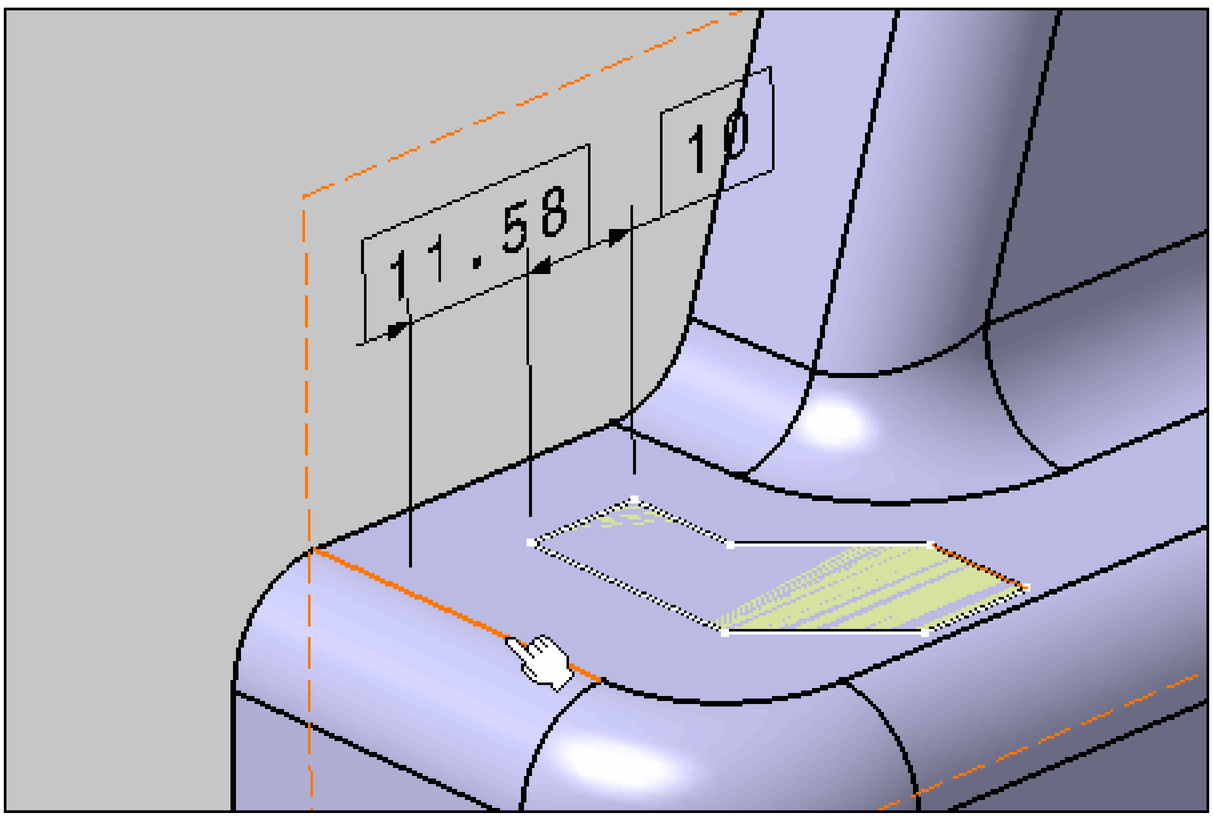
During the conversion, converted annotations remain selected in the specification tree, the geometry area and the dialog box.



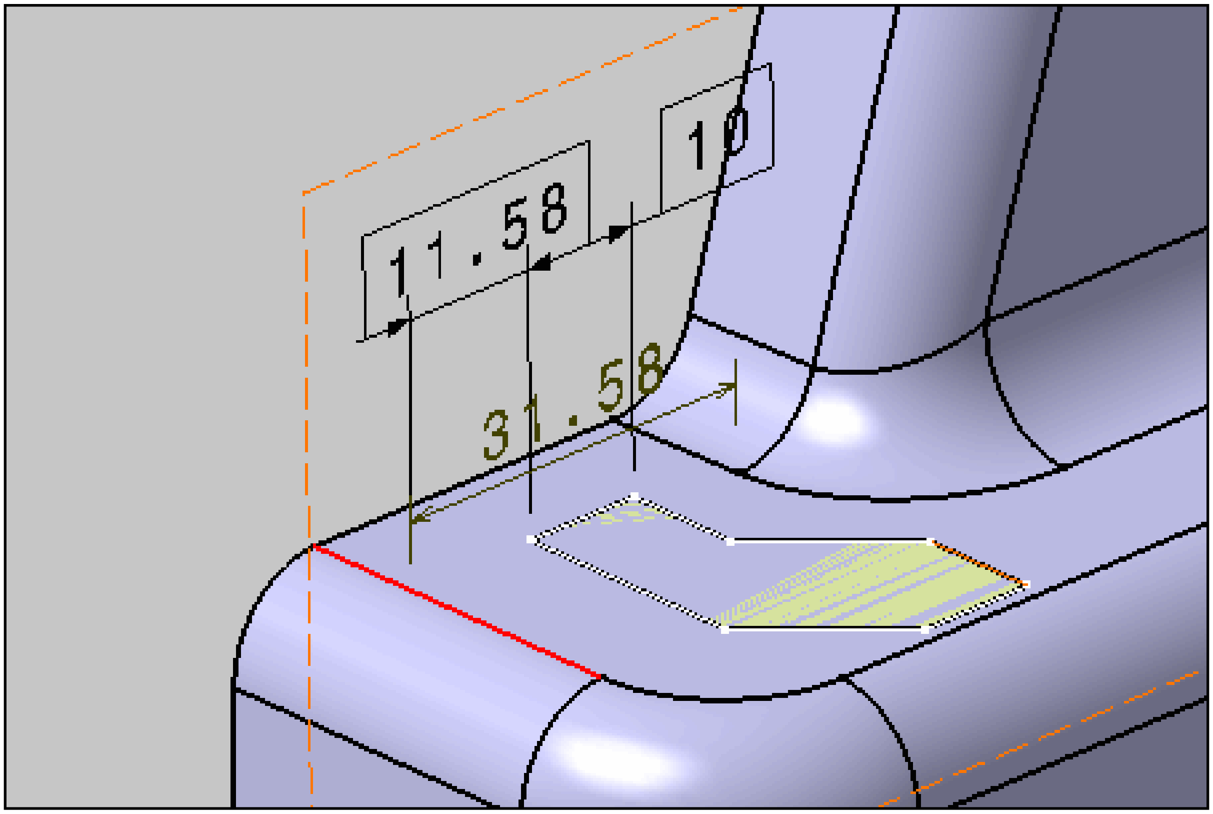
5. Click **Start creation mode** in the **Framed Dimensions Management** dialog box.

6. Select edges as shown on the part.

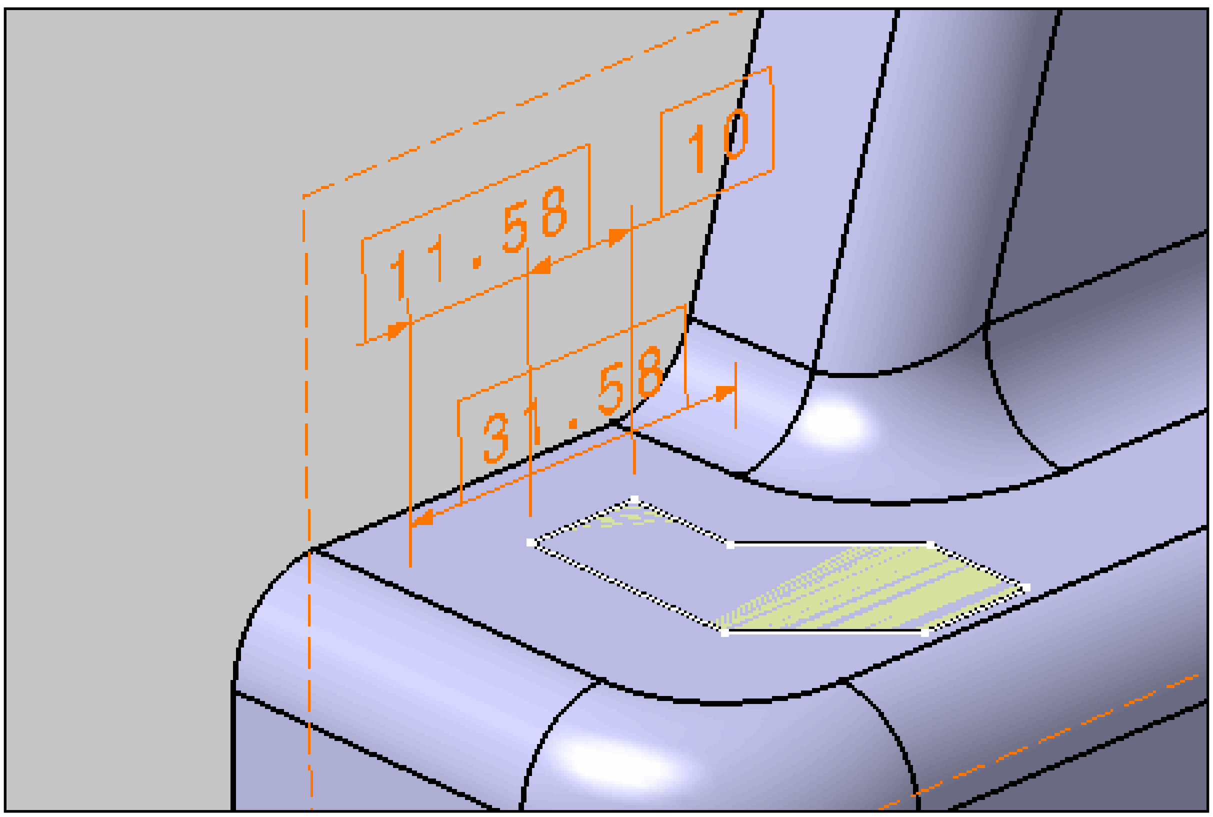




The basic dimension is pre-created.

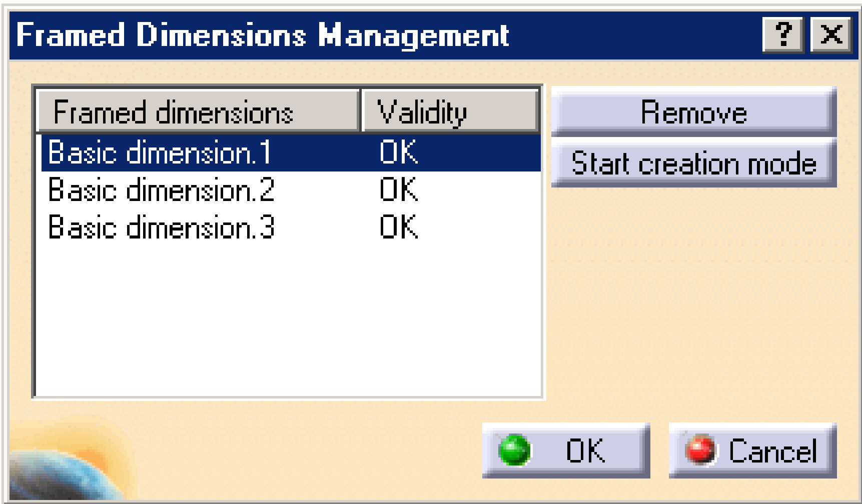


7. Click in the free space to create the basic dimension.

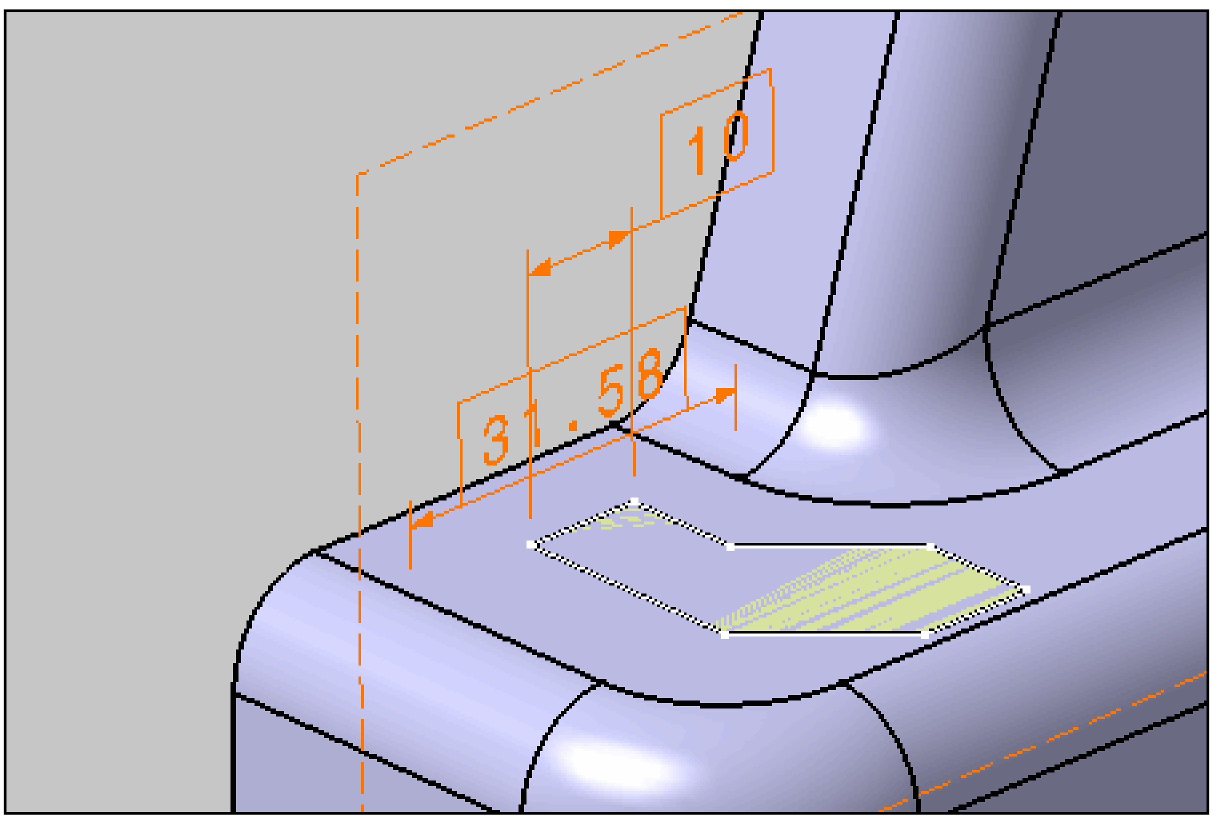


8. Click **End creation mode** in the **Framed Dimensions Management** dialog box.

9. Select **Basic Dimension.1** in the **Framed Dimensions Management** dialog box, and click **Remove**.



The basic dimension has been removed.



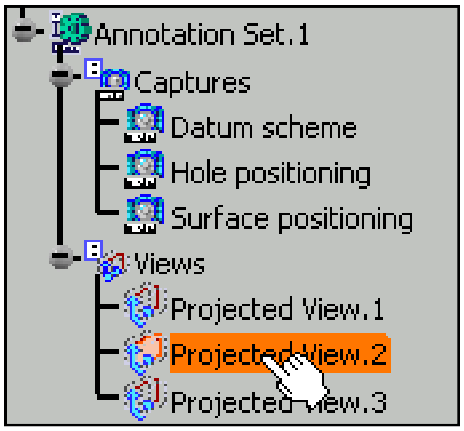
10. Click **OK** in the **Framed Dimensions Management** dialog box.

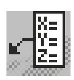
Creating Coordinate Dimensions

This task shows how to create a coordinate dimension based on the coordinates of a selected point. See Dimension Units for more information on dimension unit display.

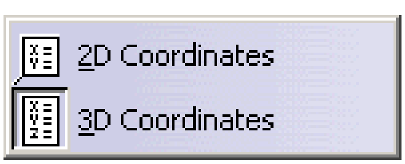
Open the Tolerancing_Annotations_06 CATPart document.

1. Activate the **Projected View.2** annotation plane.

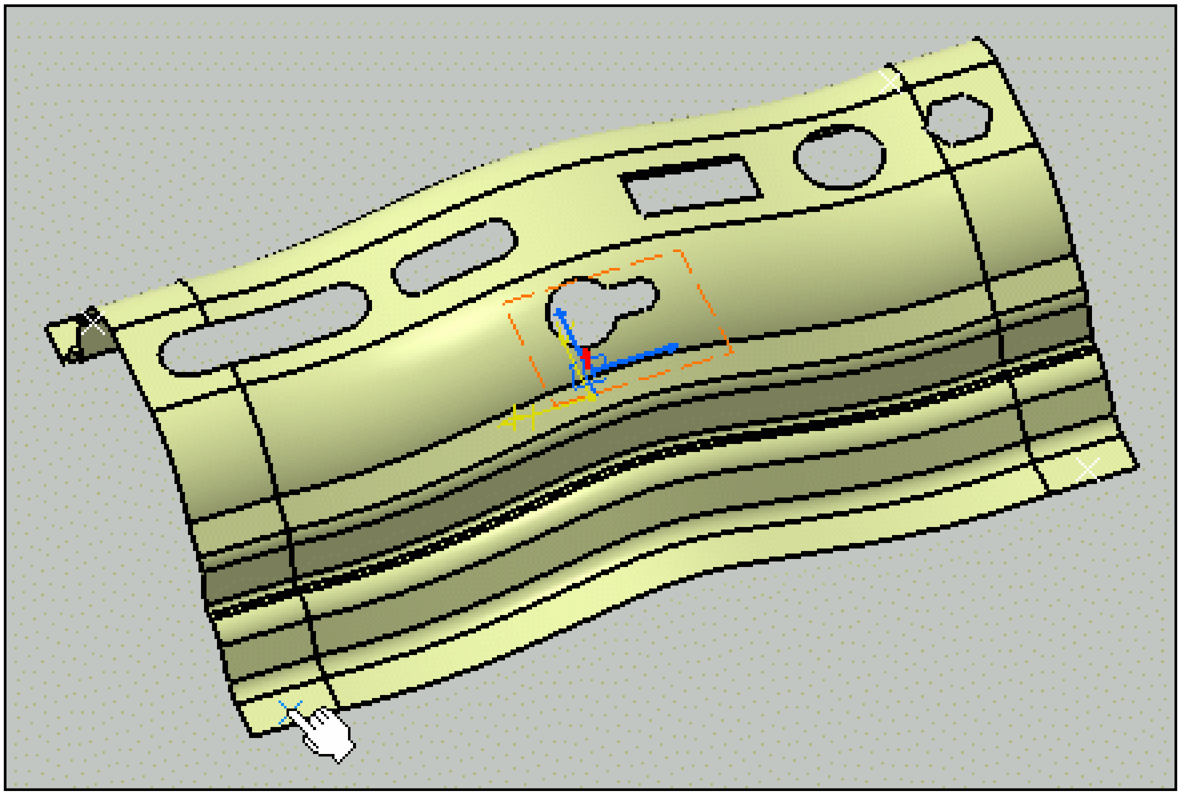


2. Click the **Coordinate Dimensions** icon. 
3. Right-click anywhere to display the contextual menu.
   - **2D Coordinates** lets you create 2D (x, y) coordinate dimensions in the active view axis system.
   - **3D Coordinates** lets you create 3D (x, y, z) coordinate dimensions in the part/product axis system.

For the purpose of this scenario, make sure **3D Coordinates** is selected.



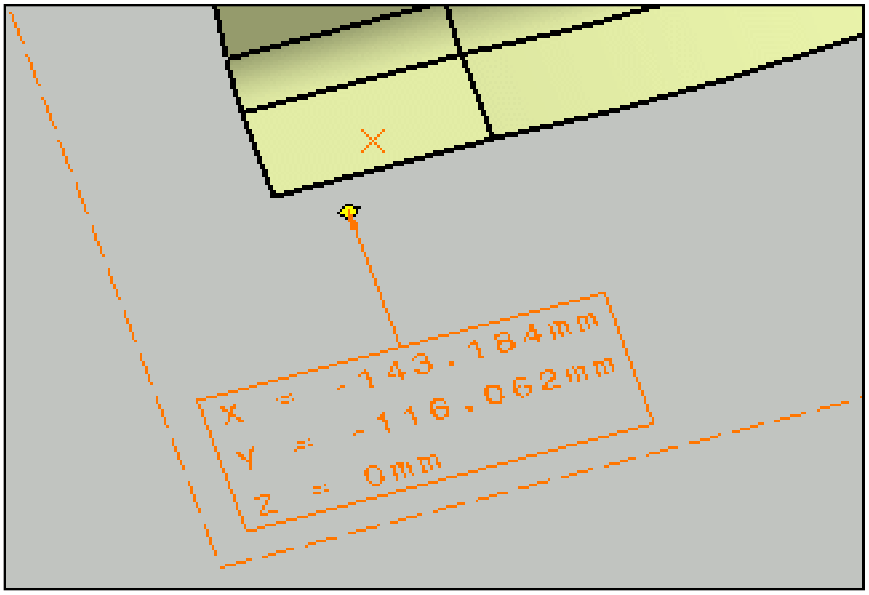
4. Select a point as shown below.



You can select the following elements: a vertex, a point (on a curve, on a plane, a coordinate), a line center or a point on a curve.

The dimension is created.

5. Drag the dimension to position it as wanted.



Once a coordinate dimension has been created, you cannot change its type (i.e. you cannot turn a 2D coordinate dimension into a 3D coordinate dimension, and vice-versa).

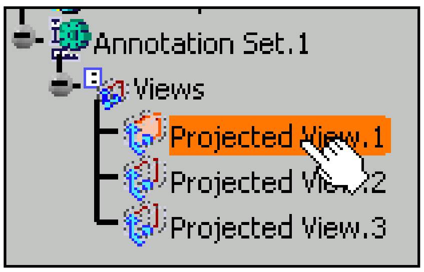
Creating Stacked Dimensions

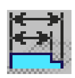
This task shows how to create stacked dimensions. See Dimension Units for more information on dimension unit display.

Open the Tolerancing_Annotations_10 CATPart document.

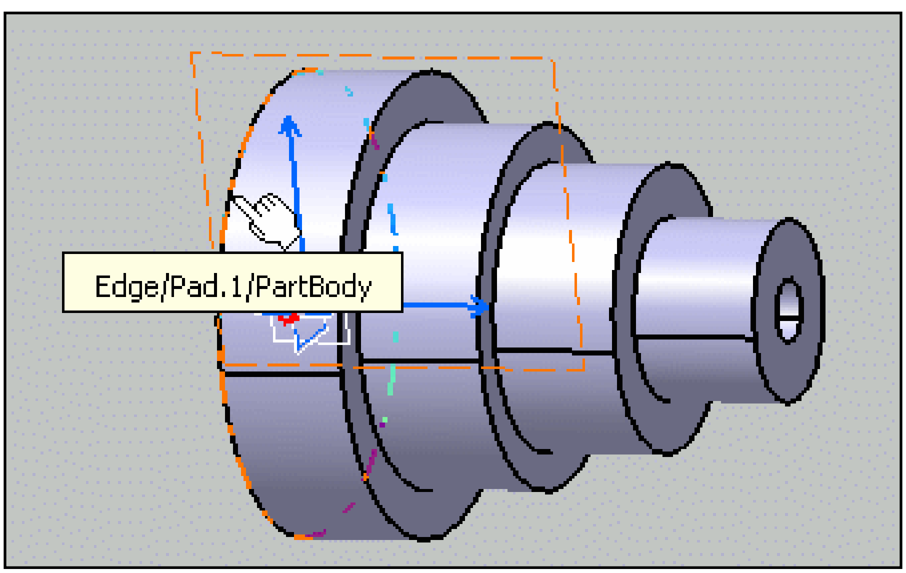
Select **Tools -> Options**. In the **Mechanical Design** category, select the **Functional Tolerancing & Annotation** sub-category, then the **Dimension** tab and check **Align stacked dimension values**.

1. Activate the **Projected View.1** annotation plane.

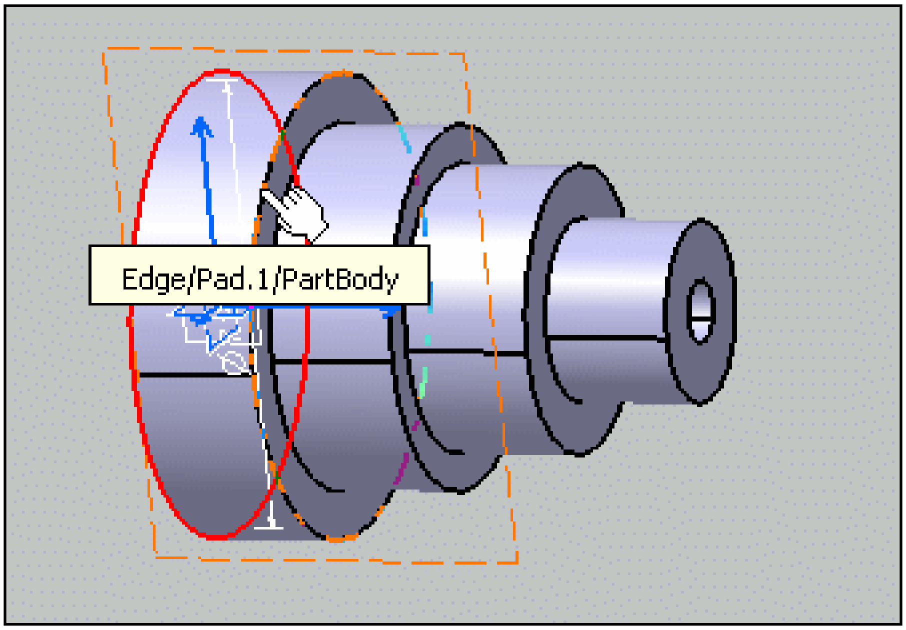


2. Click the **Stacked Dimensions** icon. 

3. Select the first element.

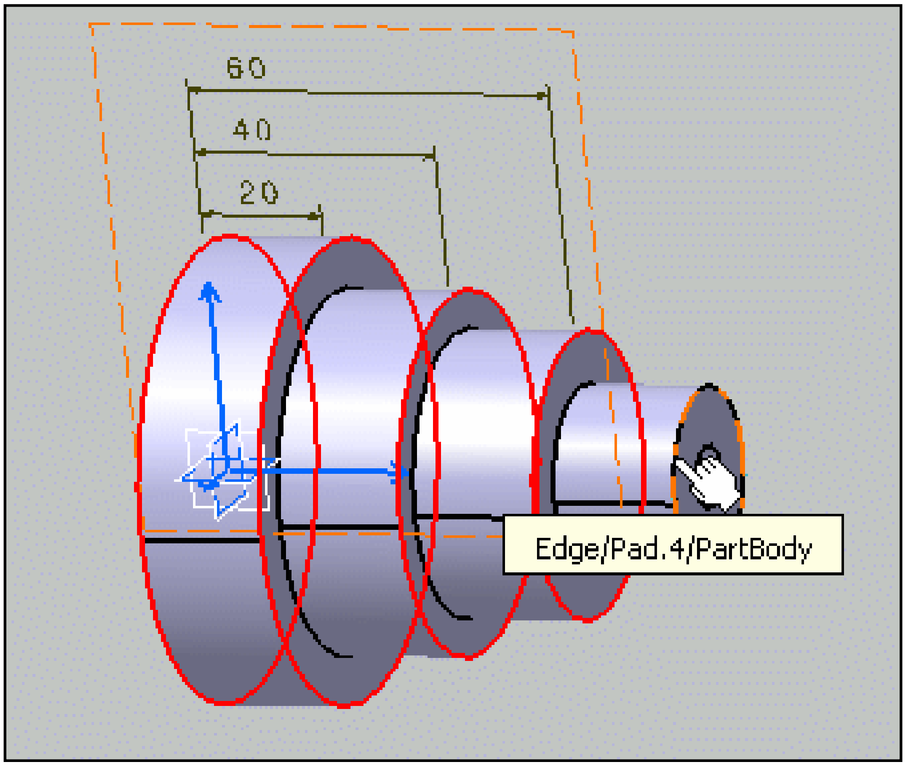


4. Select the second element.

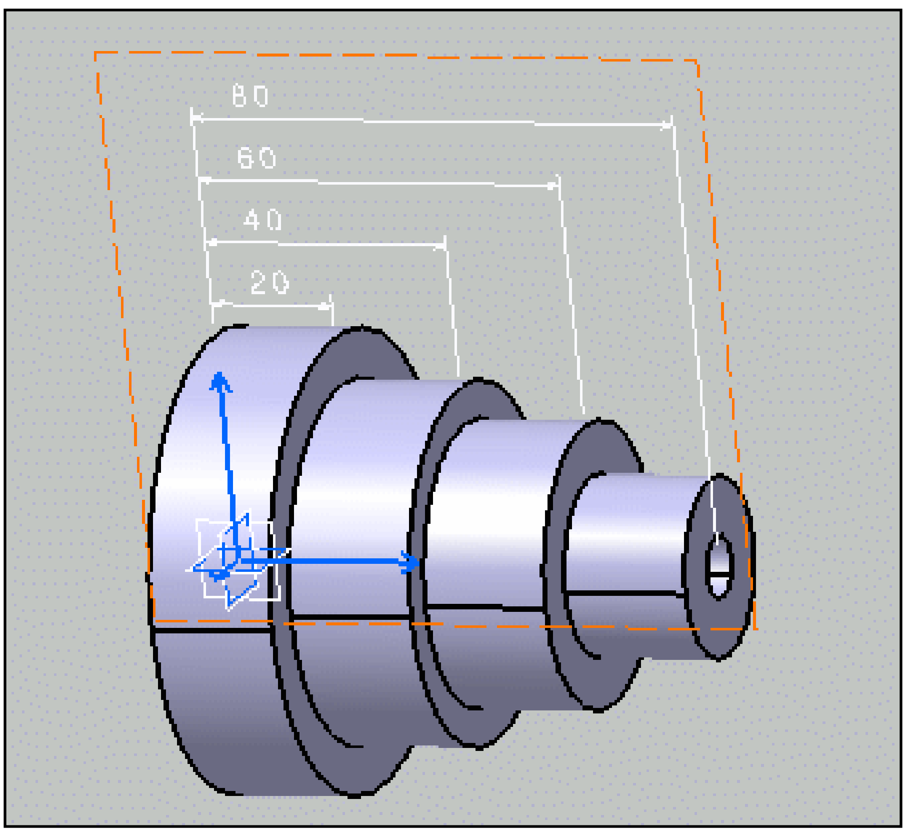


You just created a first dimension within the stacked dimensions system. You can then proceed to create as many dimensions as necessary.

5. Select, one after the other, the third, fourth and fifth elements to create three additional dimensions within the stacked dimensions system.



6. Click in the free space to validate and end the dimension creation. You can notice that the stacked dimension values are aligned.



You can set the dimension properties in the **Dimension Properties** toolbar as described in Creating Dimensions.

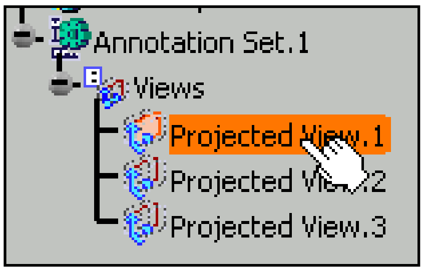
Creating Cumulated Dimensions

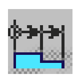
This task shows how to create cumulated dimensions. See Dimension Units for more information on dimension unit display.

Open the Tolerancing_Annotations_10 CATPart document.

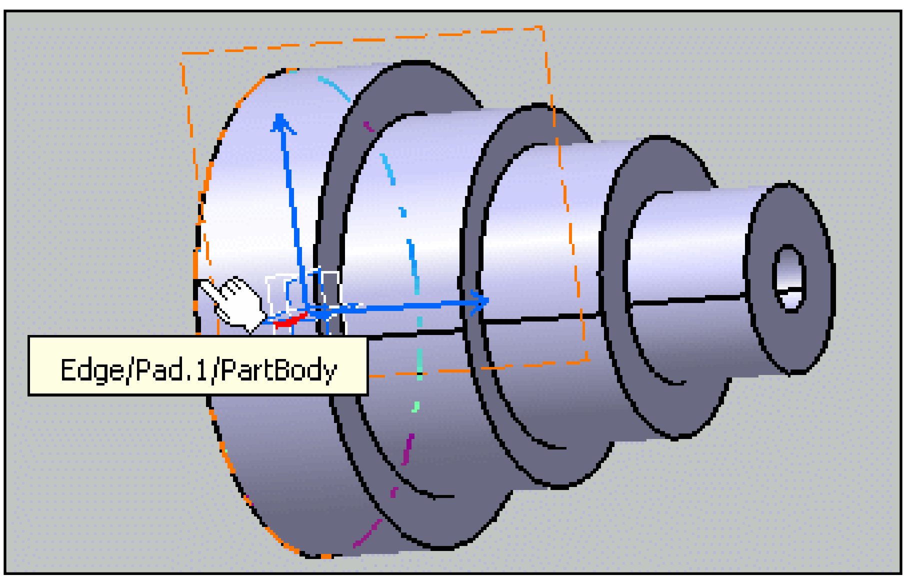
Select **Tools -> Options**. In the **Mechanical Design** category, select the **Functional Tolerancing & Annotation** sub-category, then the **Dimension** tab and check **Align cumulated dimension values** and optionally select **Automatically add a funnel**.

1. Activate the **Projected View.1** annotation plane.

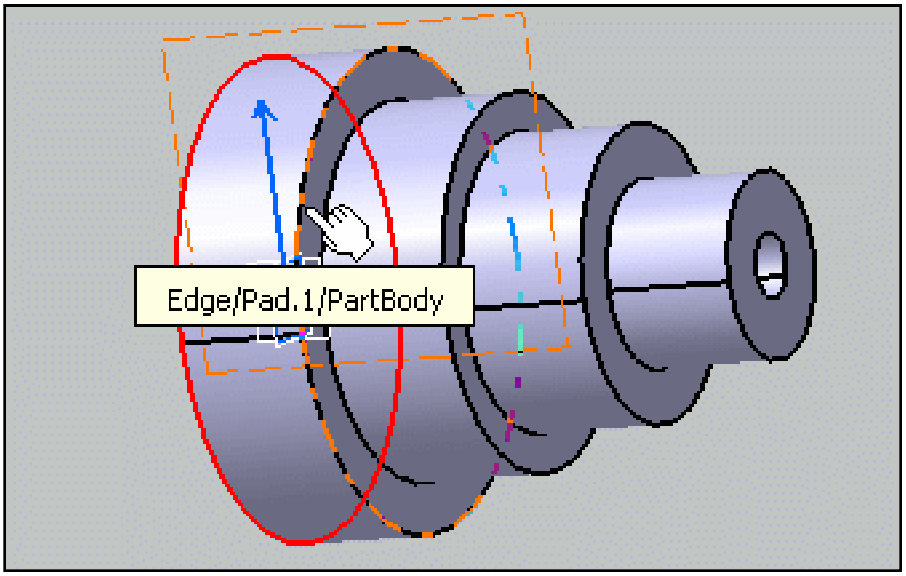


2. Click the **Cumulated Dimensions** icon. 

3. Select the first element.

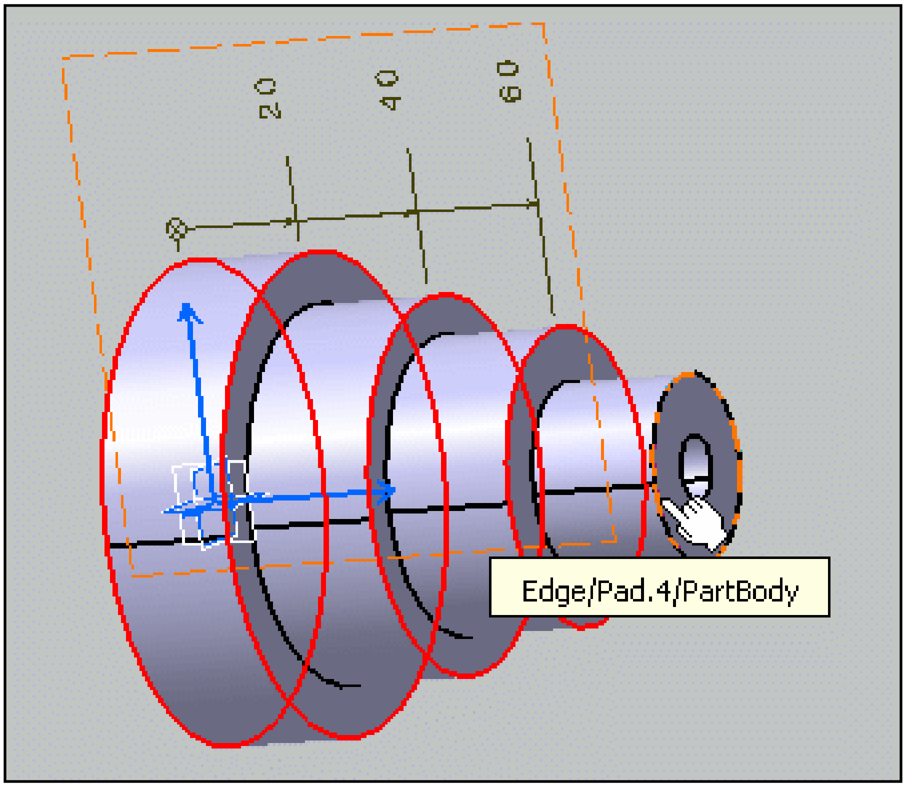


4. Select the second element.

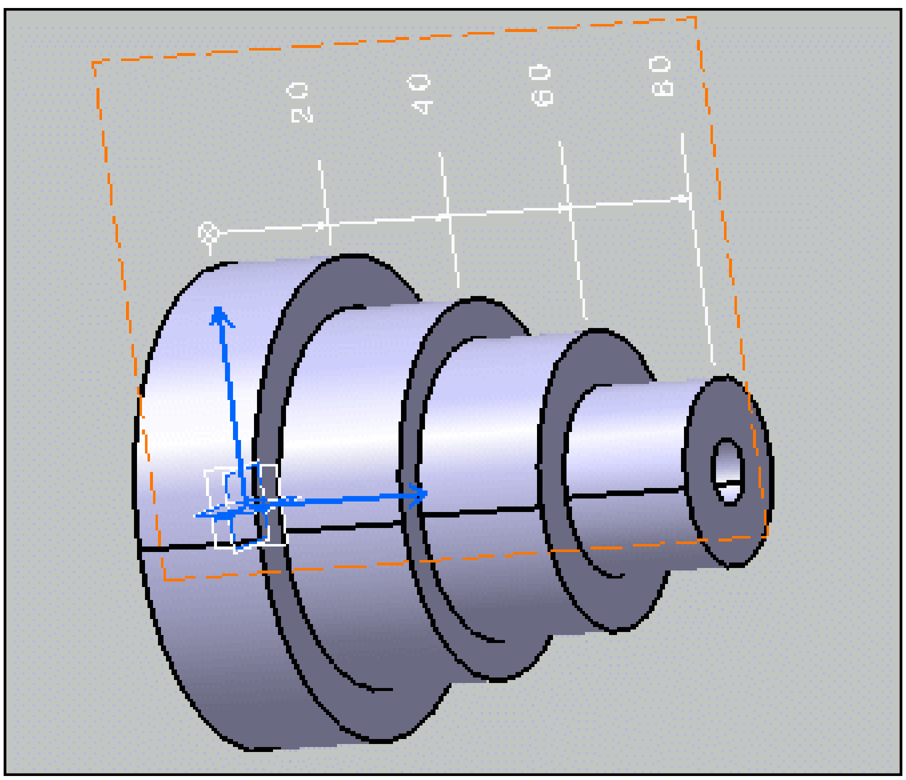


You just created a first dimension within the cumulated dimensions system. You can then proceed to create as many dimensions as necessary.

5. Select, one after the other, the third, fourth and fifth elements to create three additional dimensions within the cumulated dimensions system.



6. Click in the free space to validate and end the dimension creation. You can notice that the cumulated dimension values are aligned.



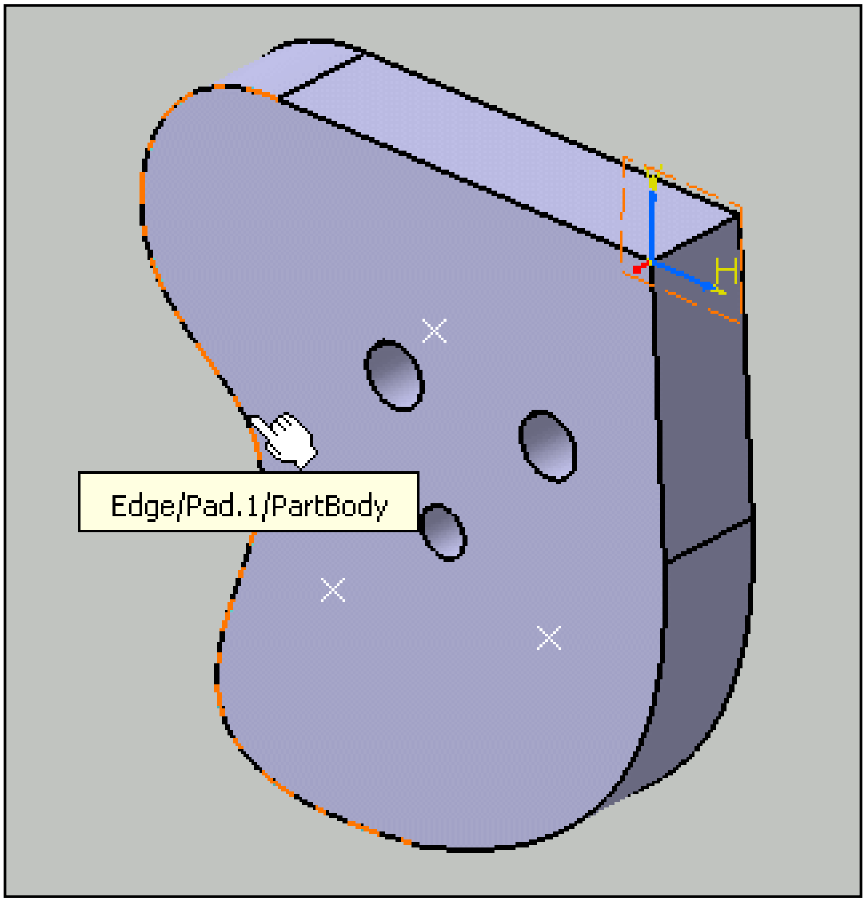
You can set the dimension properties in the **Dimension Properties** toolbar as described in Creating Dimensions.

Creating Curvilinear Dimensions

This task shows how to create curvilinear dimensions. A curvilinear dimension measures the overall length of a curve. See Dimension Units for more information on dimension unit display.

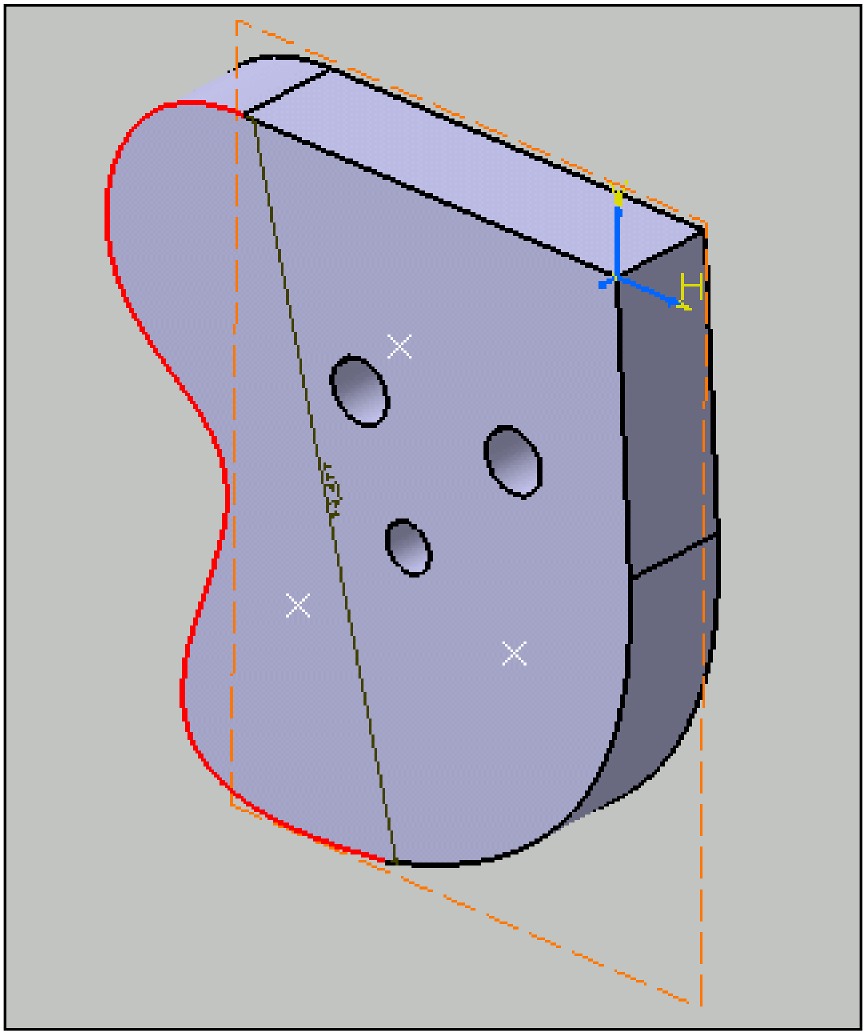
Open the Tolerancing_Annotations_11 CATPart document.

1. Click the **Curvilinear Dimensions** icon. 
2. Select a curve or an edge as shown below.



With an Electrical Harness Installation license, you can also select starting and ending points, planar faces or splines, which lets you define the total length of several harness routing curves. For more information on how the 3D Tolerancing and Annotations workbench integrates to the Electrical Harness product, refer to the *Electrical Harness Installation User's Guide*.

The dimension is previewed. By default, the dimension line is linear.

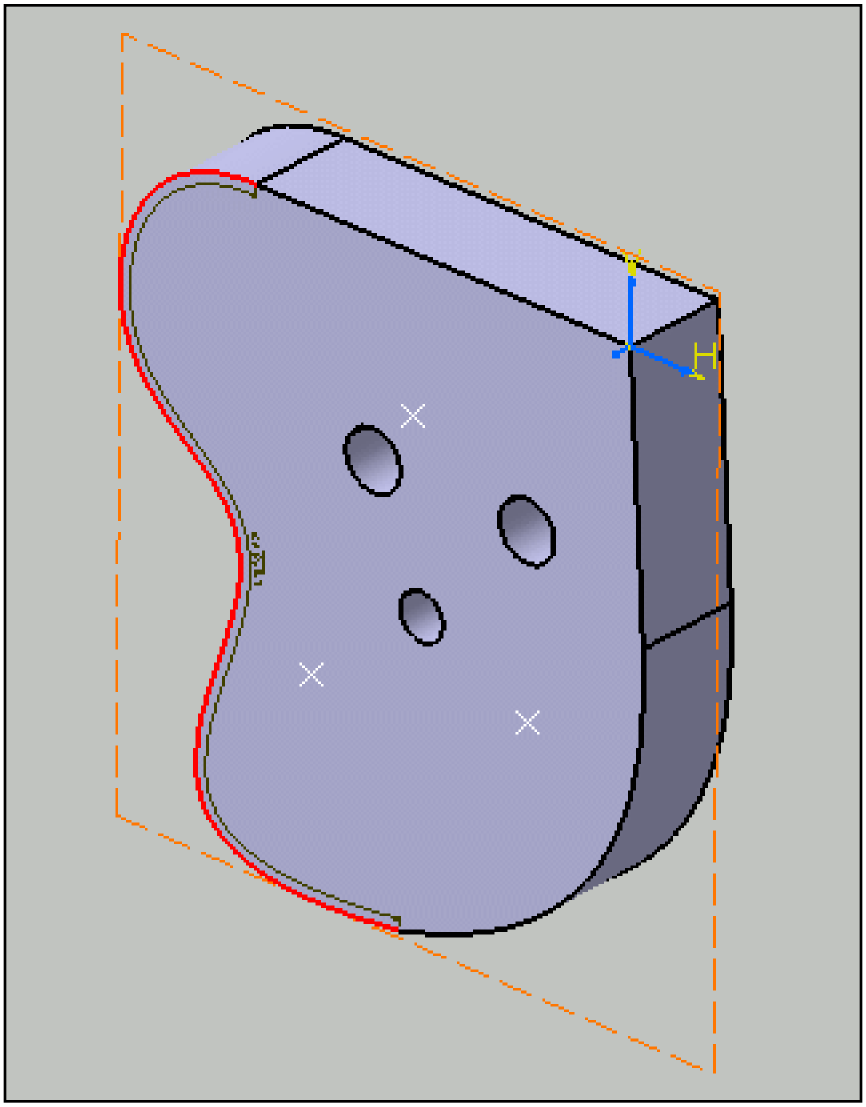


4. Right-click to display the contextual menu.

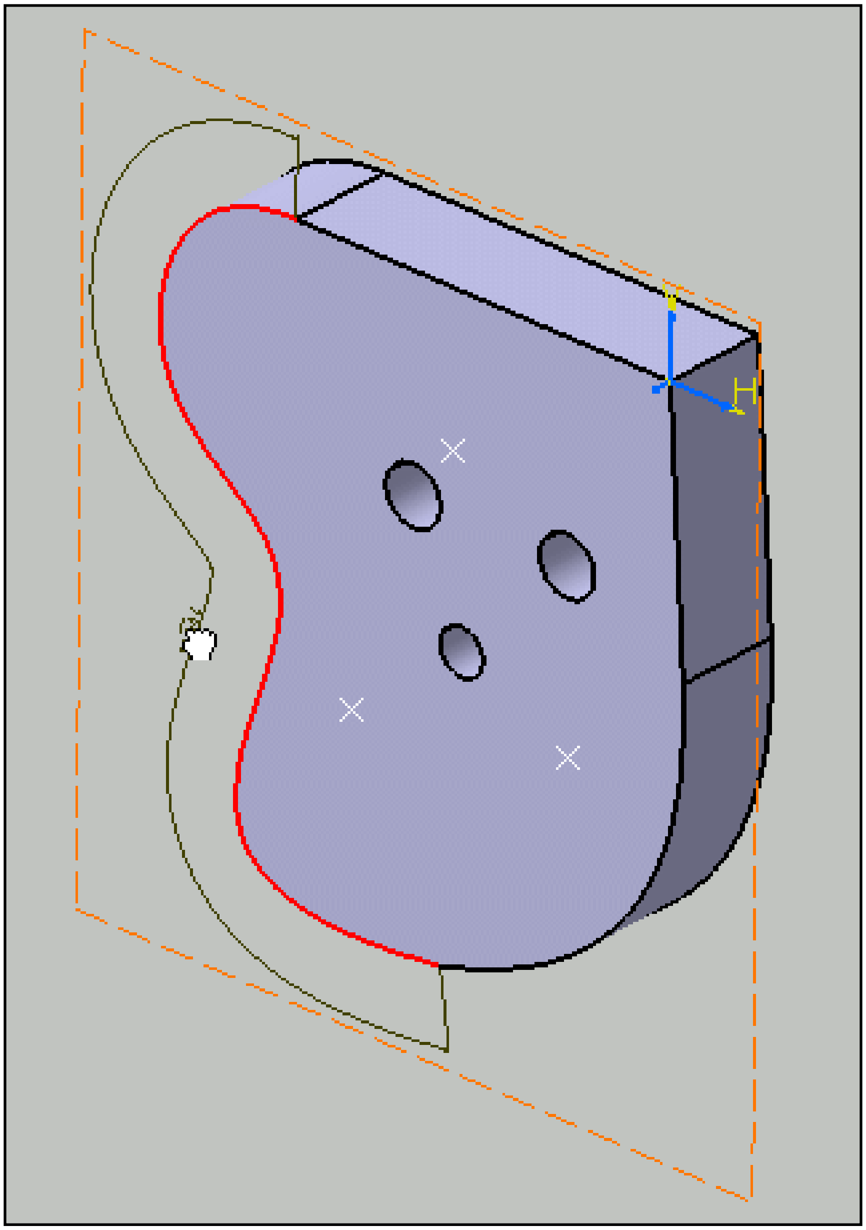
5. Select a representation mode for the dimension line:

- **Offset** displays the dimension line as an offset curve of the projection of the measured curve.
- **Parallel** displays the dimension line as a parallel curve of the projection of the measured curve.
- **Linear** displays the dimension line as a straight line, parallel to the direction defined by the limit points of the measured curve.

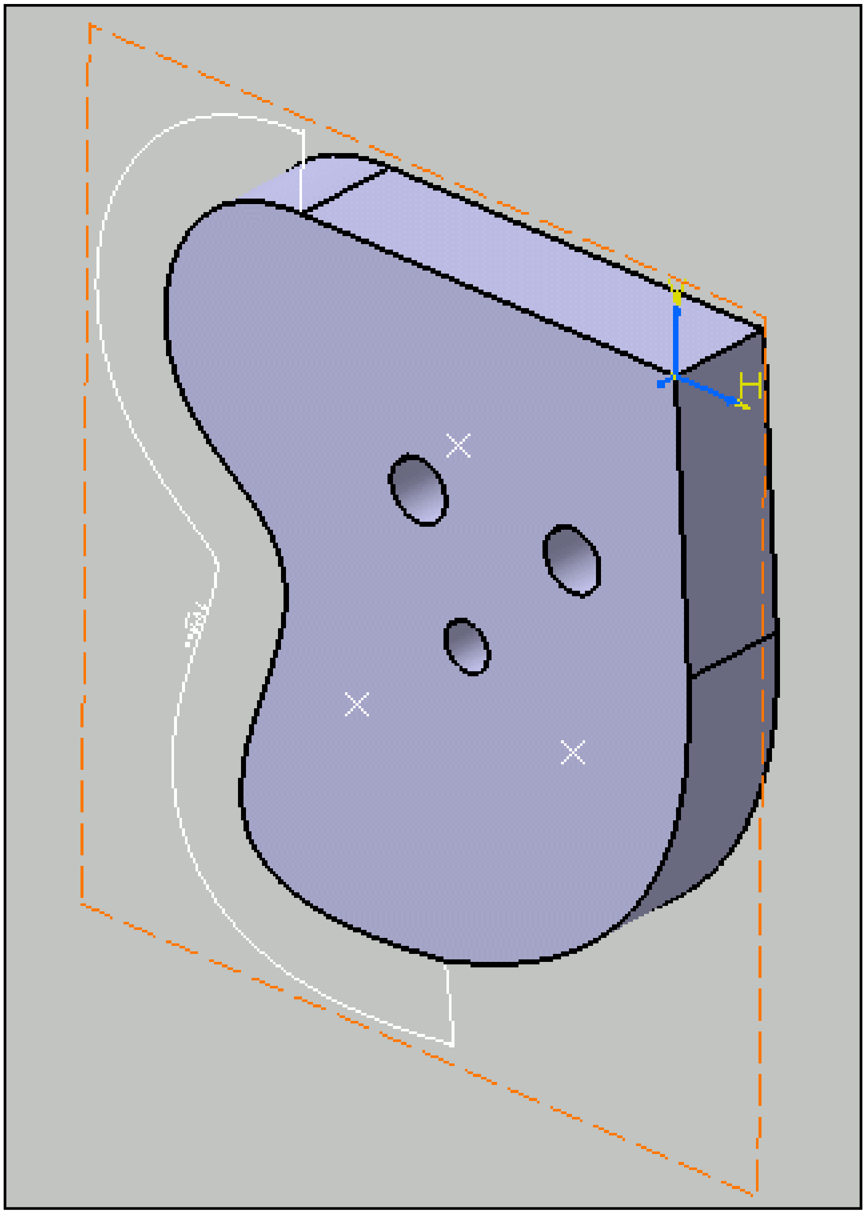
Select **Offset** for example. The dimension line is modified accordingly.



6. Optionally drag the dimension line and/or the dimension value to position them as wanted.



7. Click in the free space to validate and end the dimension creation. The semi-arc symbol displayed over the dimension value symbolizes a curvilinear length dimension. You can now handle the dimension just like any other dimension.



More about curvilinear dimensions

- You can set the dimension properties in the **Dimension Properties** toolbar as described in Creating Dimensions.
- The curvilinear length symbol is defined in the standards.
- You cannot measure only part of a curve.
- In some cases, depending on the curve and on the offset value, the offset representation mode cannot be computed (because the dimension line cannot intersect with itself along the curve): you will not be able to position the dimension further than a certain limit.

- You cannot change the dimension line representation mode after the dimension has been created.

Generating Dimensions

This task shows you how to generate dimensions automatically.

This command allows you to generate the following dimensions:

- Some or all hole dimensions from a hole parameters (except if these parameters cannot be associated with an existing geometrical element to generate dimensions: diameter of a tapered hole out of the part, depth for a blind hole setting a "through hole", etc.)
- Pad/Pocket length; if two limits are defined, two dimensions will be generated between the sketch plane and each limiting face, and sketch constraints.
- Multi-Pad/Multi-Pocket lengths, and sketch constraints.
- Shaft/Groove distance constraints that are parallel or perpendicular to the shaft/groove axis direction, distance between shaft/groove axis and point and/or straight line and/or circle (arc or complete) as a half dimension diameter if the shaft/groove is not complete (value of the single angle or total of the 2 less than 360°), as regular diameter if the shaft/groove is complete, angle, and sketch constraints.
- Part Design Chamfer features; dimensions are generated according to the Part Design chamfer feature definition: for a chamfer with a length x length definition, the dimension format will be distance x distance, whereas for a chamfer with a length x angle definition, the dimension format will be distance x angle dimension. However, you should be aware of the fact that if tolerances are applied to chamfer parameters, only the tolerance applied to the first parameter will be generated. Chamfer dimensions cannot be edited, but they can be modified via the **Dimension Properties** toolbar and the **Edit Properties** command.
- Part Design Thread features; thread diameter, depth and pitch parameter dimensions can be generated. Dimension generation automatically generates a thread symbolic representation.
  Note that:
  - any tolerance attached to the pitch or diameter parameter will not be generated.
  - generating a pitch parameter dimension requires the generation of the corresponding thread diameter parameter dimension.
  - if the thread is not a metric one, the prefix will be a diameter symbol instead of the letter M.
- You could select **User Feature** and explore them in order to find Part features allowed in Generative command.
  The command will recursively search each compatible internal Part features (in the same way **User Feature** containing other **User Features**) in order to show parameters (if nothing could be analysed by generative command, no parameter will appear).
  Moreover, user can immediately select all of them in generative command list as if User Feature does not exist.
  All parameters (published or not during User Feature creation) are seen in list and corresponding dimensions could be now created.

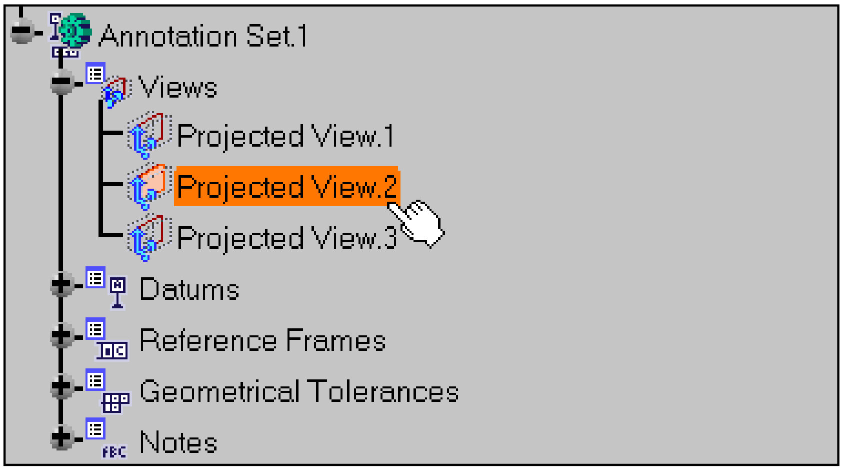
Remember that:

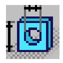
- Dimensions are associated with the design of a part, including the Mean Dimension behavior.
- When parameter tolerances are still defined, they are set to the dimension tolerances.
- Modifying dimension tolerances modifies the parameter tolerances and vice-versa.

Open the Annotations_Part_04.CATPart document:

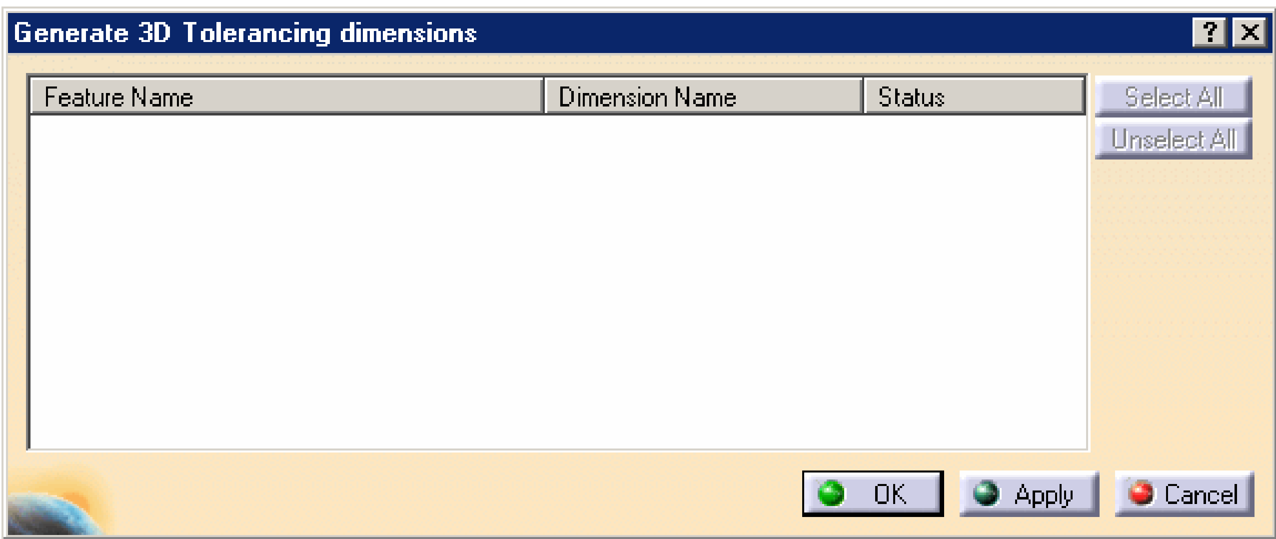
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Activate the **Projected View.2** annotation plane.

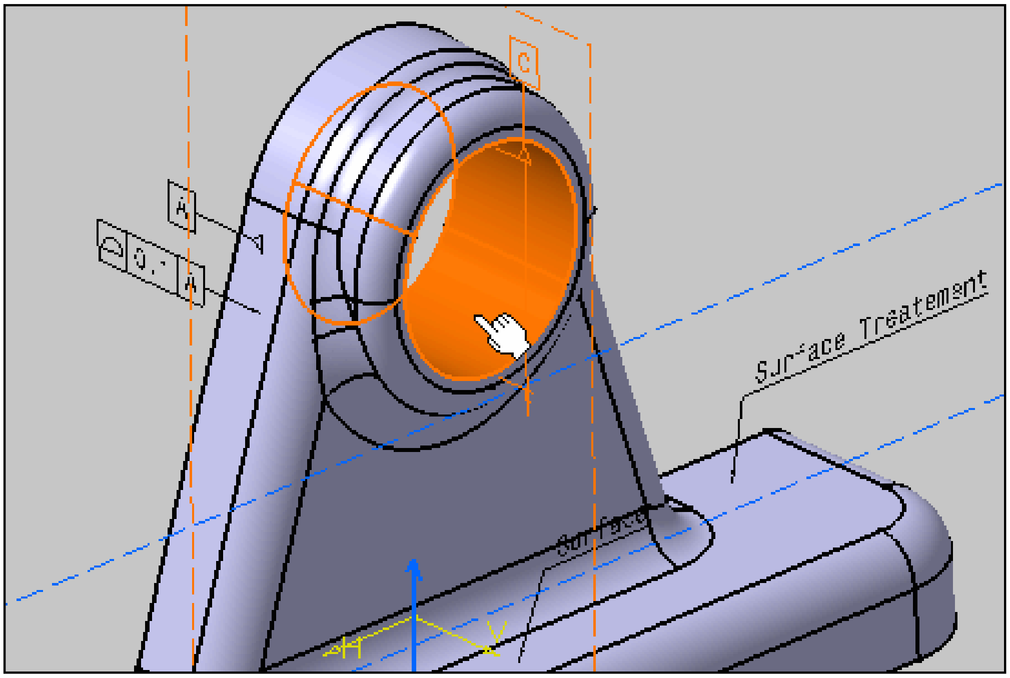


2. Click the **Generative Dimension** icon: 

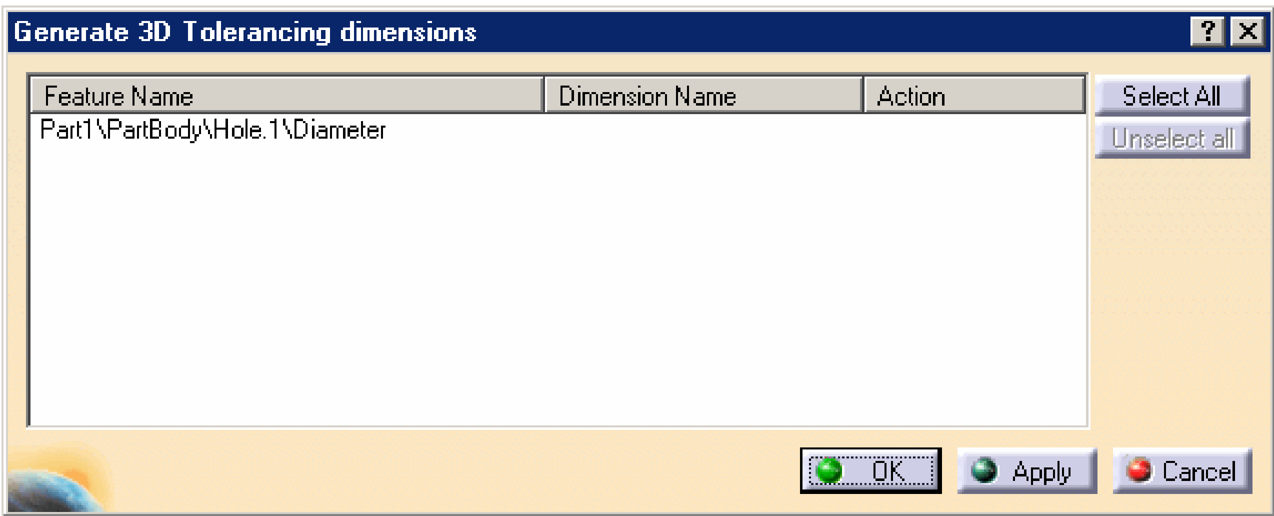
The **Generate 3D Tolerancing Dimensions** dialog box appears.



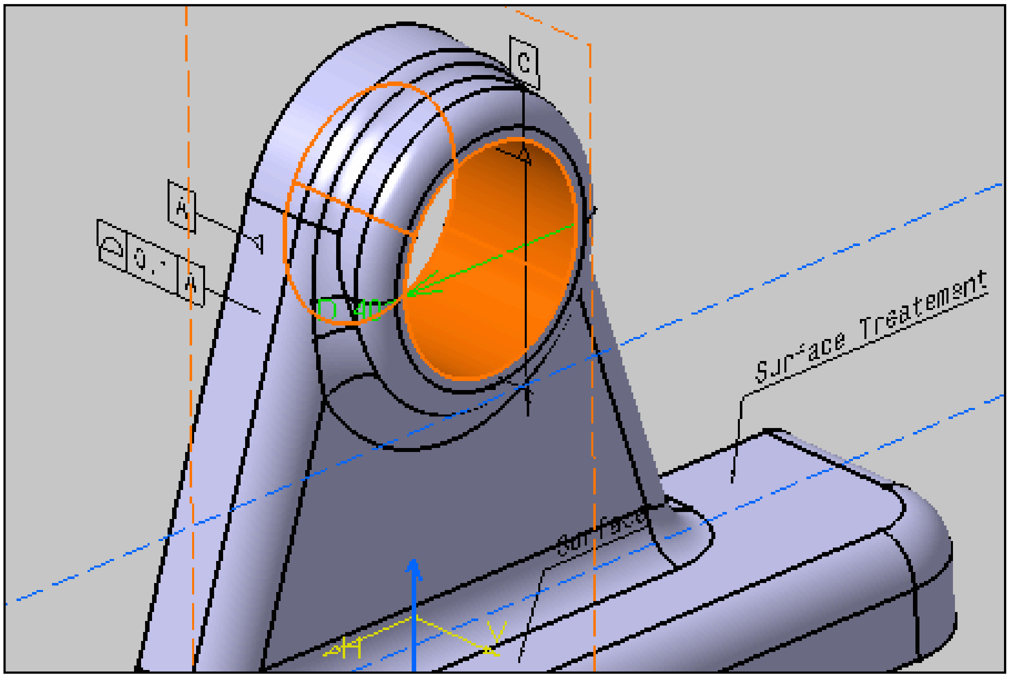
3. Select the hole as shown on the part.



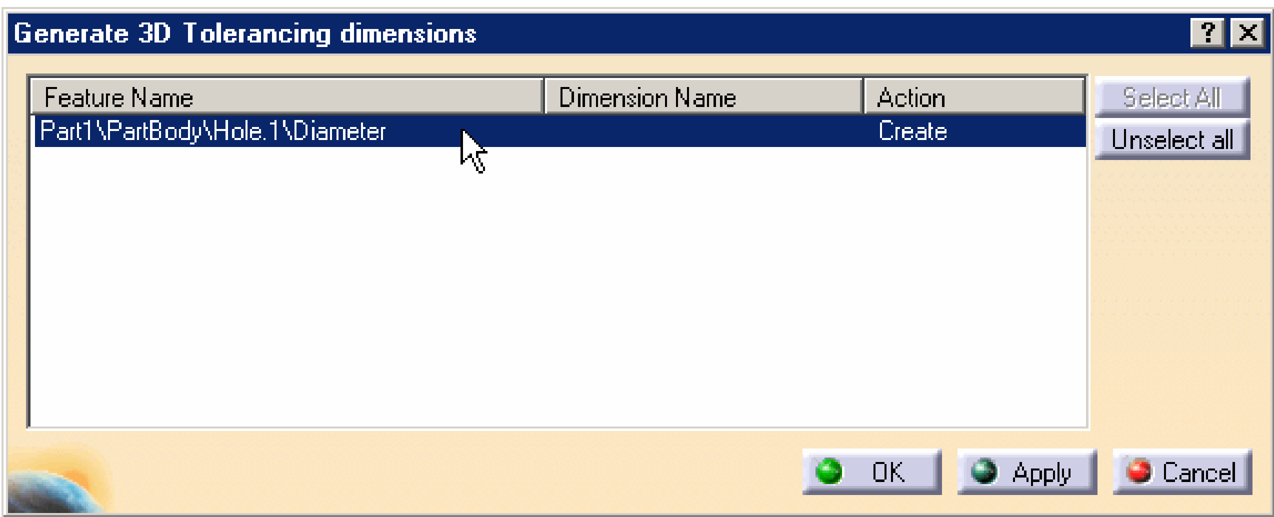
The hole parameter is displayed in the dialog box.



Also, the hole parameter is shown on the part.

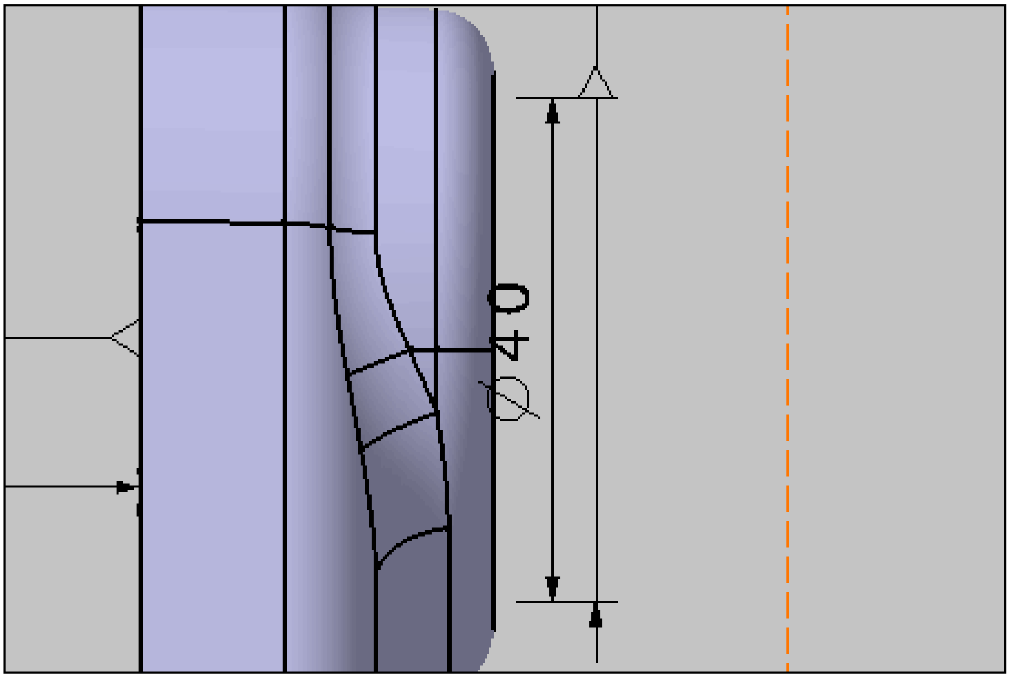


4. Select the **Diameter** parameter in the dialog box. You can also select the parameter on the geometry.

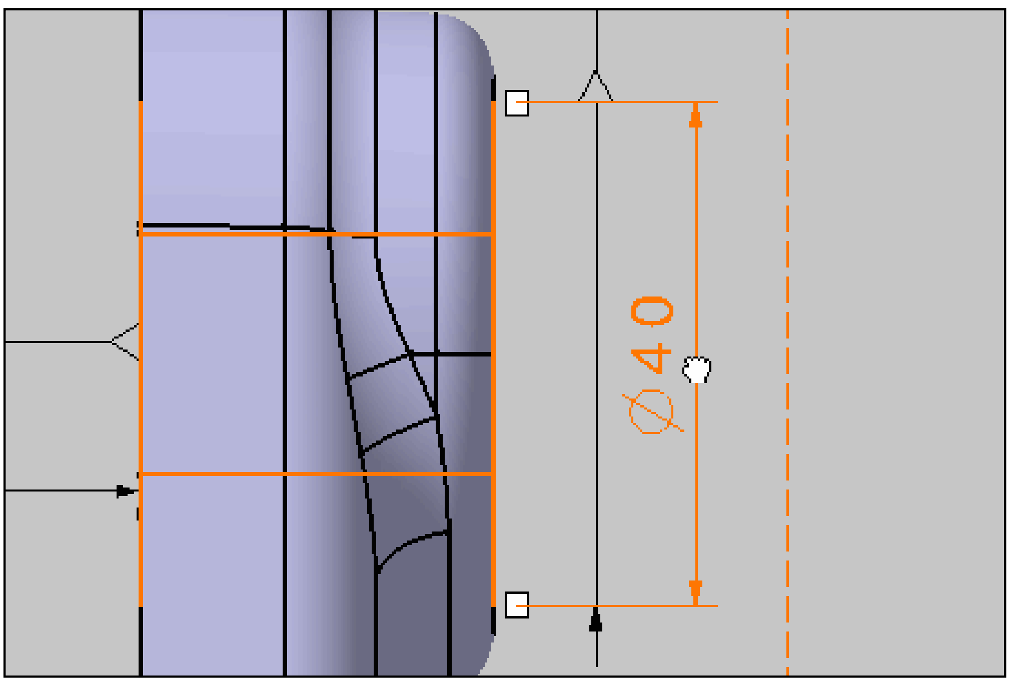


5. Click **OK** in the dialog box.

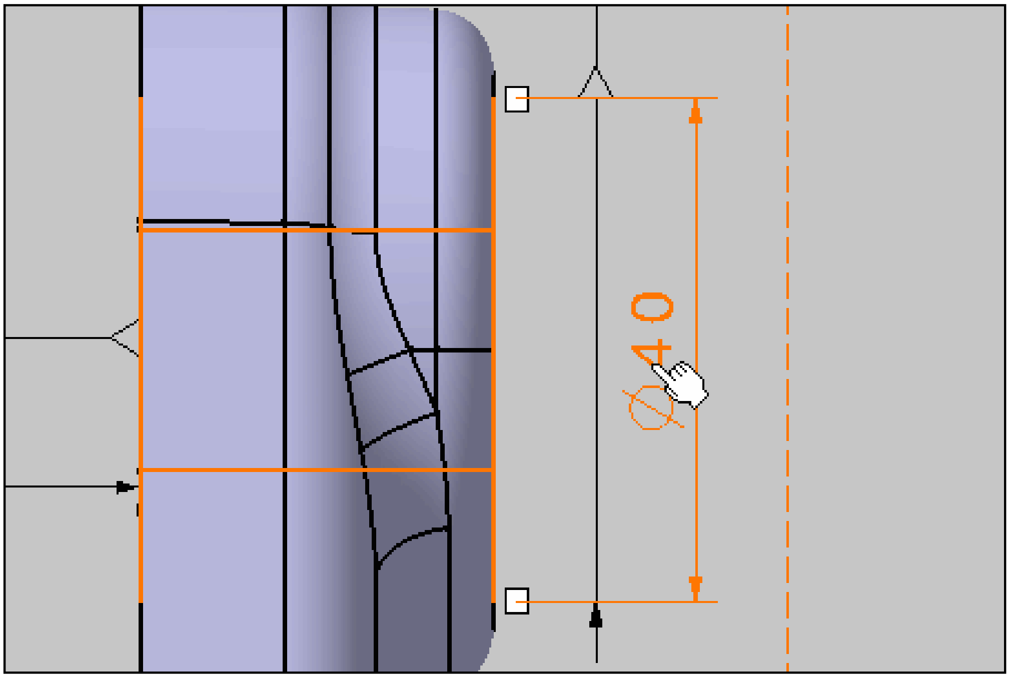
The diameter dimension is created.



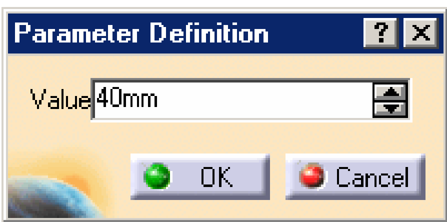
6. Drag the dimension.



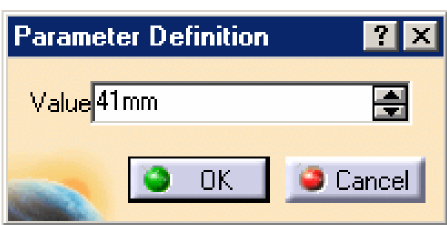
7. Right-click the dimension and select **Edit Generative Parameter** from the contextual menu.



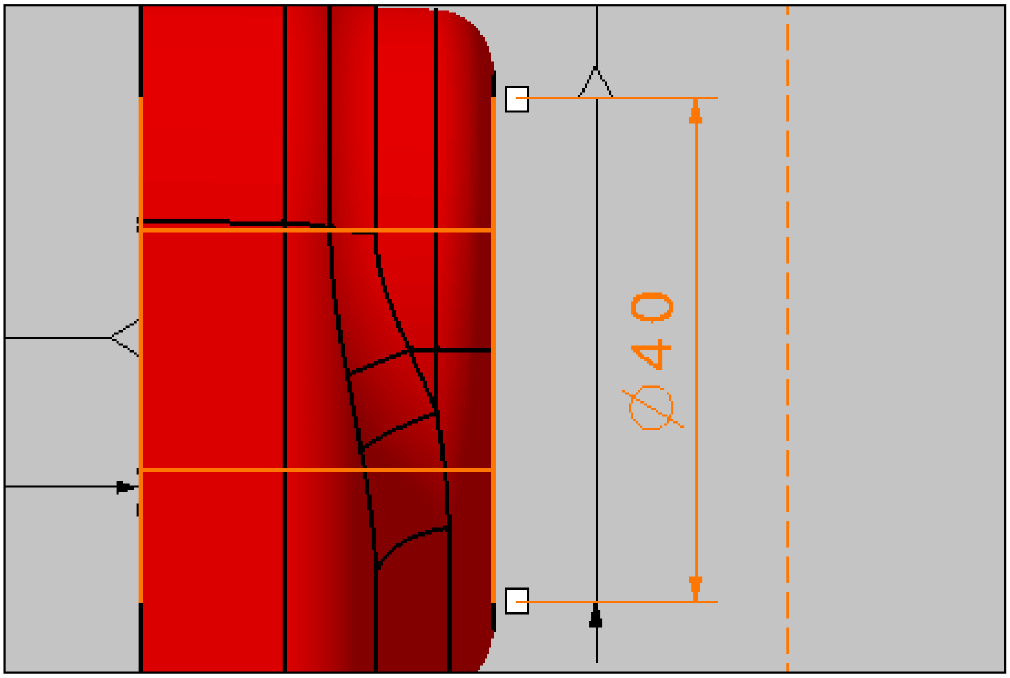
The **Parameter Definition** dialog box appears.
This dialog box allows you to modify the hole parameter and edit the part.

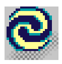


7. Set the value to 41mm and click **OK**.

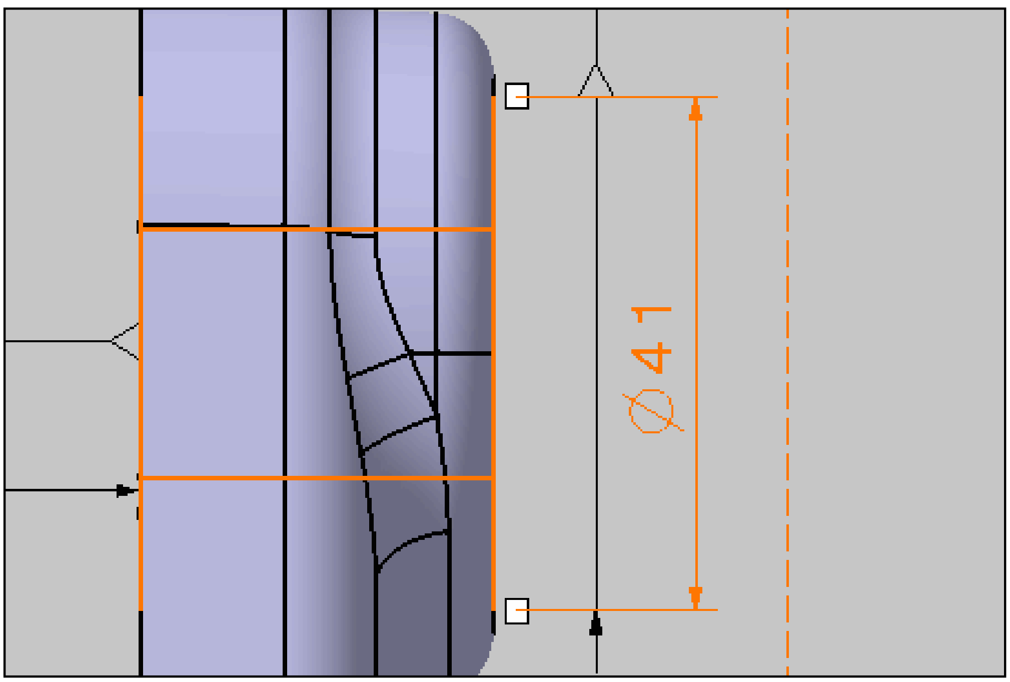


The part turns red because it has been modified but not updated.



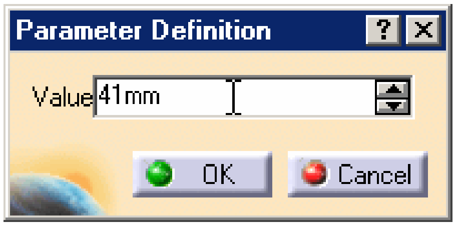
8. Click the **Update** icon: 

The part's hole and diameter dimension are modified.

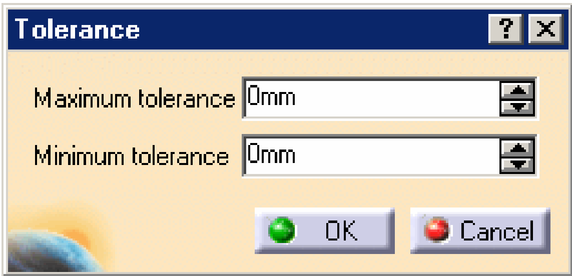


9. Right-click the dimension and select **Edit Generative Parameter** from the contextual menu again.

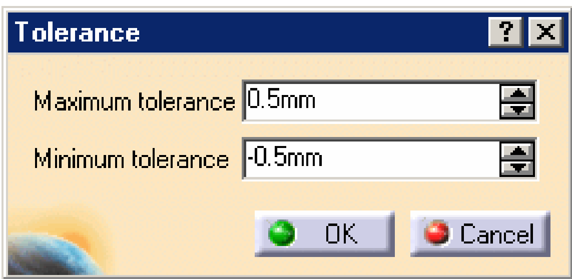
10. Right-click in the value field and select the **Add Tolerance...** from contextual menu.



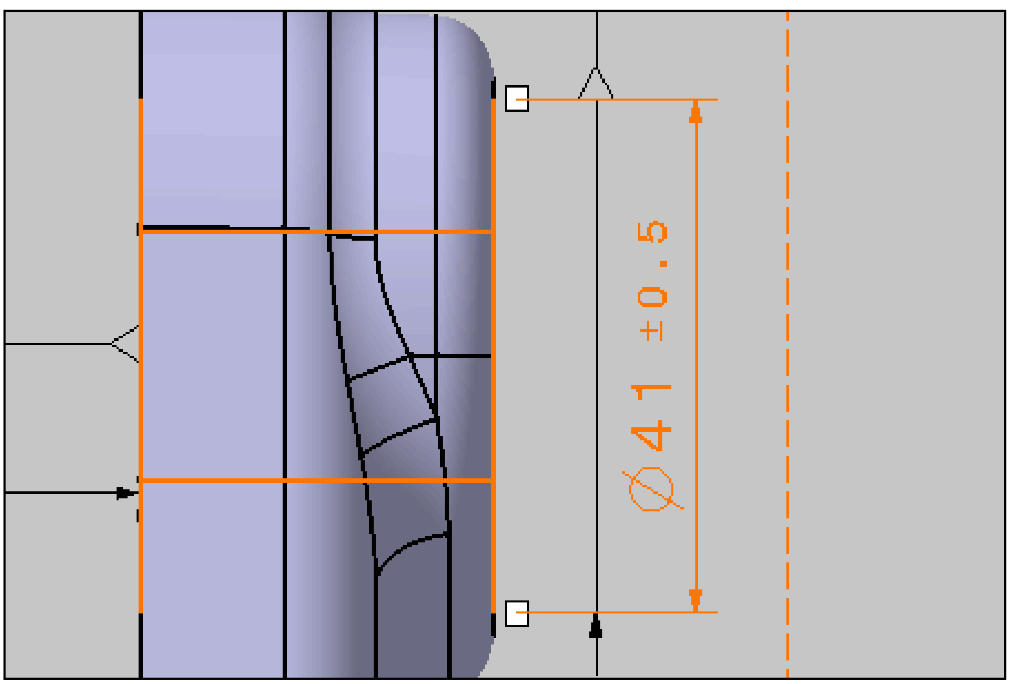
The **Tolerance** dialog box appears.
This dialog box allows you to modify the hole parameter tolerances.



11. Set the tolerances to 0.5 and -0.5

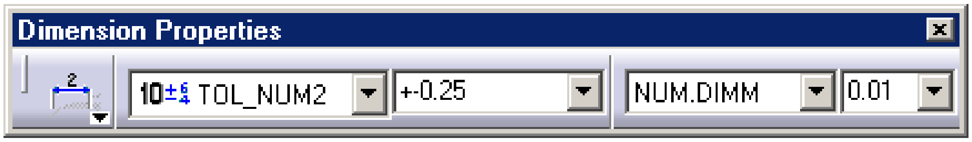


Tolerances are displayed on the dimension.

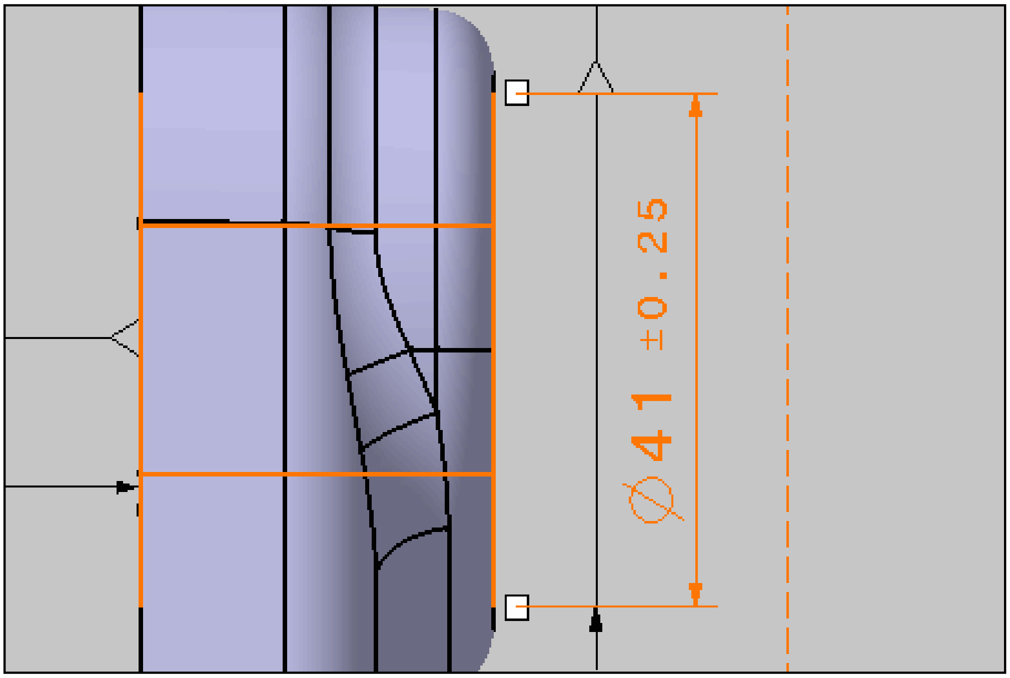


12. Click **OK** in the **Tolerance** and **Parameter Definition** dialog boxes.

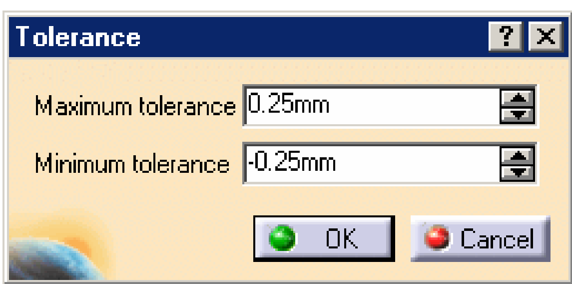
13. Modify the tolerances in the **Dimension Properties** toolbar to +-0.25.



Tolerances are updated on the dimension.



The parameter's tolerances are also updated.

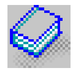


Instantiating a Note Object Attribute

This task will show you how to instantiate a Note Object Attribute (NOA) annotation and modify its comments and hyperlinks.
See Creating Note Object Attribute task and Note Object Attribute concept.

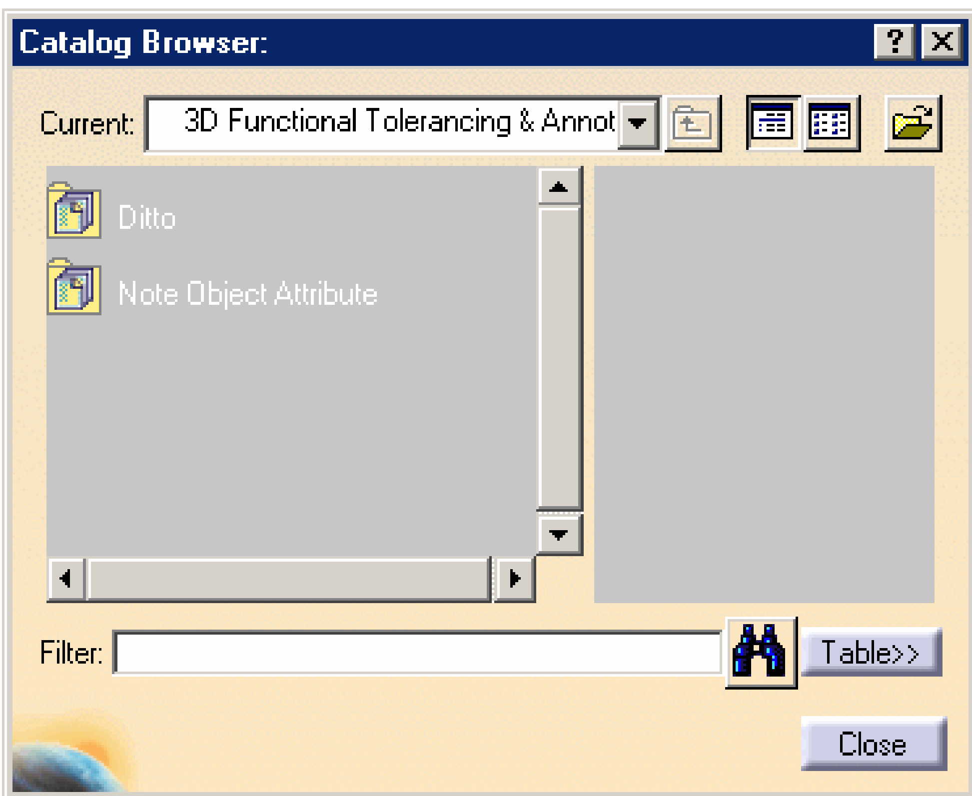
Open the Tolerancing_Annotations_01 CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Click the **Display with Browser** icon: 

   The **Catalog Browser** dialog box appears.

2. Click the **Open File** icon and select for the Component catalog document.

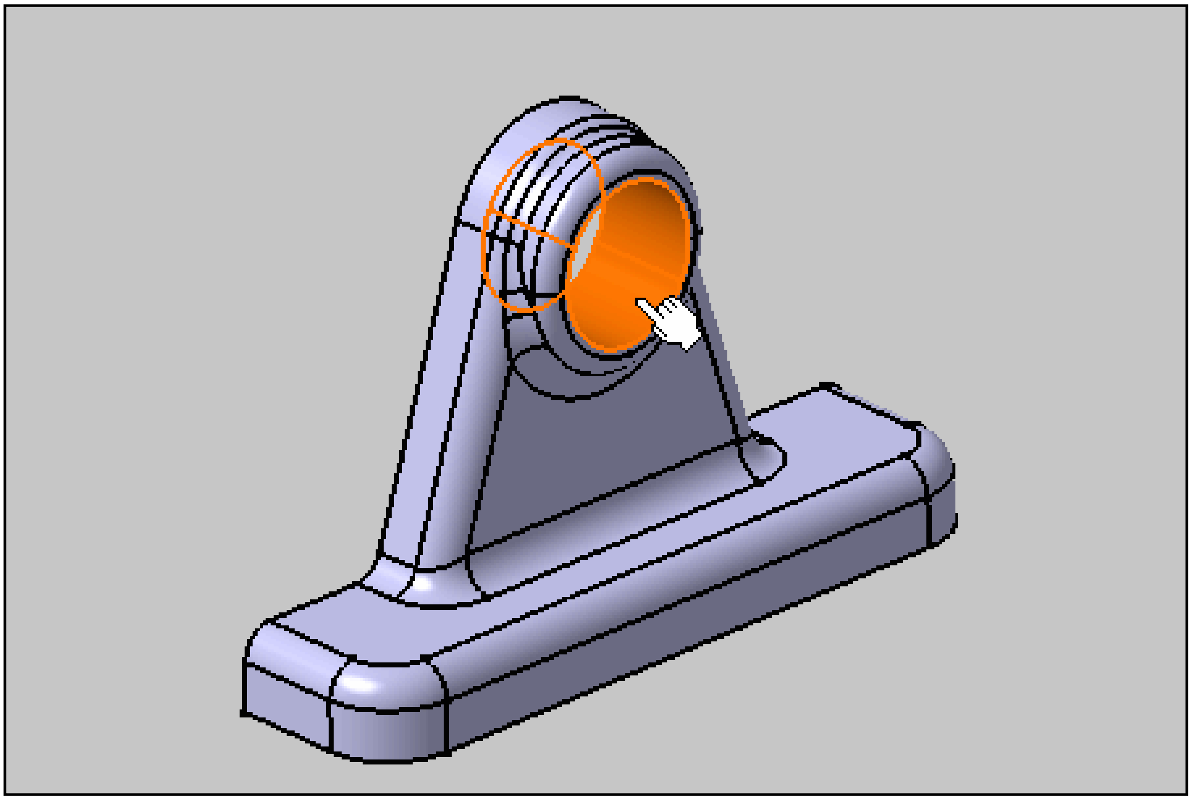


3. Double-click the **Note Object Attribute** component family item.

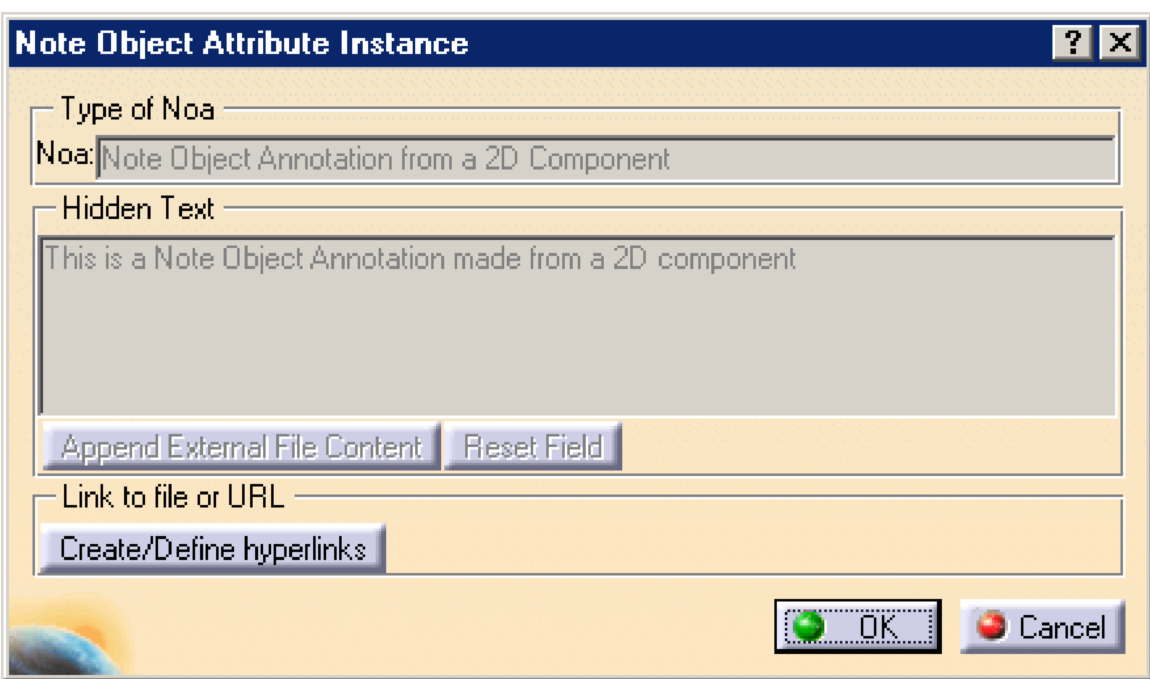
4. Double-click the **Note Object Attribute Using 2D Component** component item.

5. Select the surface as shown on the part.

This scenario illustrates the instantiation of a note object attribute by selecting geometry, but you can also select any Part Design or Generative Shape Design feature in the specification tree. In this case, the created annotation will not be attached to the selected feature, but to its geometrical elements at the highest level.

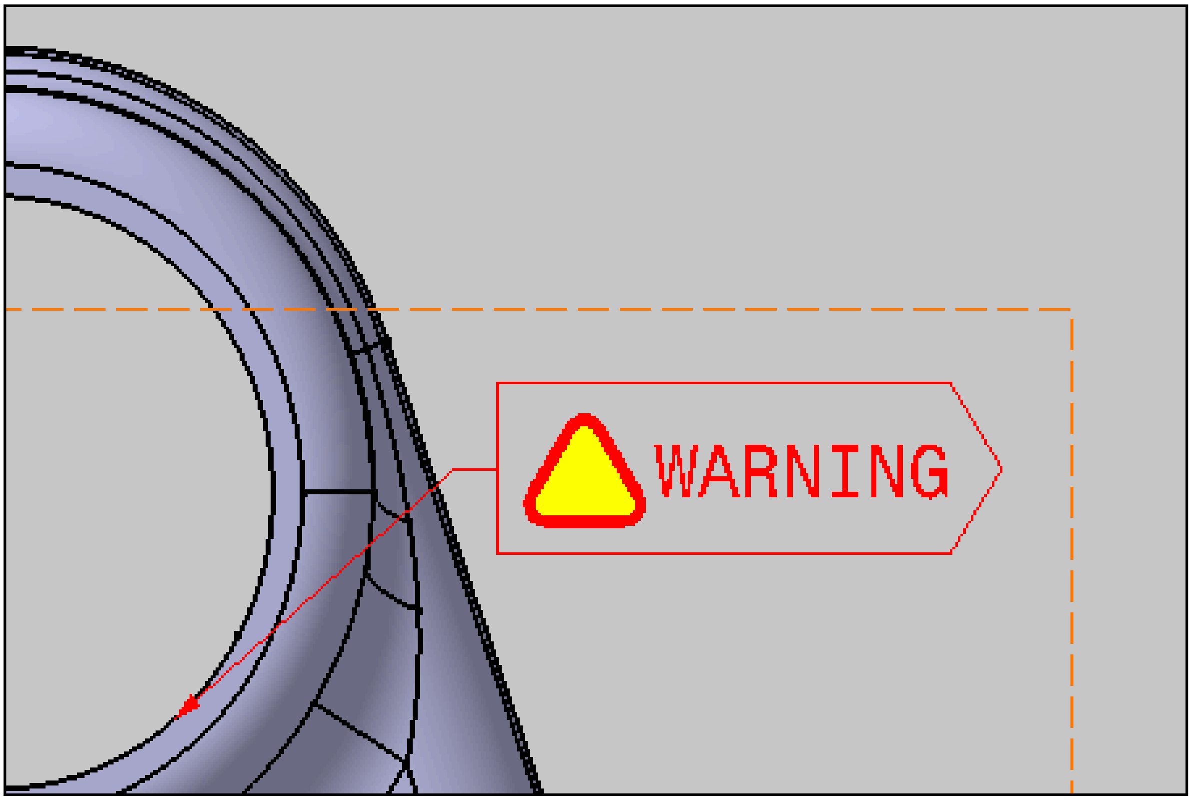


The **Note Object Attribute Instance** dialog box appears. The hidden text specified with the note object attribute is not modifiable. To unlock it, see Creating from a 2D Component; in this case, the dialog box is enabled.



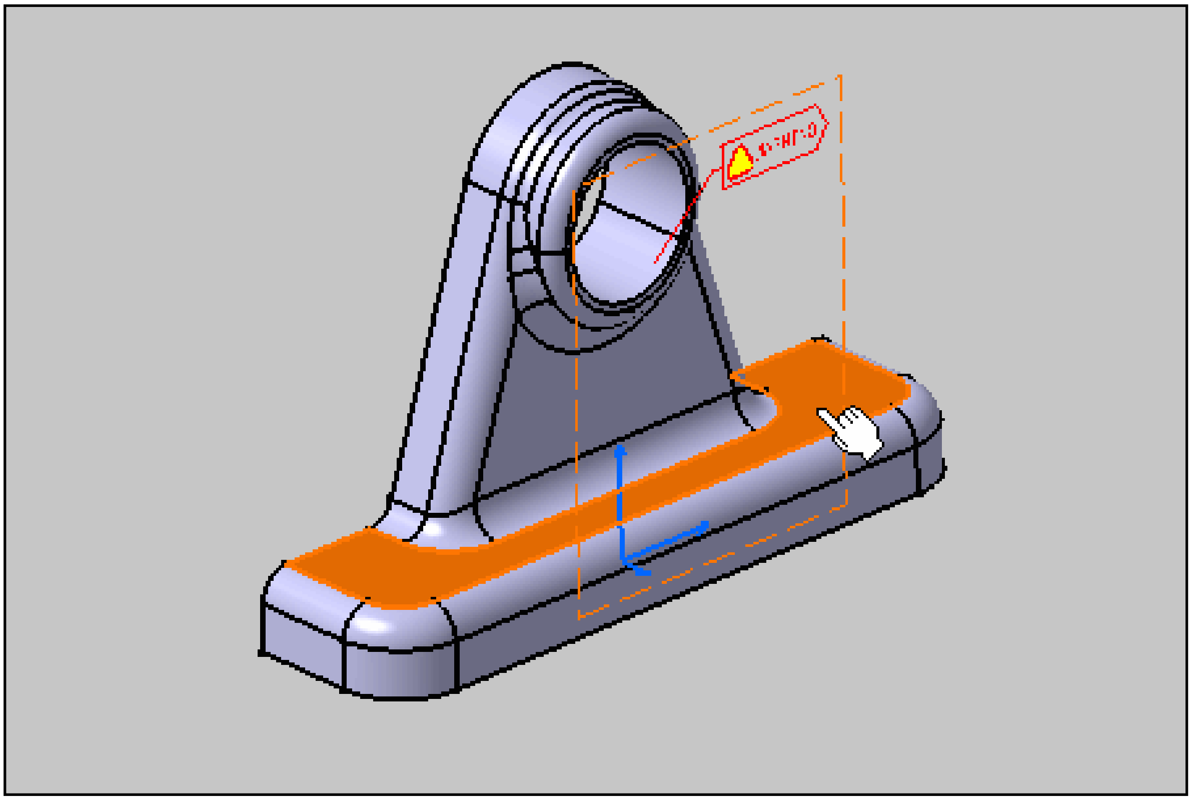
6. Click **OK**.

The Note Object Attribute is instantiated.
To customize its graphic properties see Managing Graphical Properties.

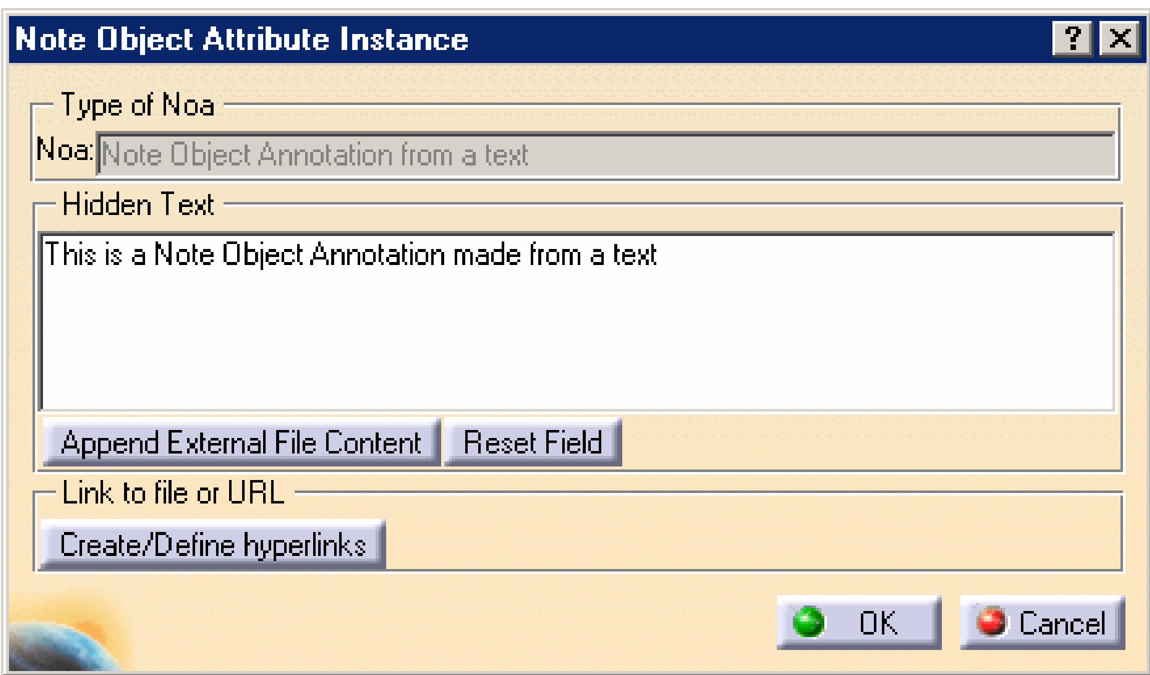


7. Double-click the **Note Object Attribute Using Text** component item in the **Catalog Browser** dialog box.

8. Select the surface as shown on the part.

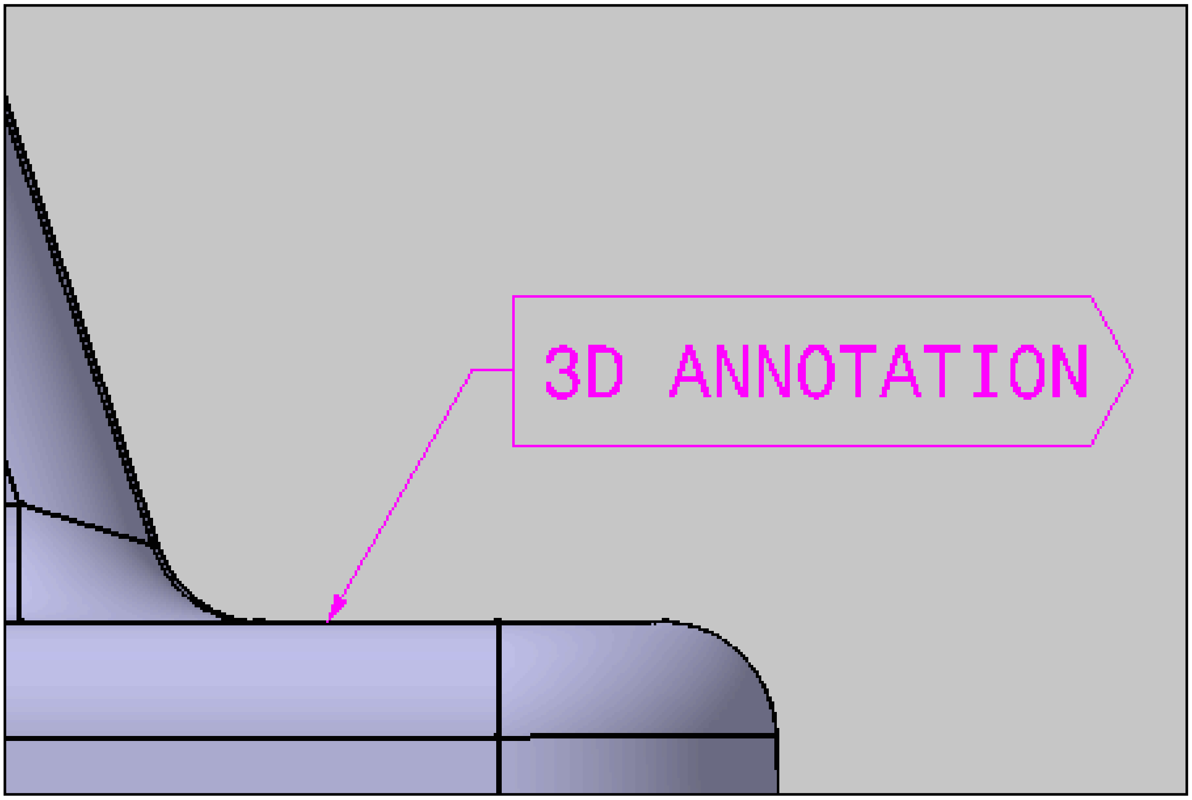


The **Note Object Attribute Instance** dialog box appears. The hidden text specified with the note object attribute is modifiable. To lock it, see Creating from a Text; in this case, the dialog box is disabled.



9. Click **OK**.

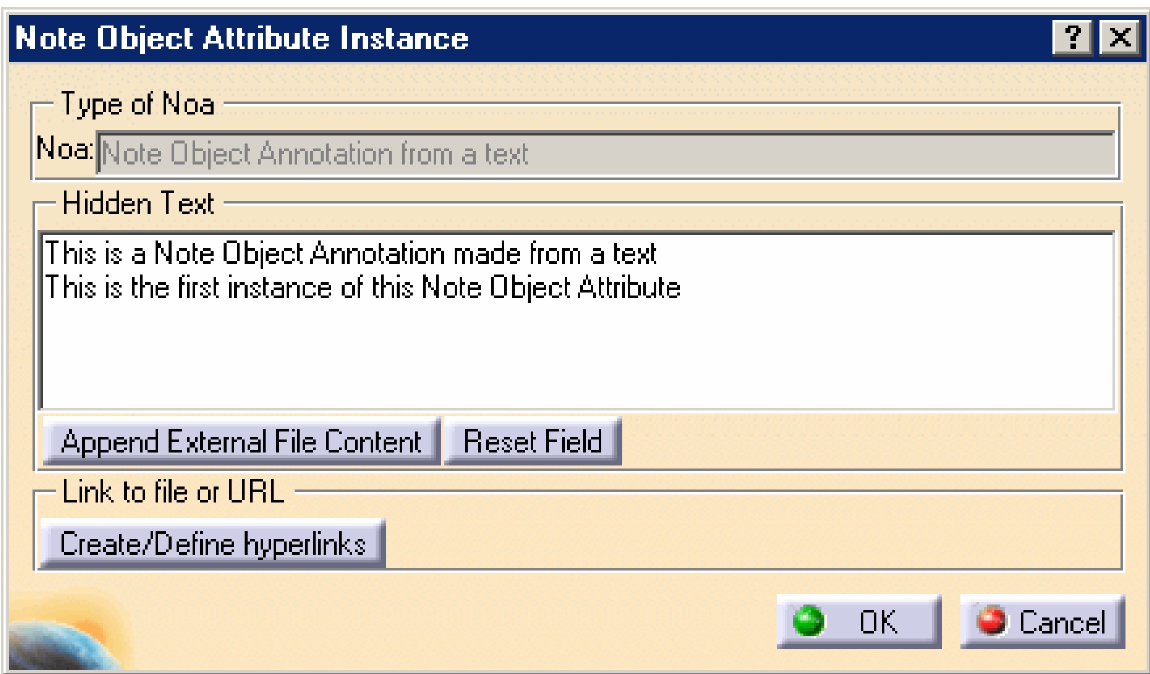
The Note Object Attribute is instantiated.



7. Double-click the **Note Object Attribute.2**.

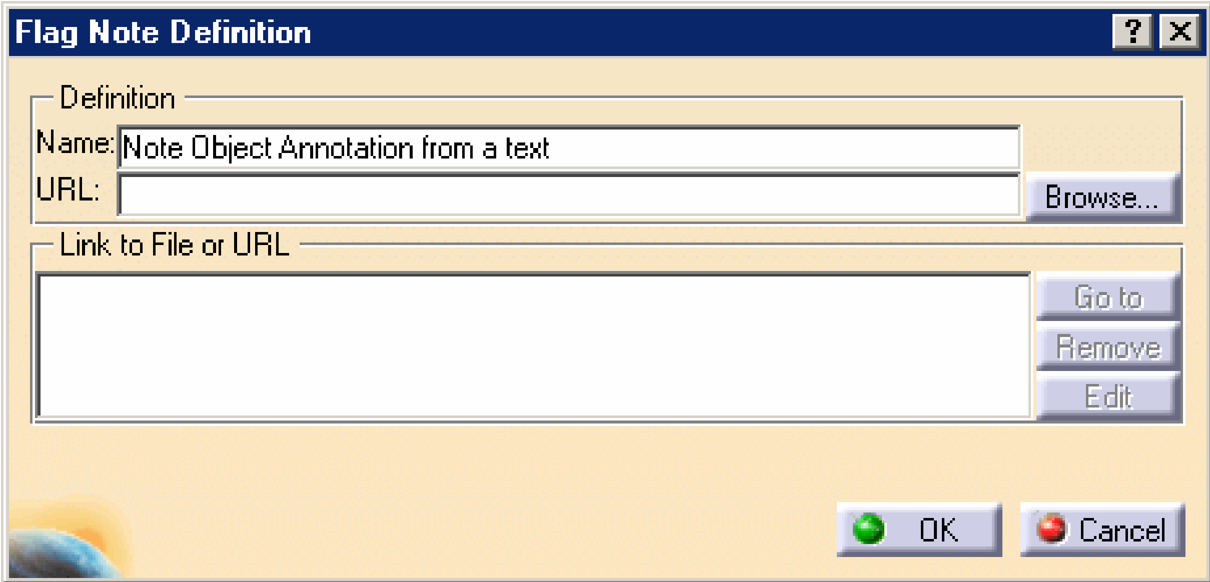
The **Note Object Attribute Edition** dialog box appears.

8. Enter the following text to modify the hidden text: **This is the first instance of this Note Object Attribute**

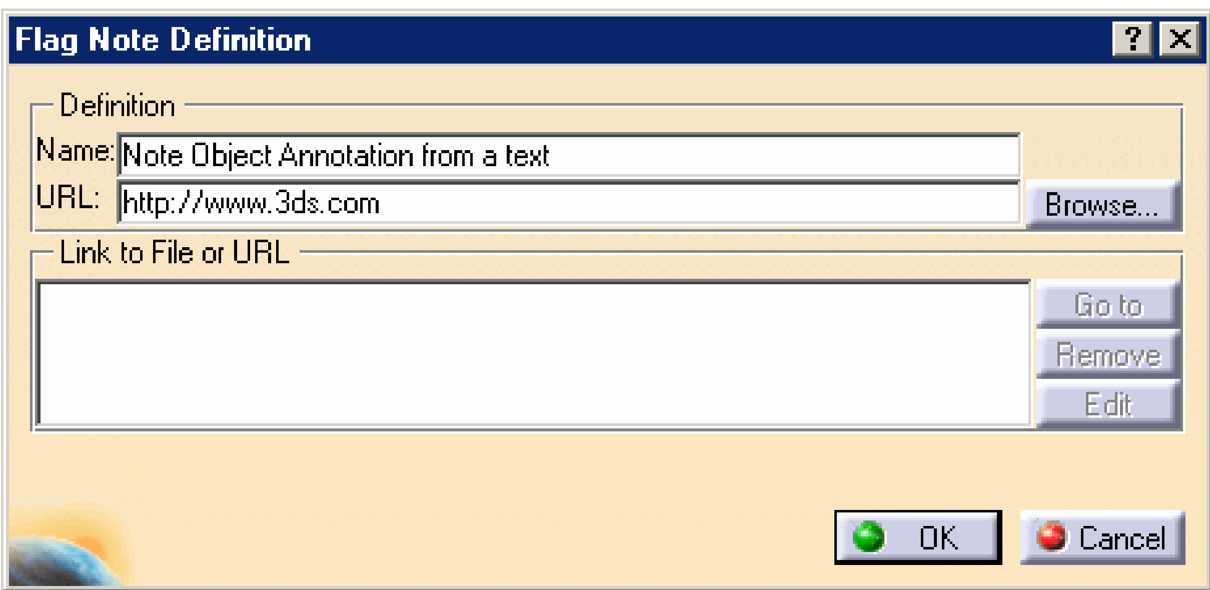


9. Click **Create/Define hyperlinks**.

The **Manage Hyperlink** dialog box appears.



10. Add the following link: http://www.3ds.com



11. Click **OK** in the **Manage Hyperlink** dialog box.

12. Click **OK** in the **Note Object Attribute Instance** dialog box.

Creating a Partial Surface

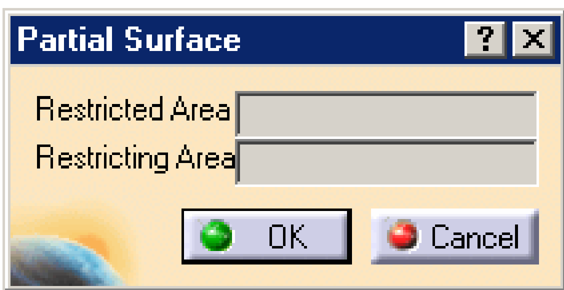
This task shows you how to create a partial surface annotation.

- A partial surface annotation allows user to define a delimited surface to be toleranced.
- The surface may be defined using Generative Shape Design workbench.
- To customize the partial surface options, see Display.

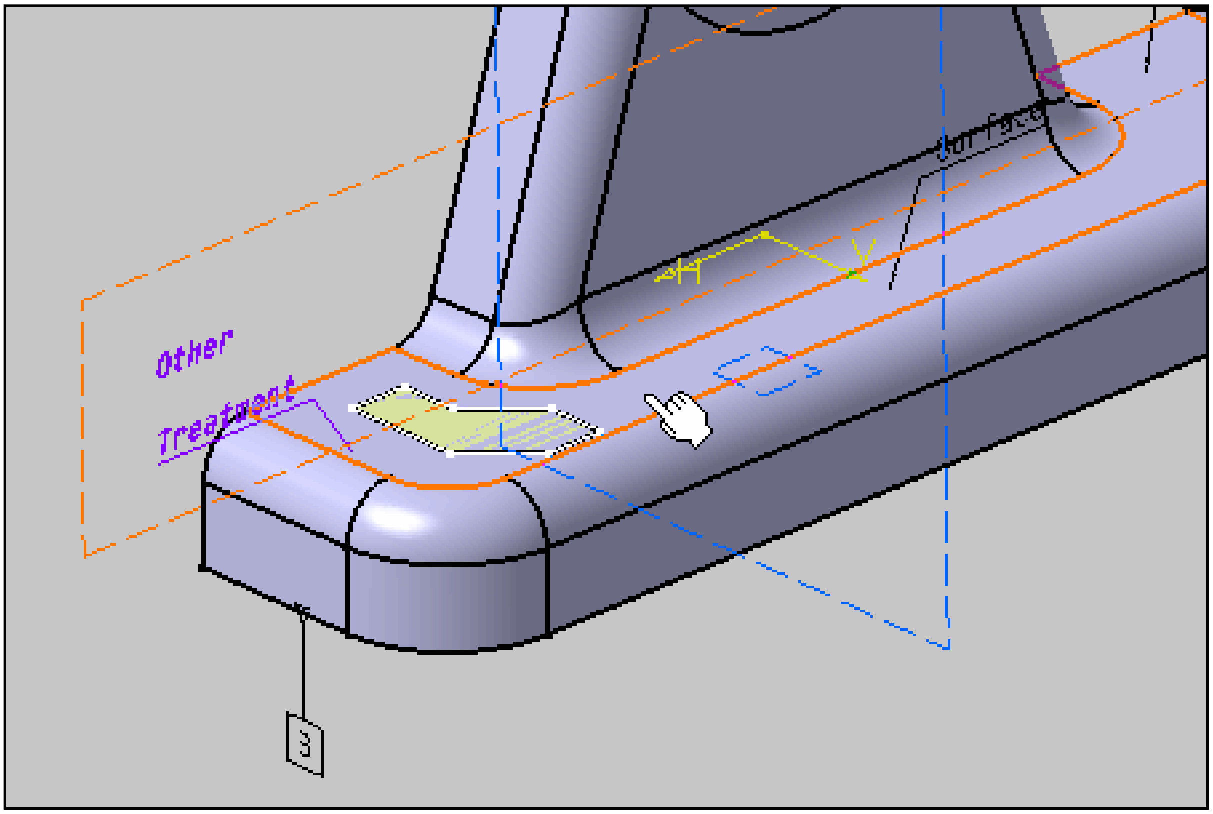
Open the Annotations_Part_04.CATPart document

1. Click the **Partial Surface** icon:

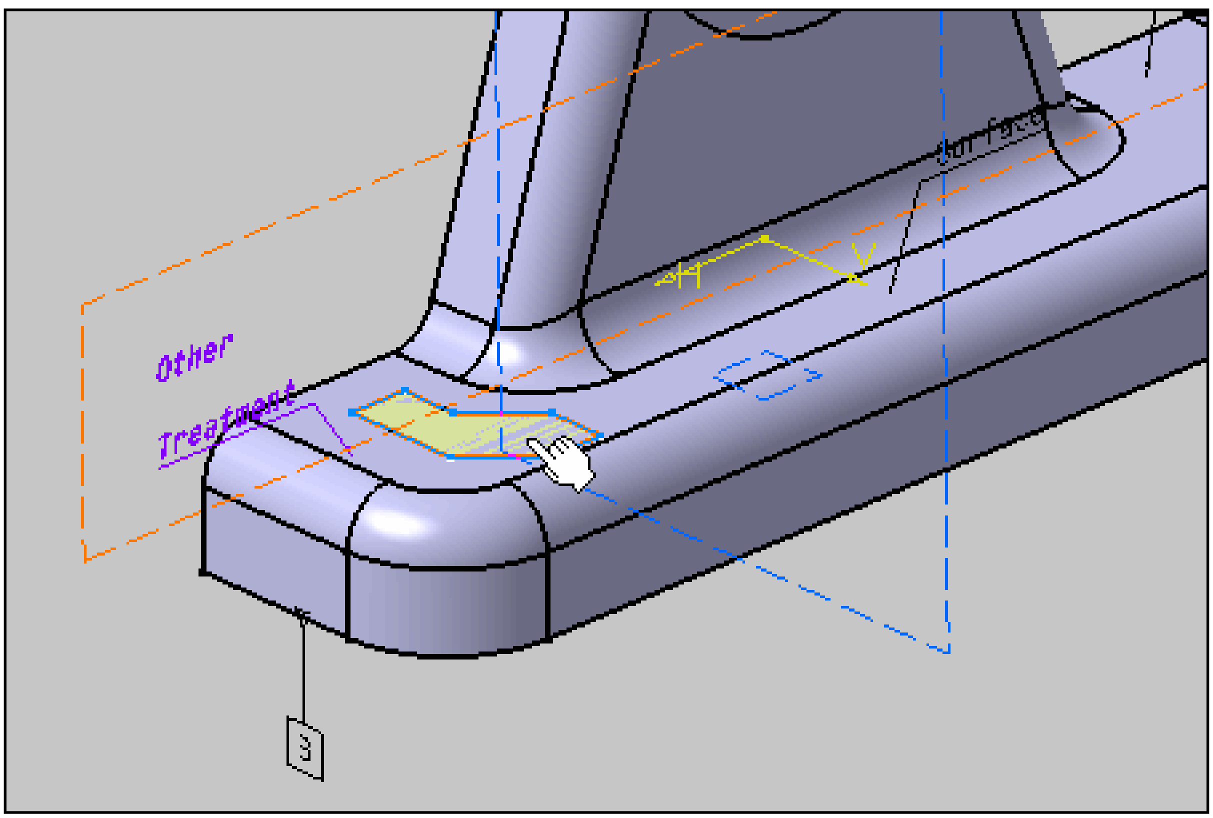
   The **Partial Surface** dialog box appears.

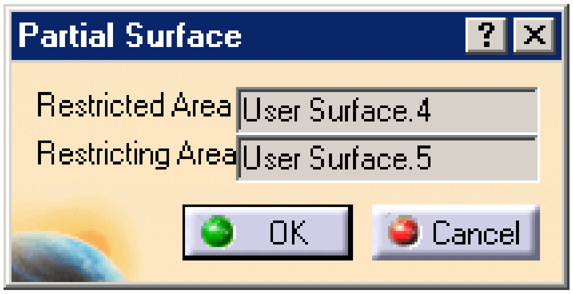


2. Select the restricted surface as shown on the part.



3. Select the restricting surface as shown on the part.





4. Click OK in the **Partial Surface** dialog box .

The **Restricted Area.1** is created.

Creating a Deviation

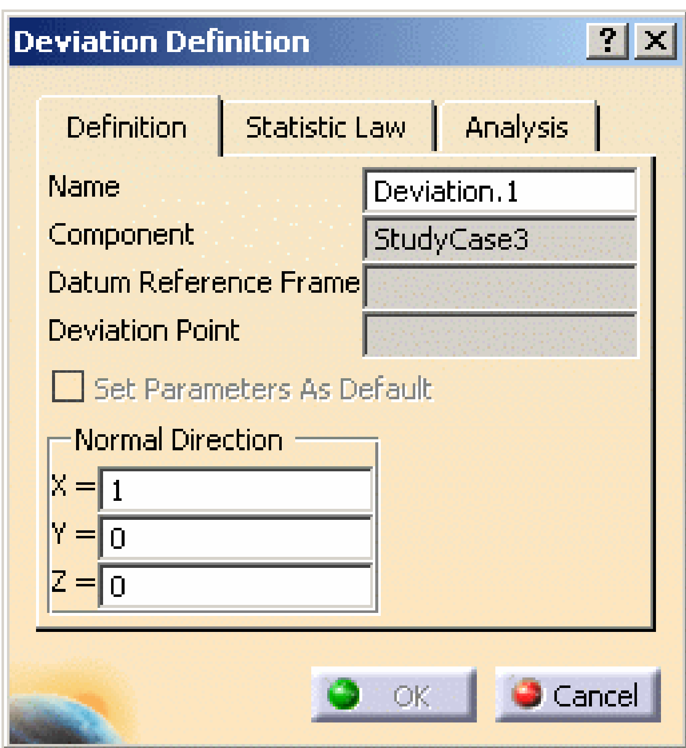
This task will show you how to create a deviation annotation on a part or product. This annotation is used in the Tolerance Analysis of Deformable Assembly workbench.

- A deviation annotation represents a specified or measured point according to a statistics law.
- The deviation annotation of an assembly component is contained in its annotation set:
  - For a leaf assembly component, deviation annotations represent the input annotations or initial annotations.
  - For a parent assembly component, deviation annotations represent the output annotations or annotations to be verified.
- A deviation annotation is always associated with a datum reference frame.
- This datum reference frame must be:
  - isostatic at least.
  - associated with the assembly component where the deviation is created.

Open the Tolerancing_Annotations_09.CATPart document.

1. Click the **Deviation** icon  in the Deviations toolbar.

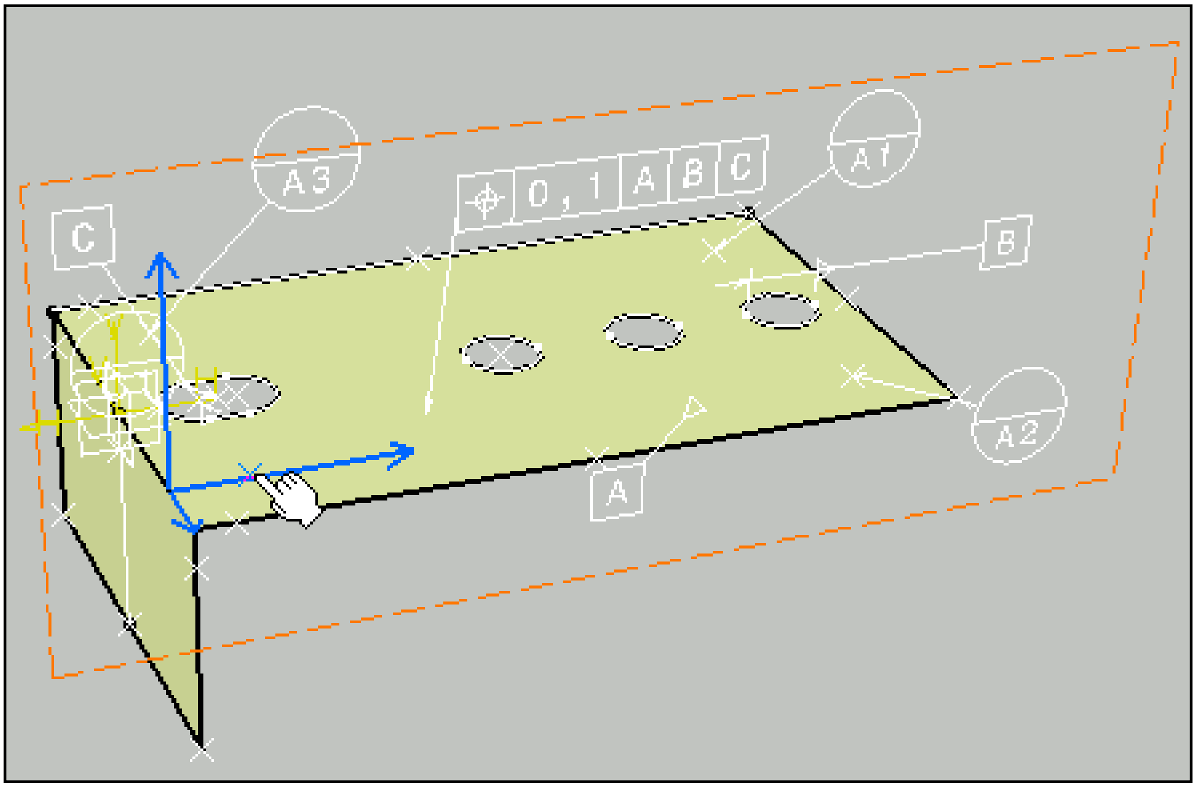
The **Deviation Definition** dialog box appears. You can notice that the **Component** field is automatically filled with the part name.



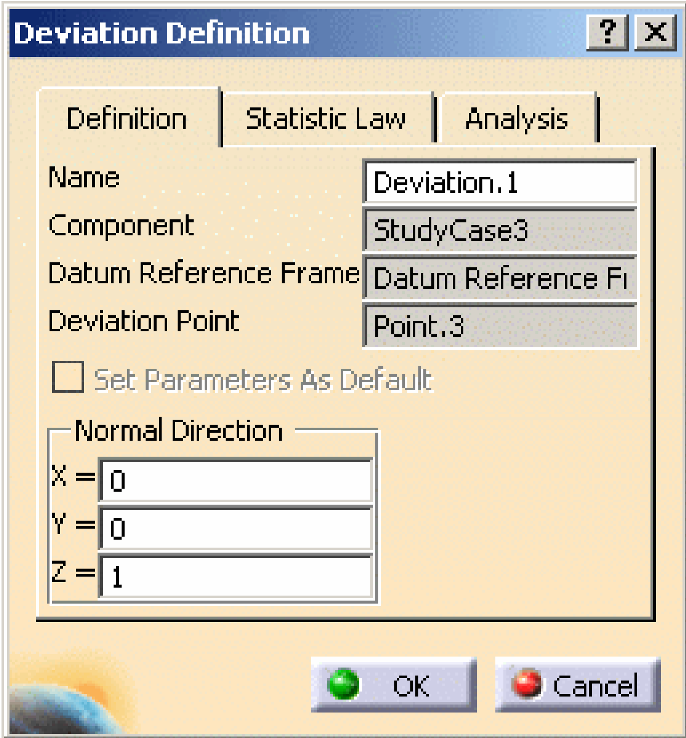
In a CATProduct document, you will need to select the assembly component.

2. Select the deviation's datum reference frame, in this case Datum Reference Frame.3.

3. Select a point or a vertex. For the purpose of this scenario, select Point.3.



4. Set Z as the normal direction.



5. Click the **Statistic Law** tab to select and define the desired law.

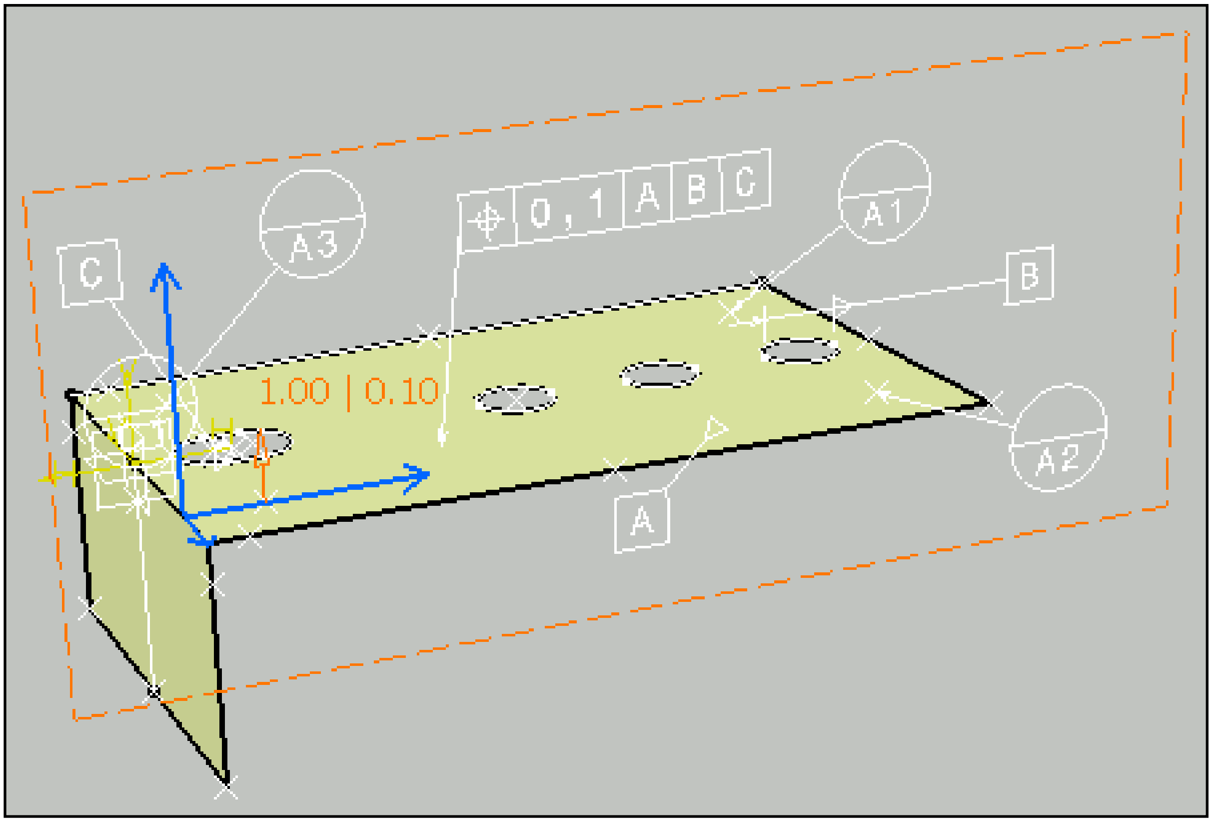
   Six laws and their parameters are available: Normal, Uniform, Constant, Pearson, Poisson and Snedecor. For more information, refer to Statistic Laws.

6. Click the **Analysis** tab to select and define the desired analysis.

   Two analyses are available:

   - Local: the deviation annotations are taken into account where they are defined only
   - Global: the deviation annotations are interpolated on the part or the product.

7. Click **OK**. The deviation annotation is created.



C
A3
0,1 A B C
A1
B
1.00 | 0.10
A2
A

Creating a Correlated Deviation

This task will show you how to create a correlated deviation annotation on an assembly component. This annotation is used in the Tolerance Analysis of Deformable Assembly workbench.

A correlated deviation annotation may be created on an assembly component.
A correlated deviation annotation represents specified or measured points according to a statistics law.
The correlated deviation annotation of an assembly component is contained in its annotation set.

For a leaf assembly component or a support, correlated deviation annotations represent the input annotations or initial annotations.

For a parent assembly component, correlated deviation annotations represent the output annotations or annotations to be verified.

A correlated deviation annotation is already associated with a datum reference frame.

This datum reference frame must:

Be isostatic at least.

Be associated with the assembly component where the deviation is created.

Clicking the **Generate Points** command in the **Deviation Definition** dialog box to generate default annotation points. These points are typical points where the component is the more flexible.

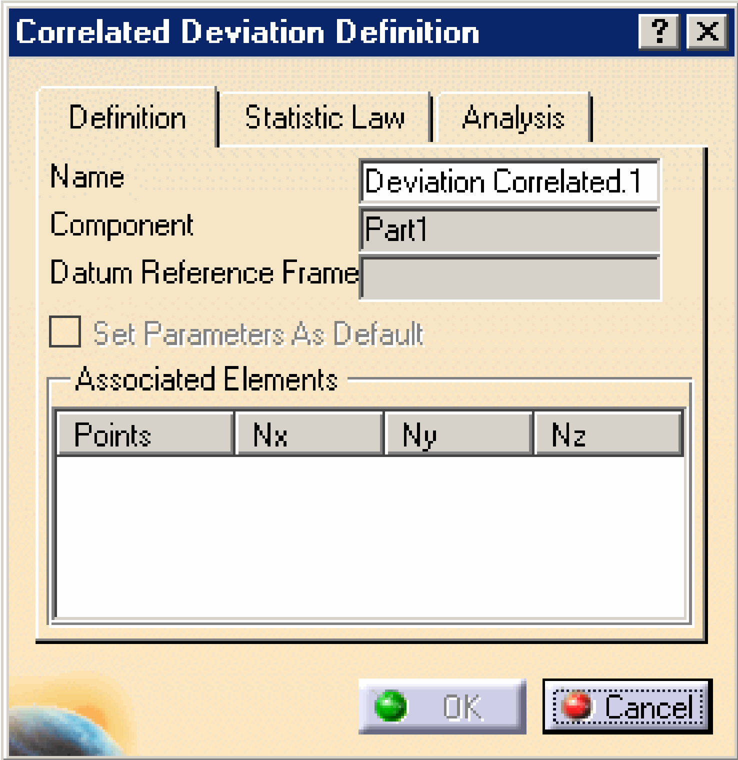
Open the document.

1. Click the **Correlated Deviation** icon:

2. Select the assembly component.

In a CATPart document the **Component** field is automatically updated with the part number.

The **Correlated Deviation Definition** dialog box appears.



In the **Statistic Law** tab you can select a measurement file to define the correlated deviation law.
When no measures file in the Statistics Law field is specified, each point of the correlated deviation is created according to a normal law with a mean of 1mm and a standard deviation of 0.1mm.

In the **Analysis** tab you can select and define the desired analysis.
Two analyses are available:

Local, the deviation annotations are take into account where they are defined only.

Global, the deviation annotations are interpolated on the part or the product.

3. Select the correlated deviation's datum reference frame.

4. Click **OK** to create the correlated deviation annotation.

Creating a Distance Between Two Points

This task will show you how to create a distance between two points annotation between two points of an assembly component or two assembly components. This annotation is used in the Tolerance Analysis of Deformable Assembly workbench.

Distance between two points annotation represents a distance to be checked between two points.
The distance between two points annotation of an assembly component is contained in the component's annotation set.
Distance between two points annotation represent the an output annotation or an annotation to be verified.

A distance between two points annotation is already associated with a positioning system or a datum reference frame.

This datum reference frame must:

Be isostatic at least.

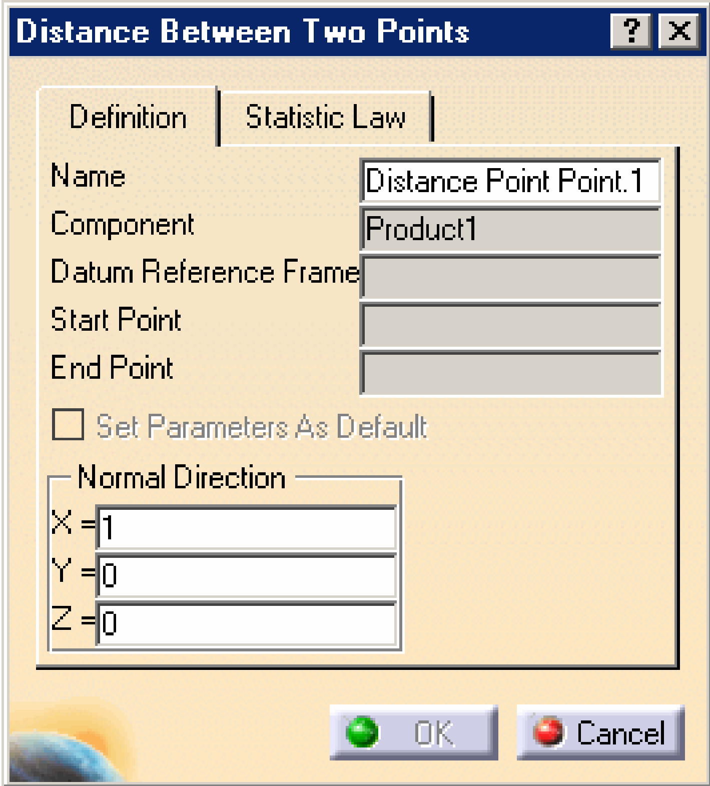
Be associated with the assembly component where the deviation is created.

Open the document.

1. Click the **Distance Between Two Points** icon:

2. Select the assembly component.

The **Distance Between Two Points** dialog box appears.



In the **Statistic Law** tab you can select and define the desired law.
Six laws and their parameters are available: Normal, Uniform, Constant, Pearson, Poisson and Snedecor. See Statistic Laws.

3. Select the distance between two points' datum reference frame.

4. Select the start point.

5. Select the end point.

6. Click **OK** to create the distance between two points annotation.

Managing Annotations

Select Annotation/Annotation Plane: select annotations according of an annotation plane or annotation planes of a set of annotations.

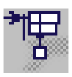
Display Annotation in the Normal View: click this icon and select an annotation.

Move a Textual Annotation: select the annotation and drag it to the desired location, or edit the entry fields from the Orientation and Position toolbar.

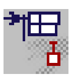
Transfer Existing Annotations: select the annotation, right-click to select the Transfer to View/Annotation Plane contextual command and select a new annotation plane.

Transfer Annotations During Creation: select the annotation, right-click to select the Transfer to View/Annotation Plane contextual menu item and select a new annotation plane.

Group Annotations: click the annotation icon you need for creating a new annotation, select the annotation to which you want to attach the new annotation and confirm the creation.



Grouping Annotations Automatically: select a set of annotation and click this icon.



Grouping and Ordering Annotations: click this icon, select the reference annotation, select the reference to be grouped.

Make the Position of a Text Associative: right-click the slave text and select the Annotation Links -> Create Positional Link contextual menu item, then select the master text.

Make the Orientation of a Text Associative: right-click the slave text and select the Annotation Links -> Create Orientation Link contextual menu item, then select the master text.

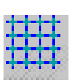
Mirror Annotations: click this icon, the reversed annotations are mirrored.

Clip Annotations Plane: click this icon, the part is clipped by the annotation plane.

Mark Non-semantic Annotations: select or not the option.

Set Annotation Parallel to Screen: select an annotation and activate this option.

Replace a Datum Reference Frame: select a reference frame annotation and select a new datum.



Use 3D Grid: click this icon and select an annotation.

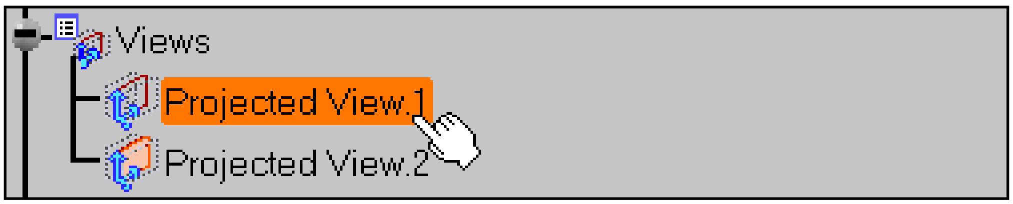
Selecting Annotation/Annotation Plane

This task shows you how to select the annotations of an annotation plane and the annotation plane of an annotation.

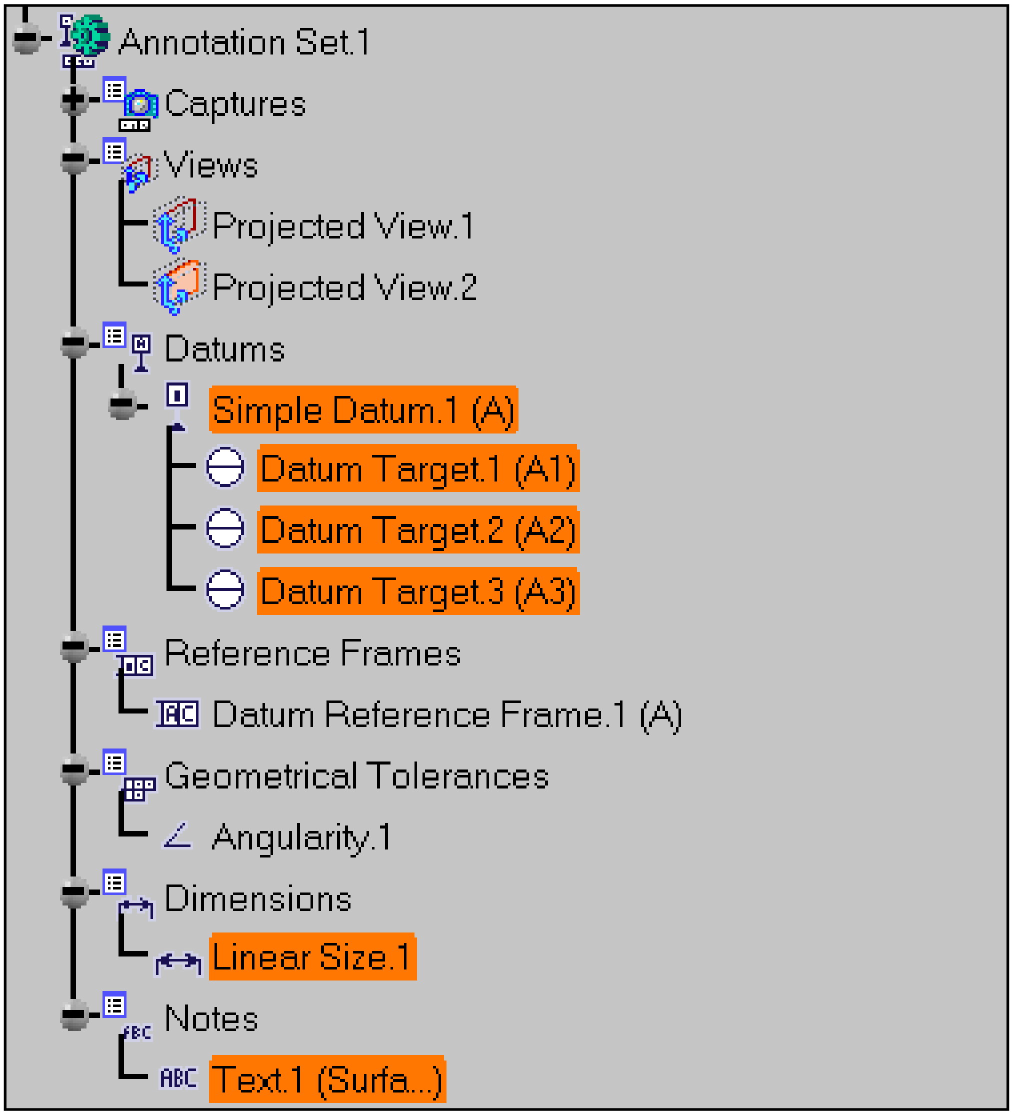
Open the Tolerancing_Annotations_05 CATPart document:

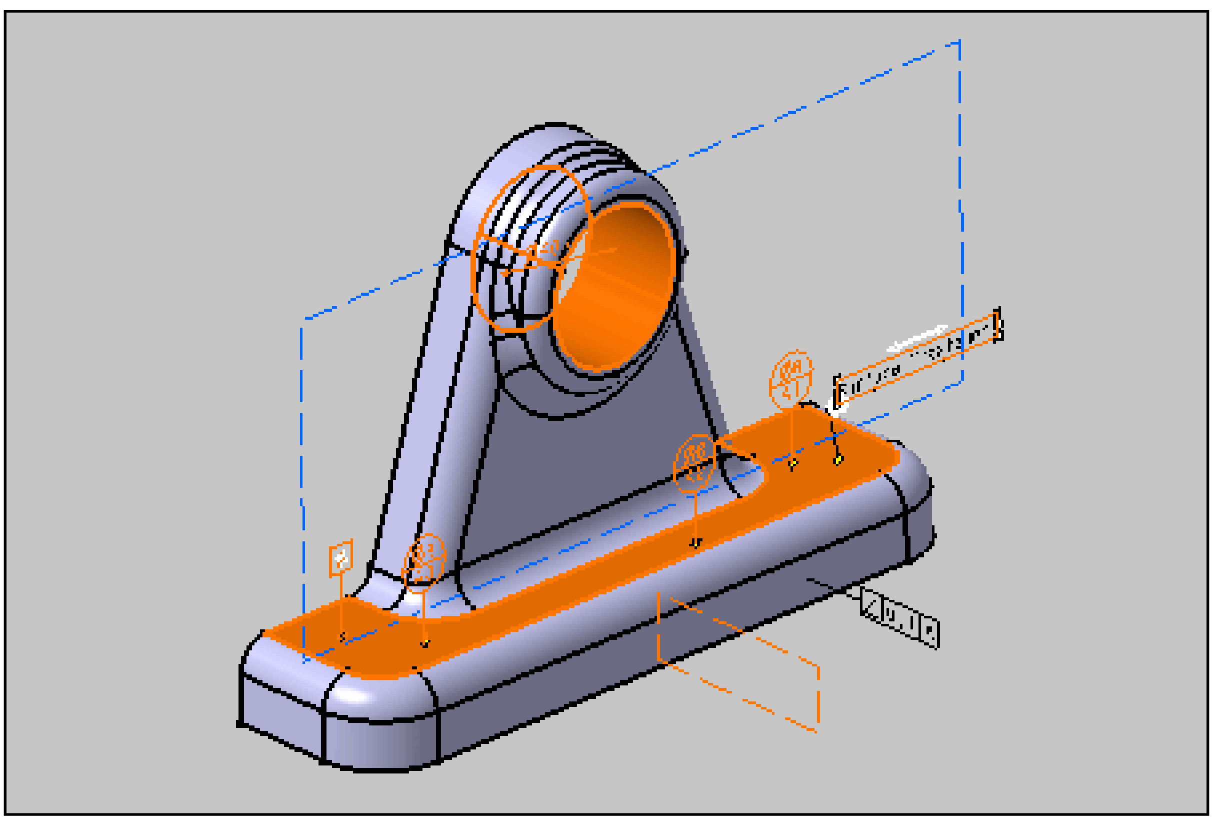
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Right-click the **Projection View.1** annotation plane and select **Select Annotations** from the contextual menu.

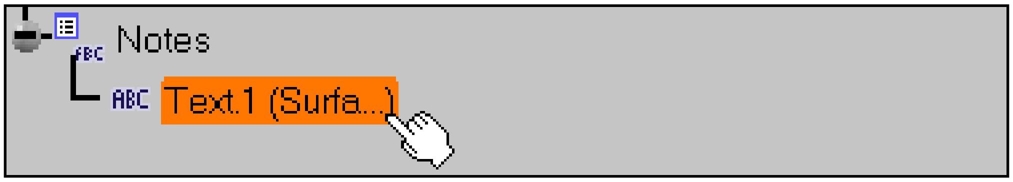


All the annotations of the **Projection View.1** annotation plane are selected in the geometry and the specification tree.
The selected annotation plane is not activated and you can select several annotation planes to perform the command.

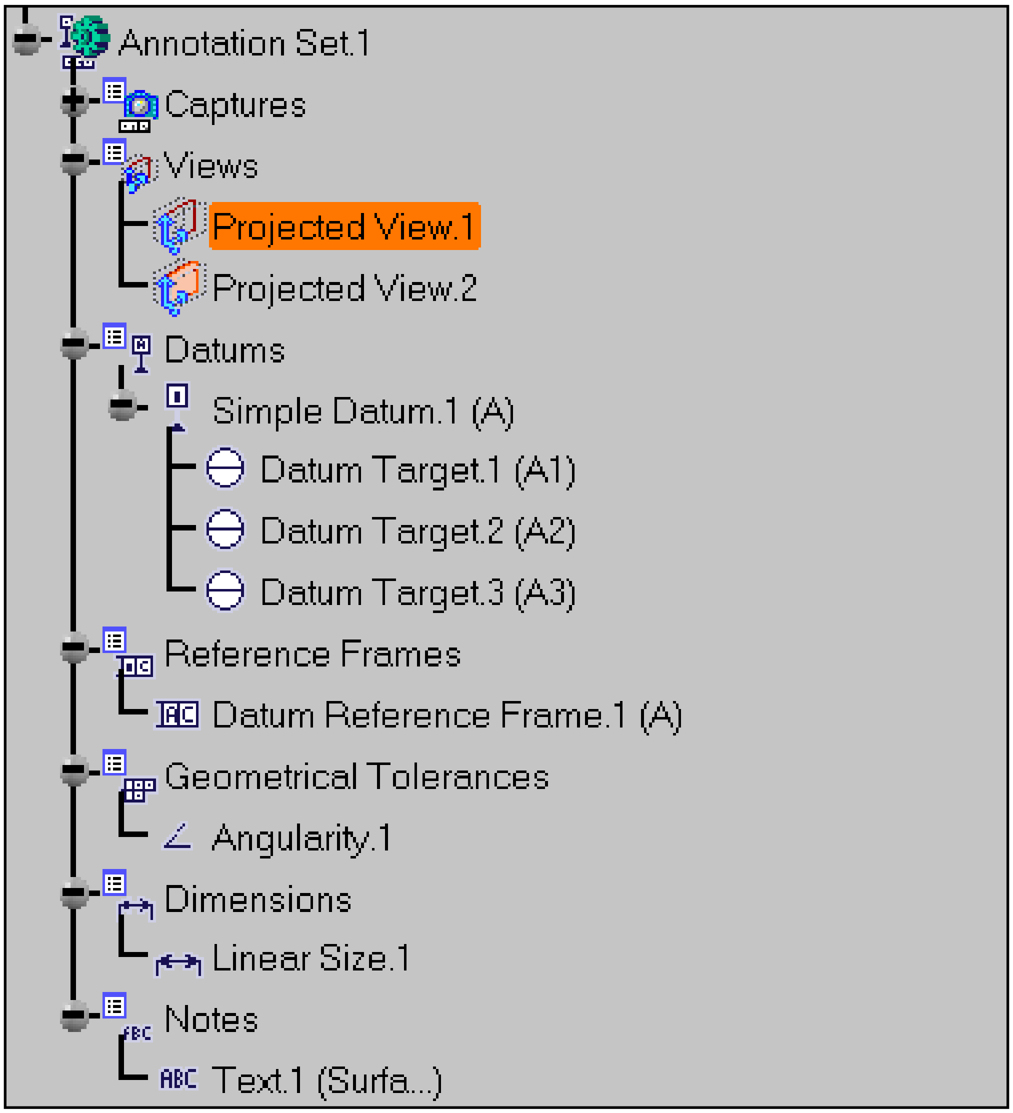

Annotation Set.1
Captures
Views
Projected View.1
Projected View.2
Datums
Simple Datum.1 (A)
Datum Target.1 (A1)
Datum Target.2 (A2)
Datum Target.3 (A3)
Reference Frames
Datum Reference Frame.1 (A)
Geometrical Tolerances
Angularity.1
Dimensions
Linear Size.1
Notes
ABC
Text.1 (Surfa...)

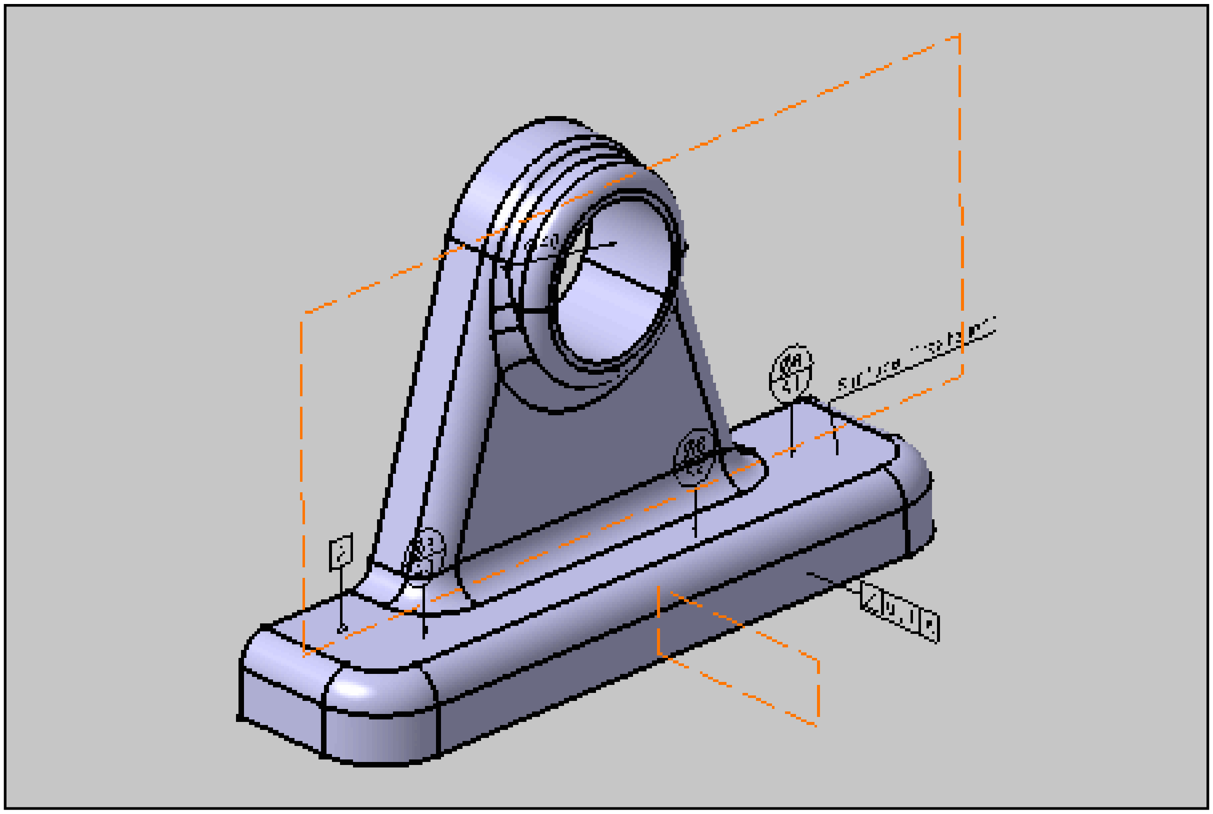


2. Right-click the **Text.1** annotation and select **Select Views/Annotation Plane** from the contextual menu.



The annotation plane of the **Text.1** annotation is selected in the geometry and the specification tree.
The selected annotation plane is not activated and you can select several annotations from different annotation planes to perform the command.



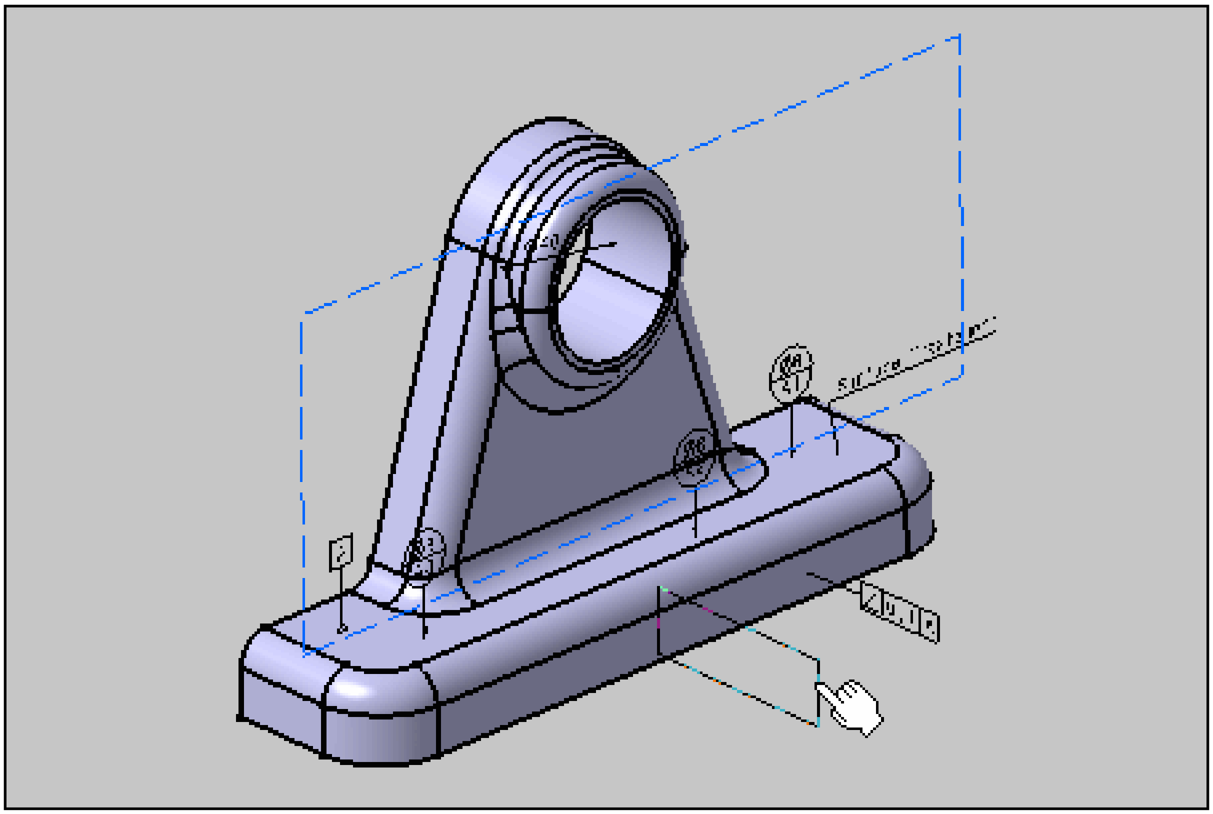


Displaying Annotation in the Normal View

This task shows you how to display an annotation plane in the normal view.
The operating mode described here applies to annotation plane, annotation and any planar element too.

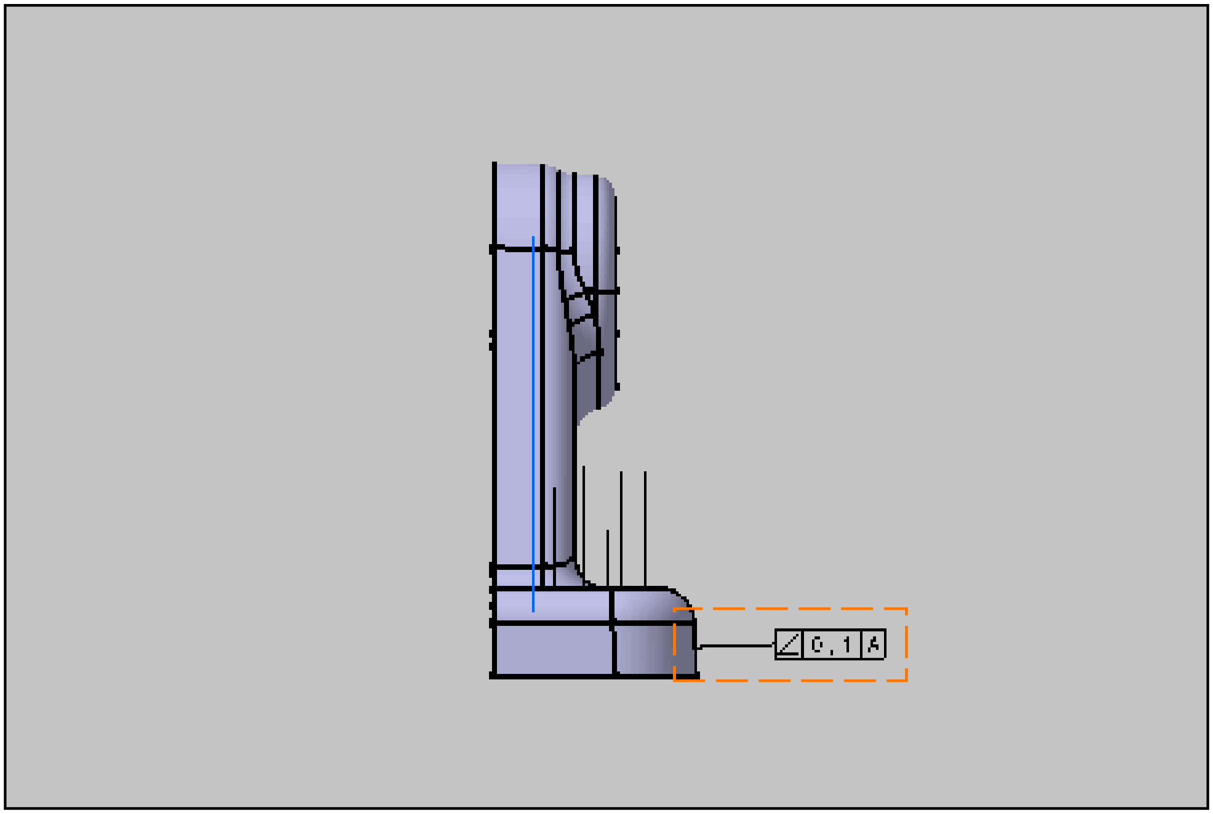
Open the Tolerancing_Annotations_05 CATPart document.

1. Select the **Projection View.2** annotation plane.



2. Select the **Normal View** icon: 

The annotation plane is displayed in the normal view.


0.1
A

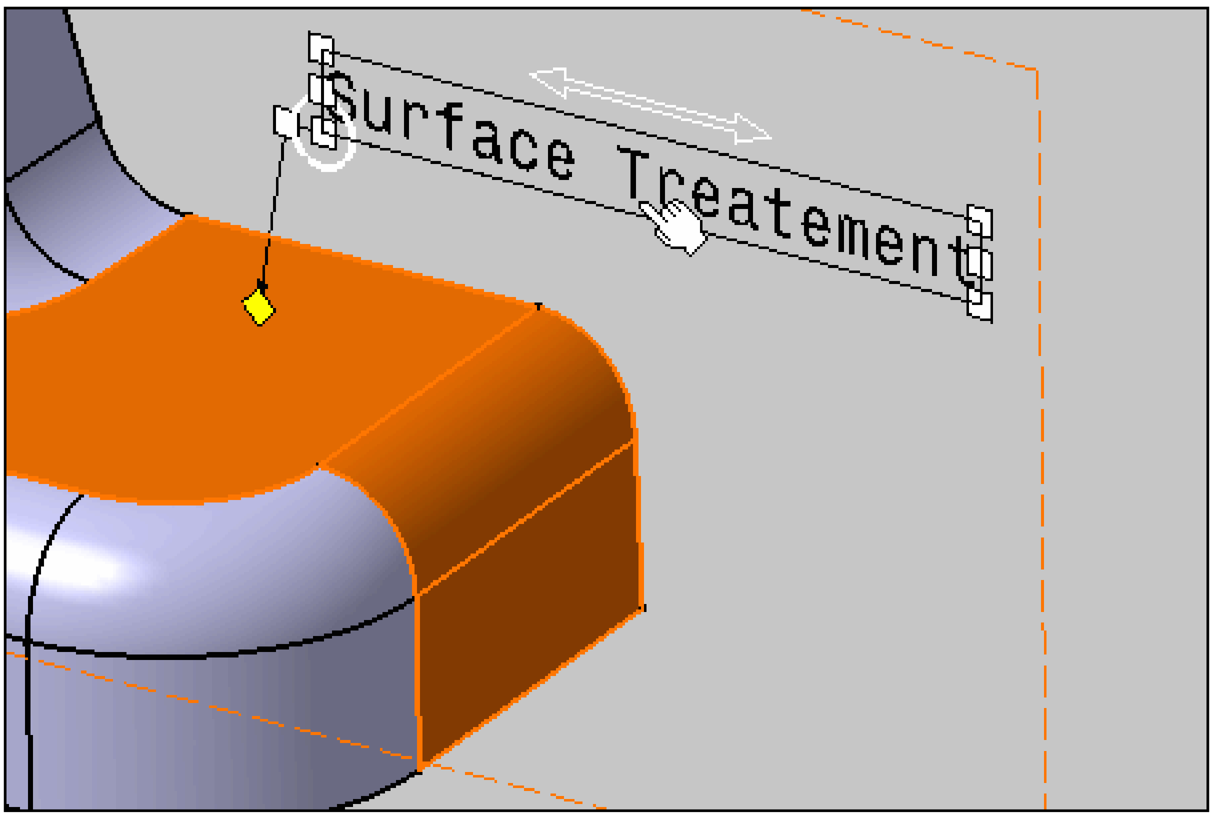
Moving Annotations

This task shows you two ways of moving a text annotation: by drag and drop, then by using coordinates. The operating mode described here applies to datum elements, datum targets and geometrical tolerances.

Open the Annotations_Part_04.CATPart document:

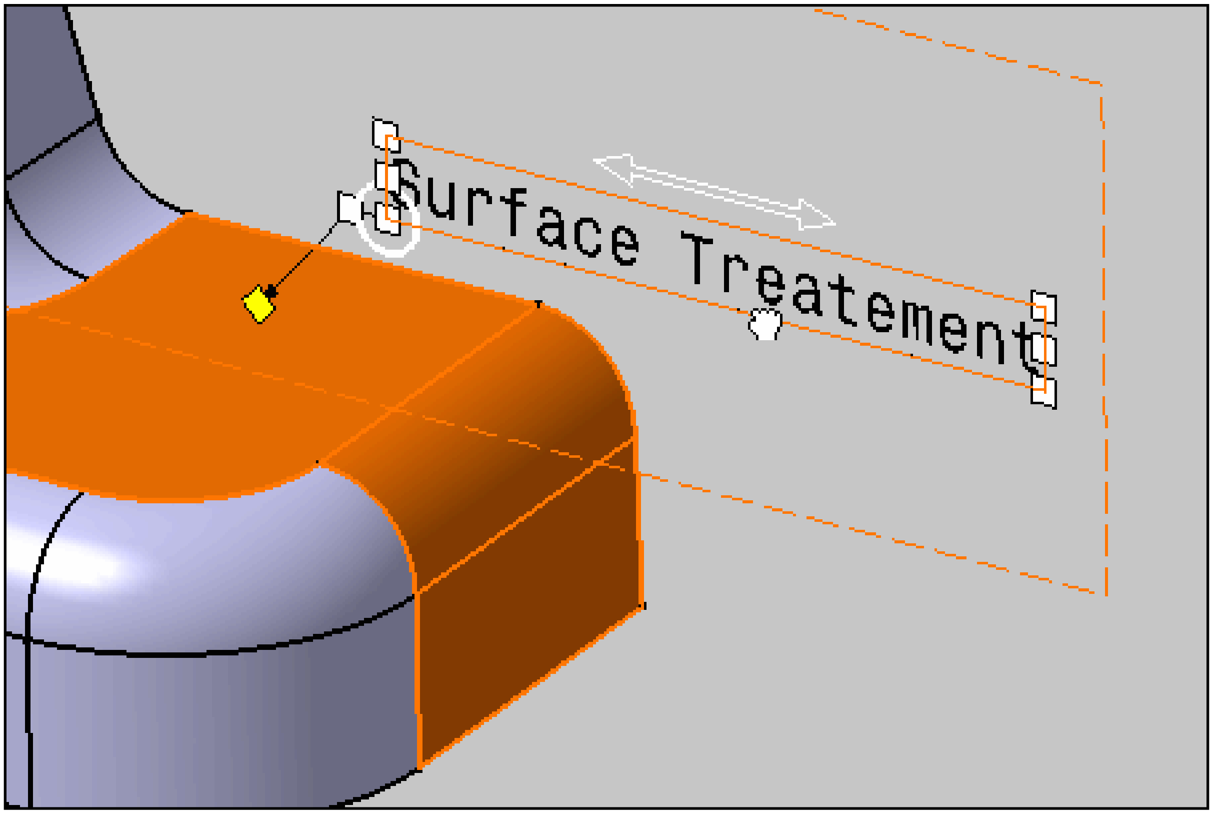
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Select the annotation text.



2. Drag it to the desired location.

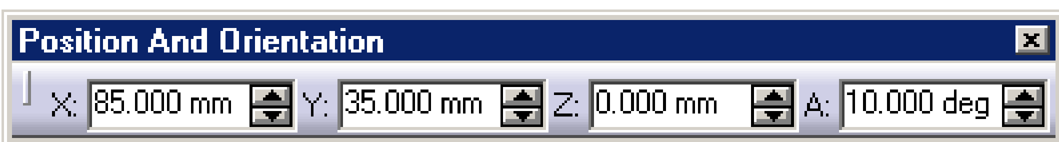
   Note that you can stretch or reduce the frame too.



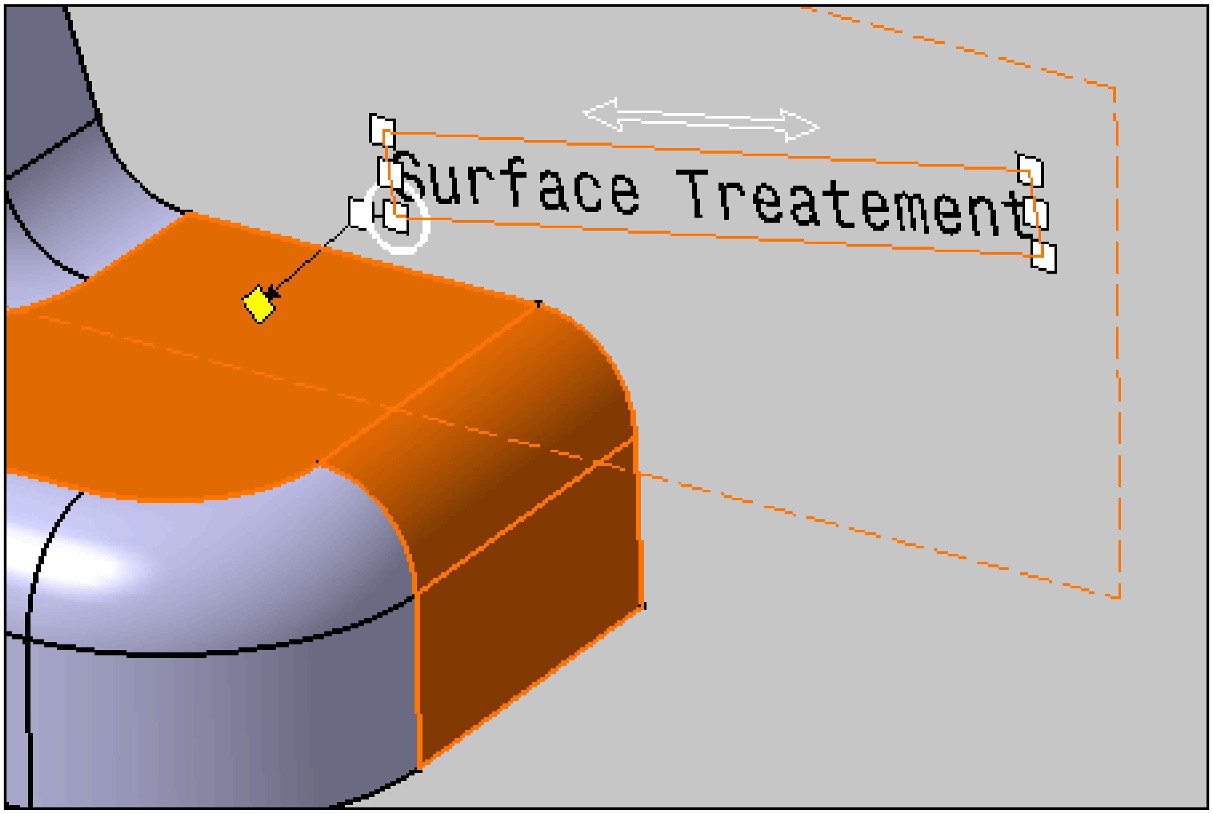
3. Move it using coordinates: enter the value of your choice in the **X, Y** field from the **Orientation and Position** toolbar, then enter another value to define the rotation **A** field.

   You can set the increment of your choice to define the rotation angle.

   For more information, refer to Customizing for 3D Functional Tolerancing & Annotations.



This is what you can obtain.


Surface Treatement

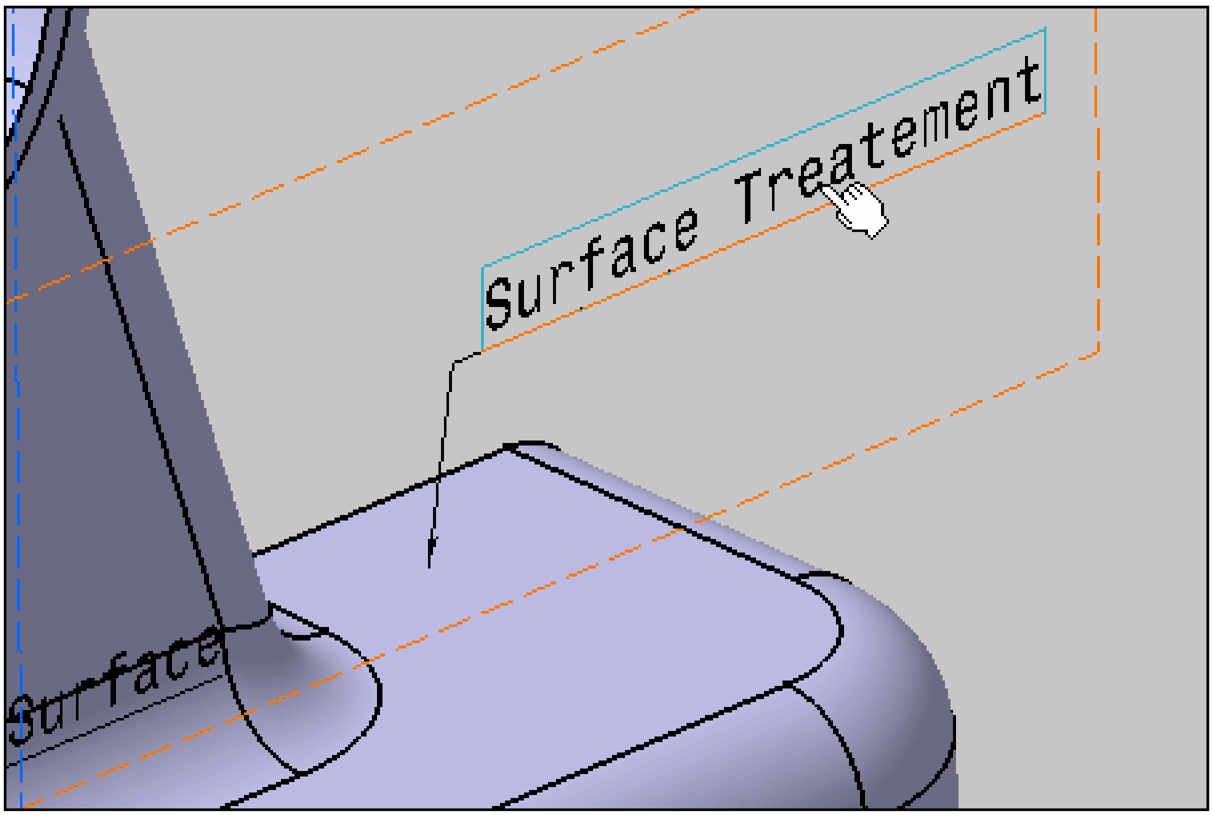
Transferring Existing Annotations

This task first shows you how to transfer an existing textual annotation from one view to another. The operating mode described here applies to datum elements, datum targets and geometrical tolerances too.

Open the Annotations_Part_04.CATPart document:

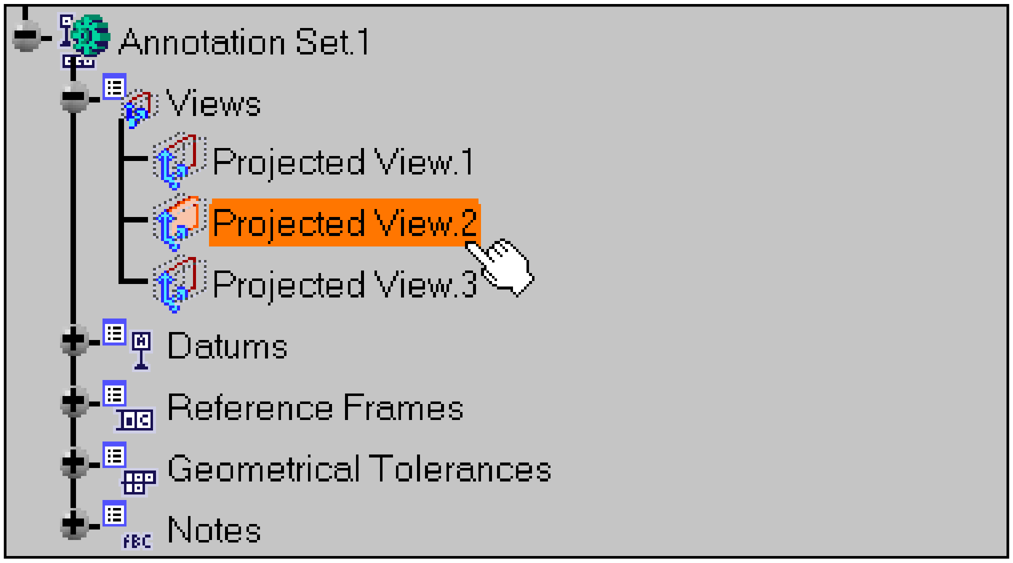
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Select the text (or its leader line).

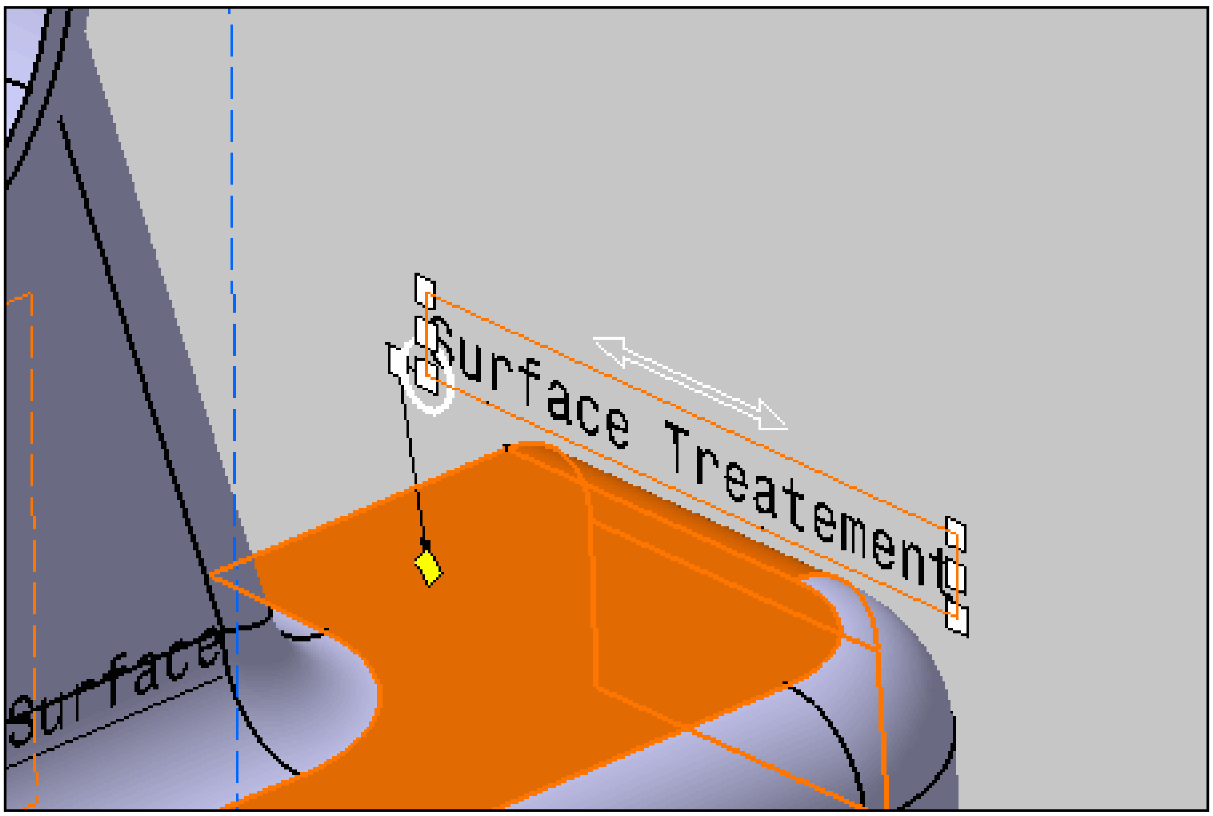


2. Right-click to select the **Transfer to View/Annotation Plane** contextual menu.

3. Select **Projected View.2** in the tree or in the geometry. You can perform the operation on non-active views only.



The textual annotation is transferred to the new view.

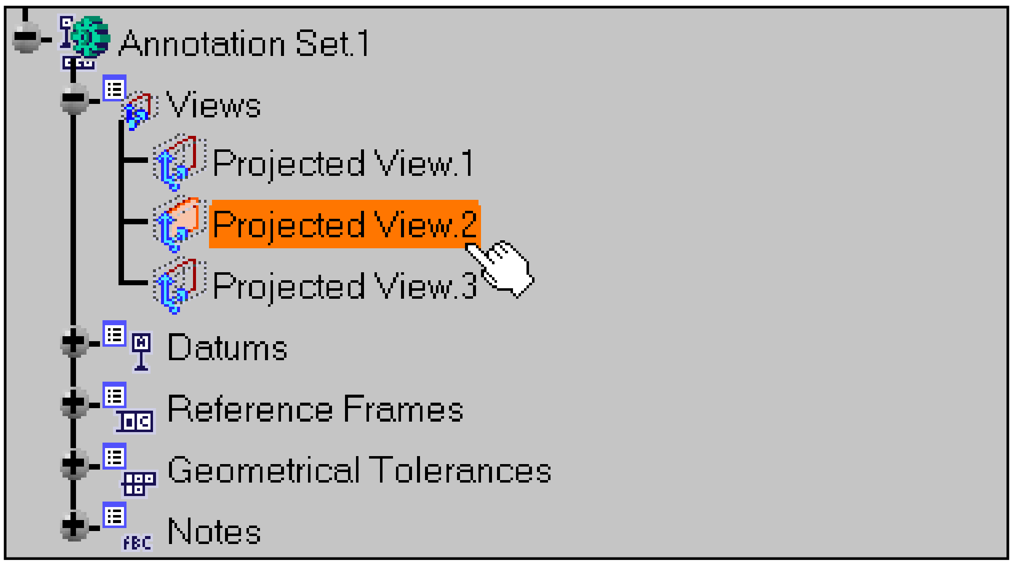


Transferring Annotations During Creation

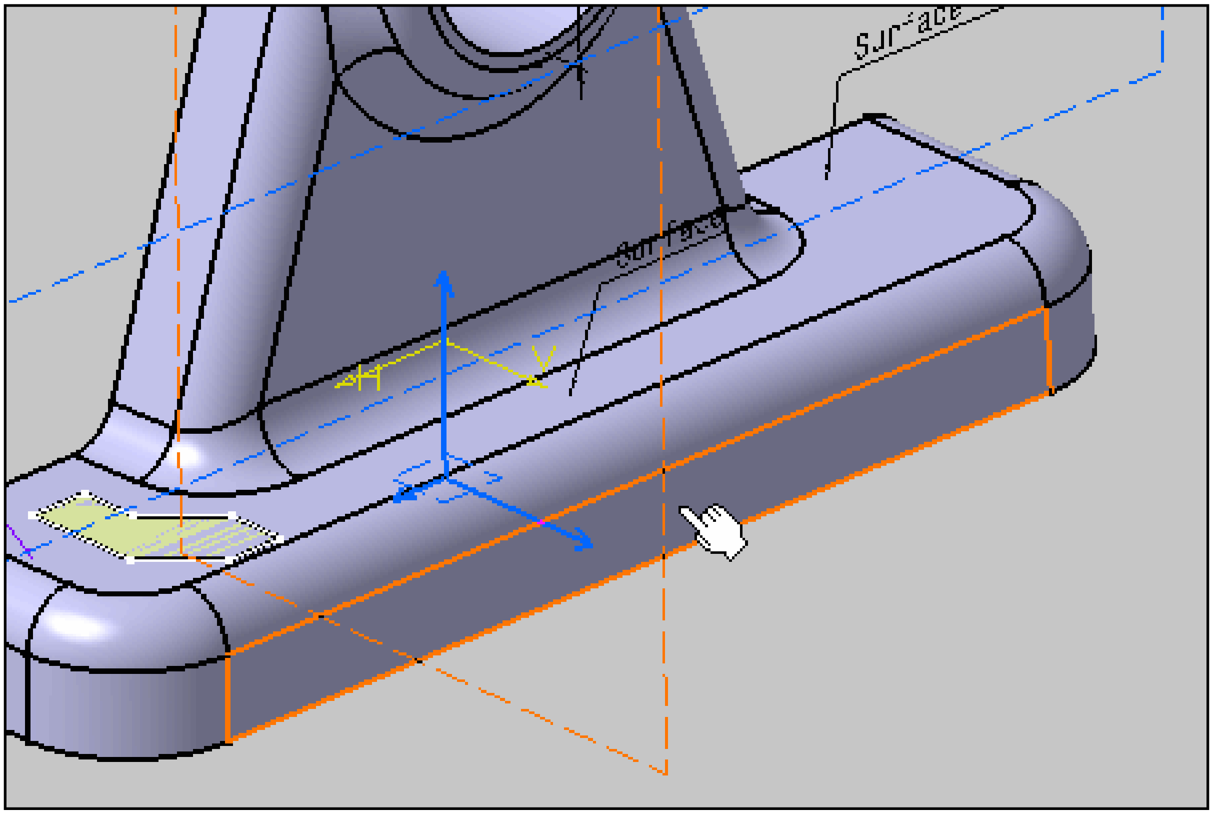
This task first shows you how to transfer the annotation you are creating to another view.
The operating mode described here applies to datum elements, datum targets and geometrical tolerances too.

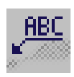
Open the Annotations_Part_04.CATPart document.

1. Select the **Projected View.2** annotation plane.



2. Select the face as shown to define a location for the arrow end of the leader.

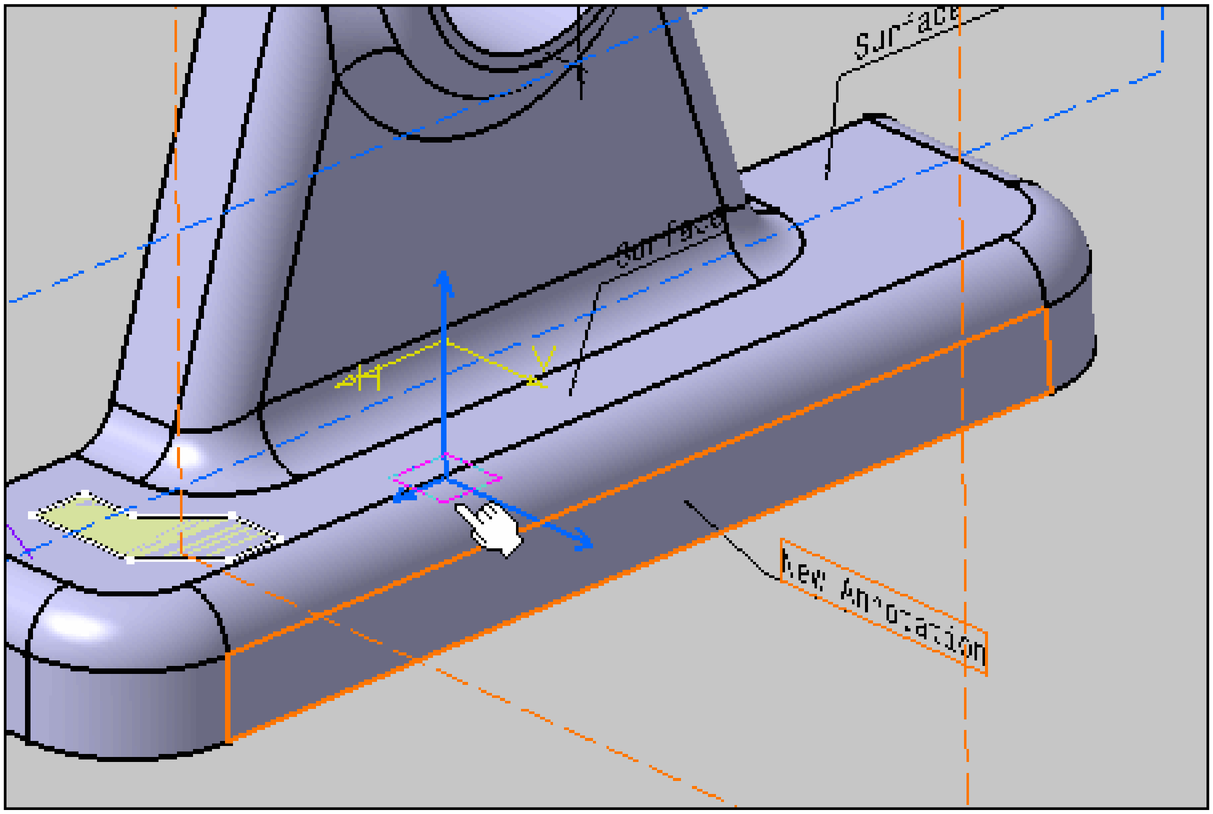


3. Click the **Text with Leader** icon: 

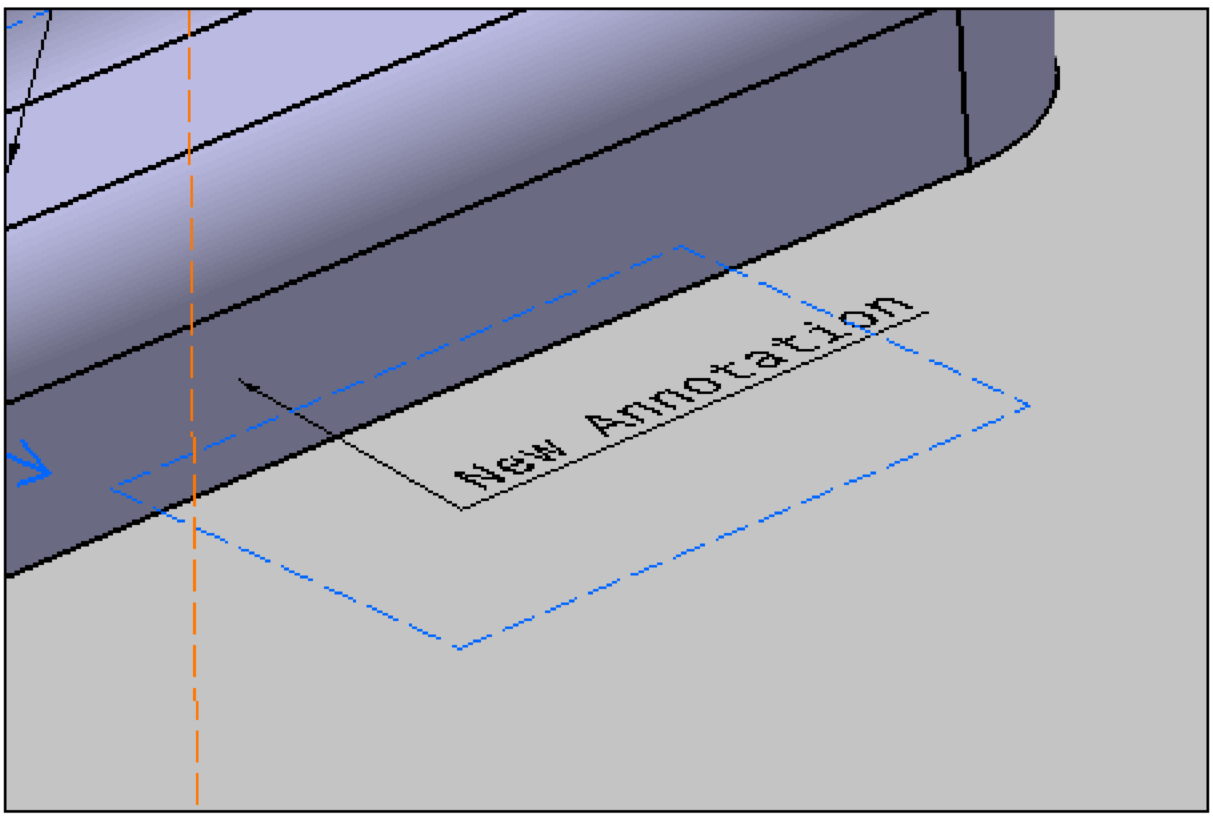
4. Enter your text, for example "New Annotation" in the **Text Editor** dialog box and click **OK**.

5. Select the **Projected View.3** annotation plane to which you want to transfer "New Annotation".

   This annotation plane (or view) must be an existing one.



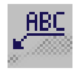
The annotation is transferred.



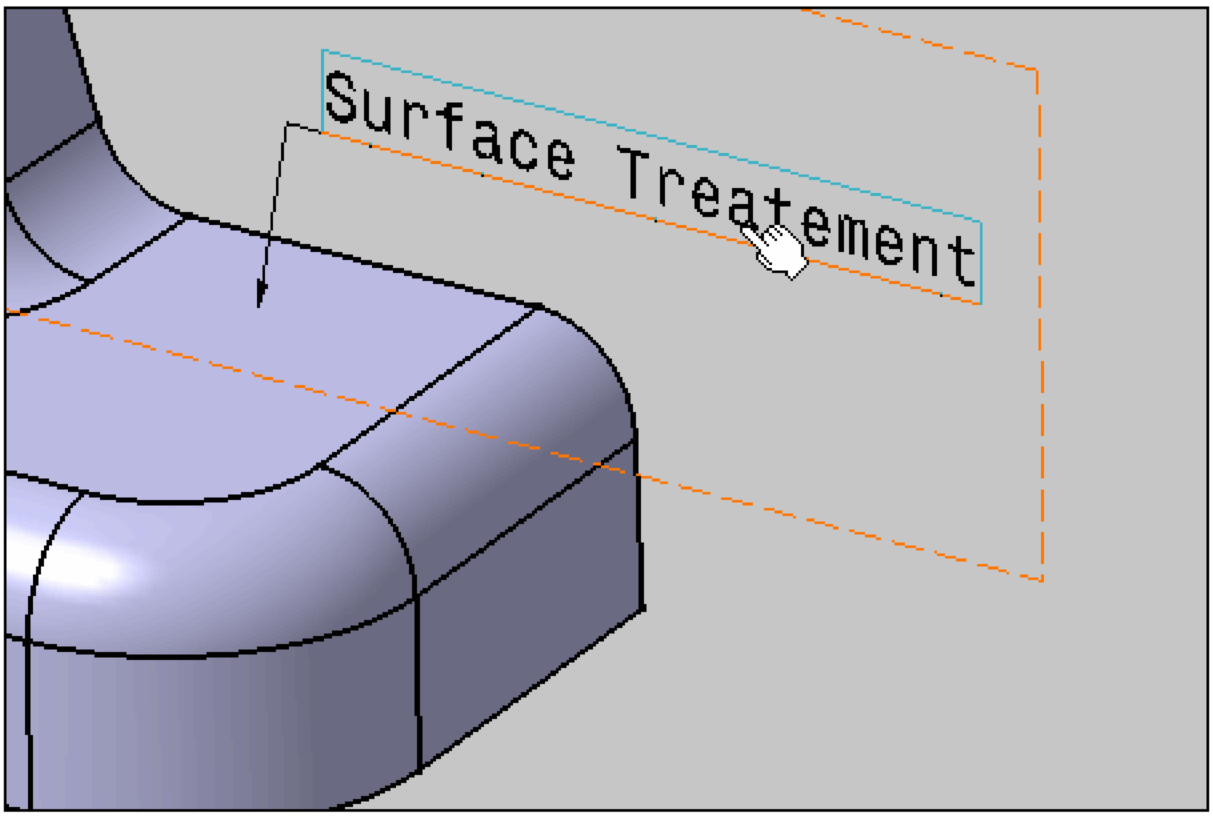
Grouping Annotations During Creation

This task shows you how to group an annotation you are creating to an existing annotation. Groups of annotations can gather as many annotations as you wish.

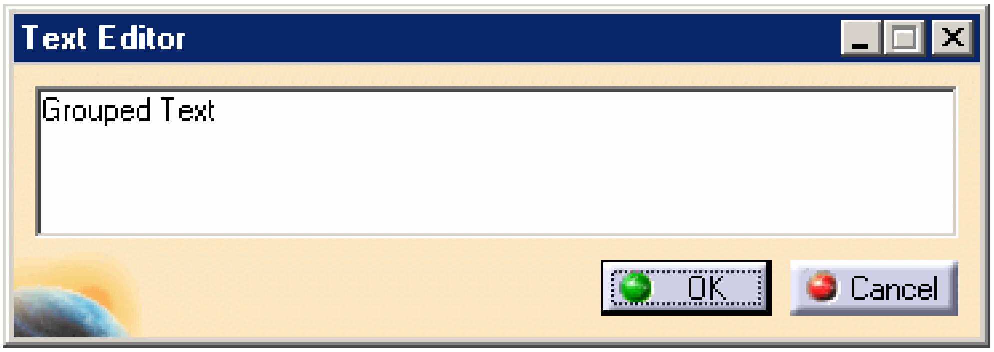
Open the Annotations_Part_04.CATPart document.

1. Click the **Text with Leader** icon: 

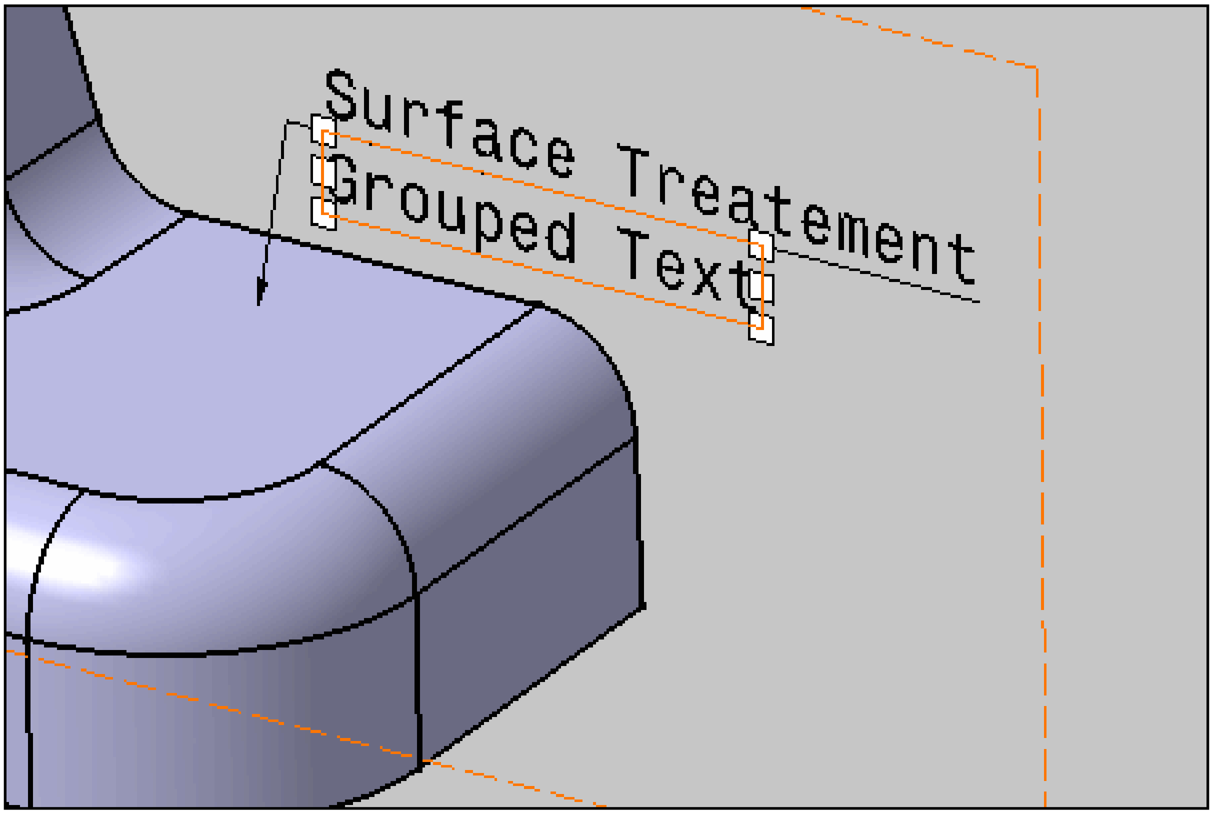
2. Select "Surface Treatment" as the annotation to which you want to attach the annotation you are going to specify.



3. Enter the text "Grouped Text" in the **Text Editor** dialog box and click **OK.**



The two annotation text are grouped.



If you need to edit the group properties, multi-select the annotations and use the Properties contextual command.

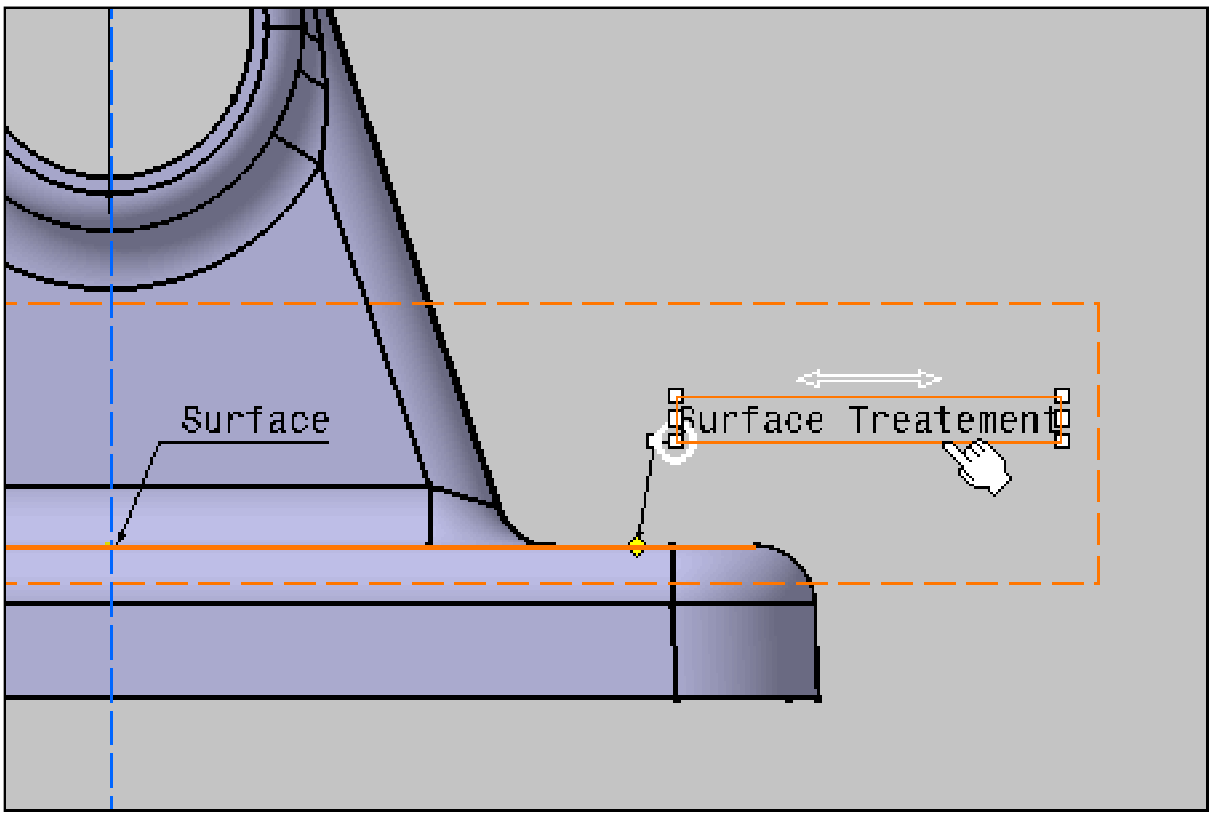
Grouping Annotations Automatically

This task shows you how to group automatically annotations.

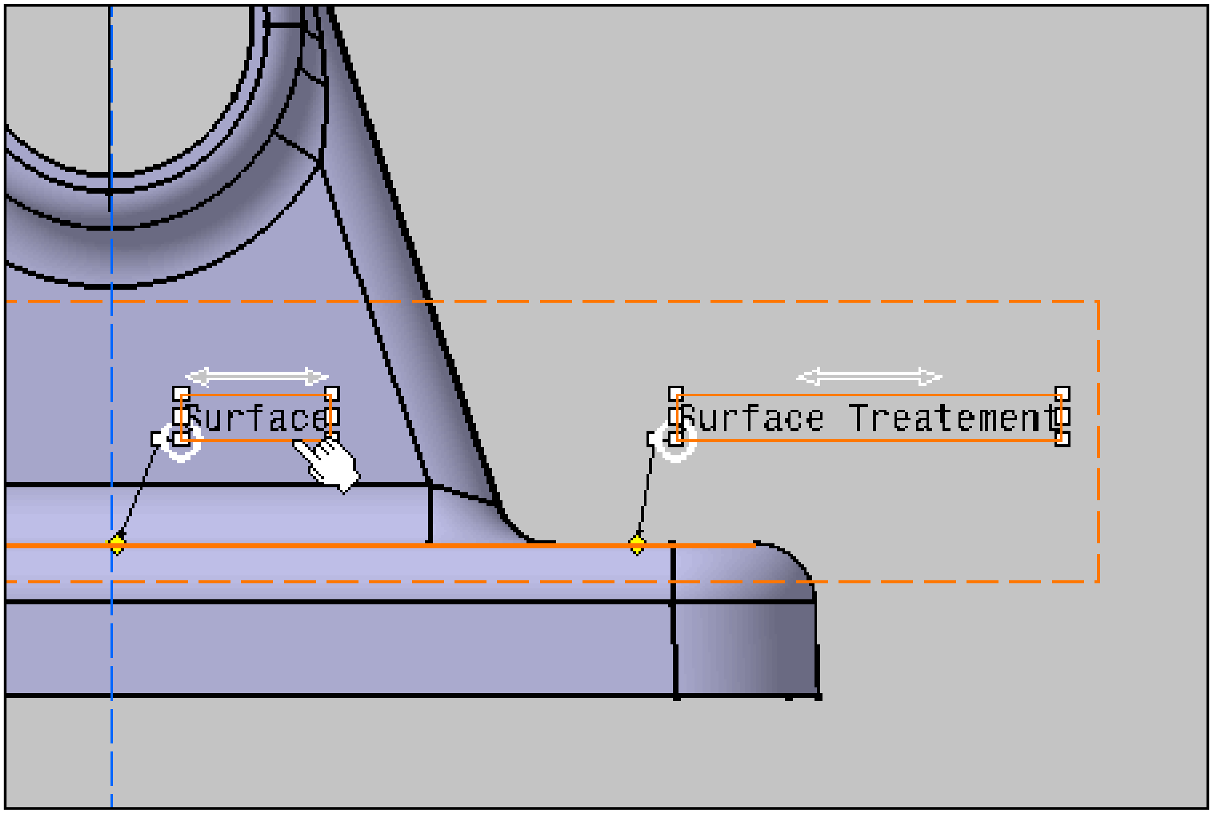
You can select as many annotations as you want.
Selected annotations must associated with the same geometrical elements.
According to the standard annotations are grouped with an specific order.

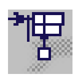
Open the Annotations_Part_04.CATPart document.

1. Select **Surface Treatment** as the first annotation to be grouped.

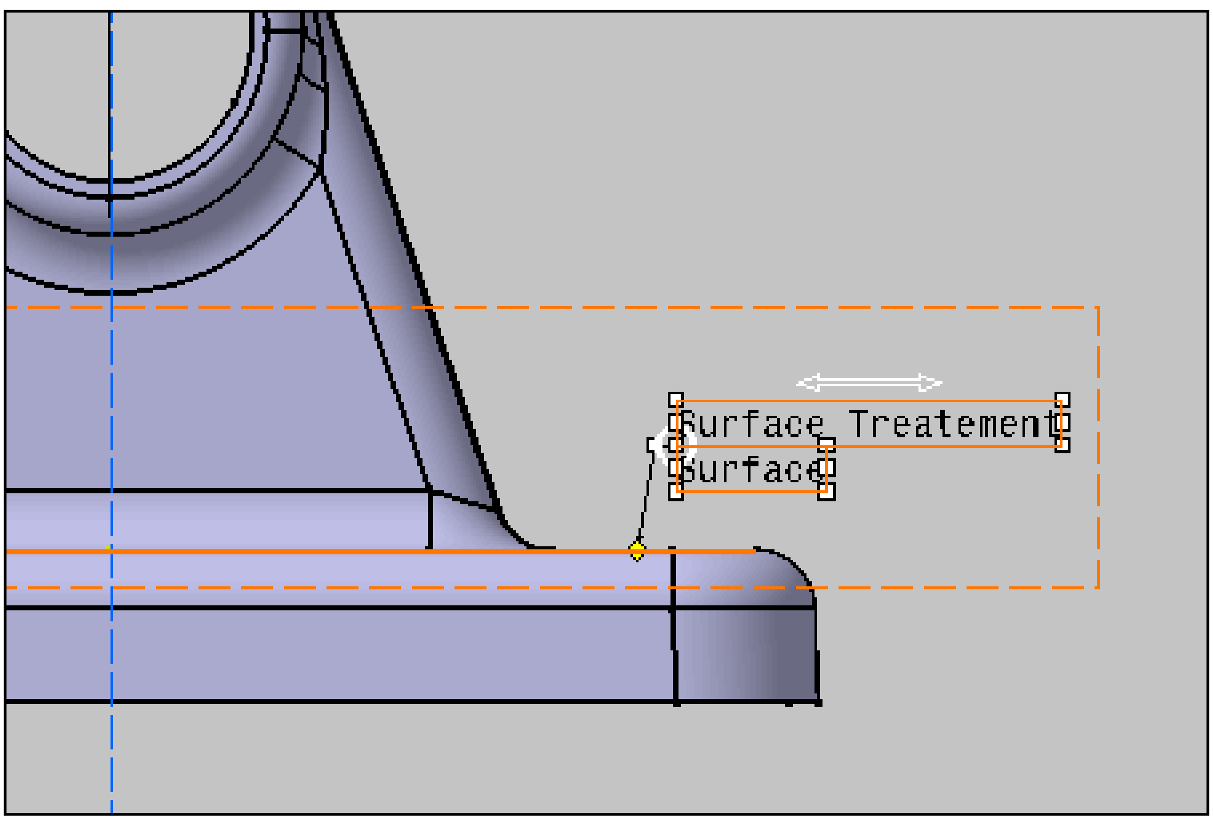


2. Multi-select **Surface** as the second annotation to be grouped.



3. Click the **Automatic Grouping** icon: 

Annotations are grouped.



Grouping and Ordering Annotations

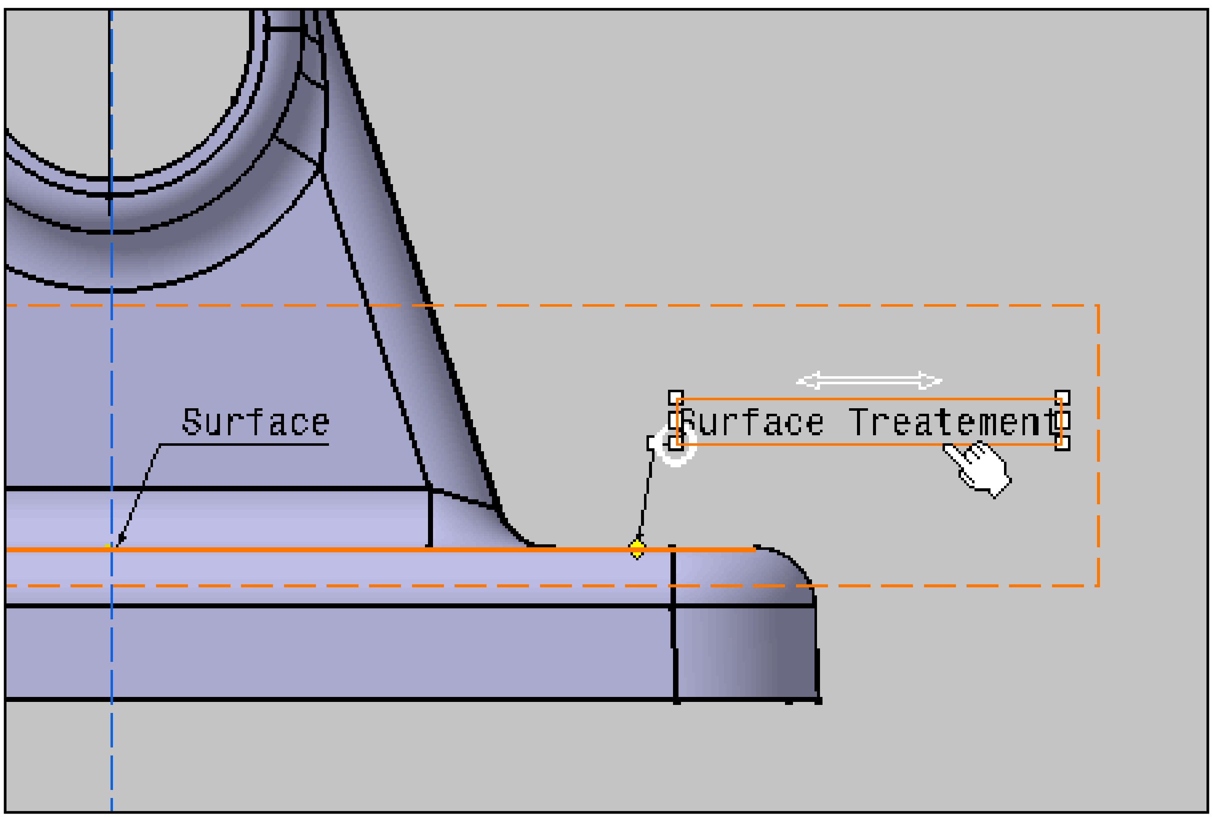
This task shows you how to group manually and order annotations.

You must select one by one annotations to be grouped.
Reference and selected annotations must be associated with the same geometrical elements.
According to the standard annotations are grouped with an specific order.

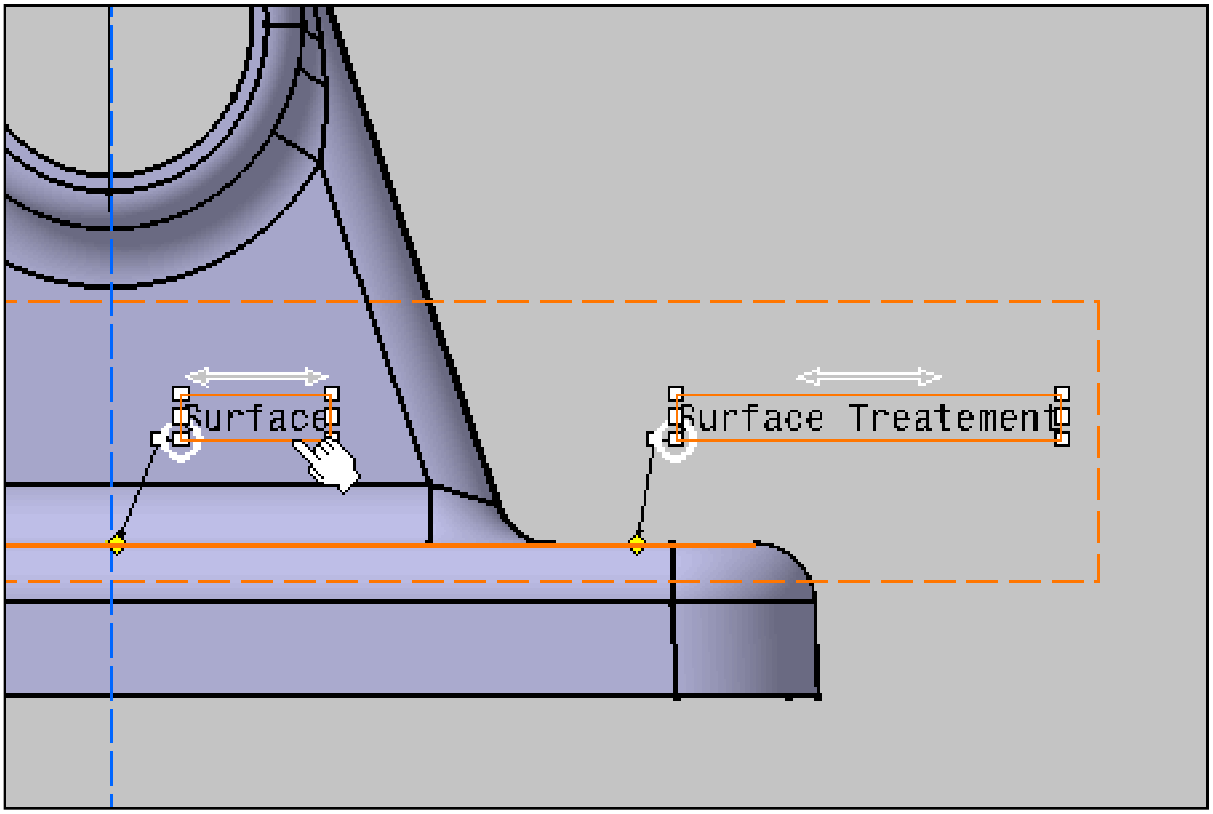
Open the Annotations_Part_04.CATPart document.

1. Click the **Manual Grouping** icon:

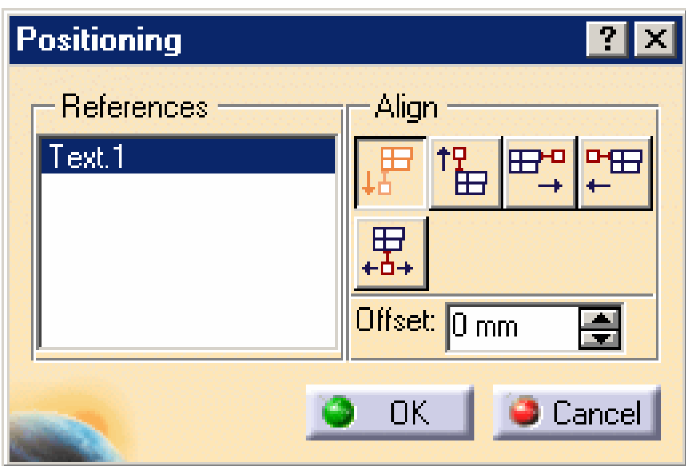
2. Select **Surface Treatment** (**Text.1**) as the reference annotation.



3. Select **Surface** (**Text.2**) as the slave annotation to be grouped.



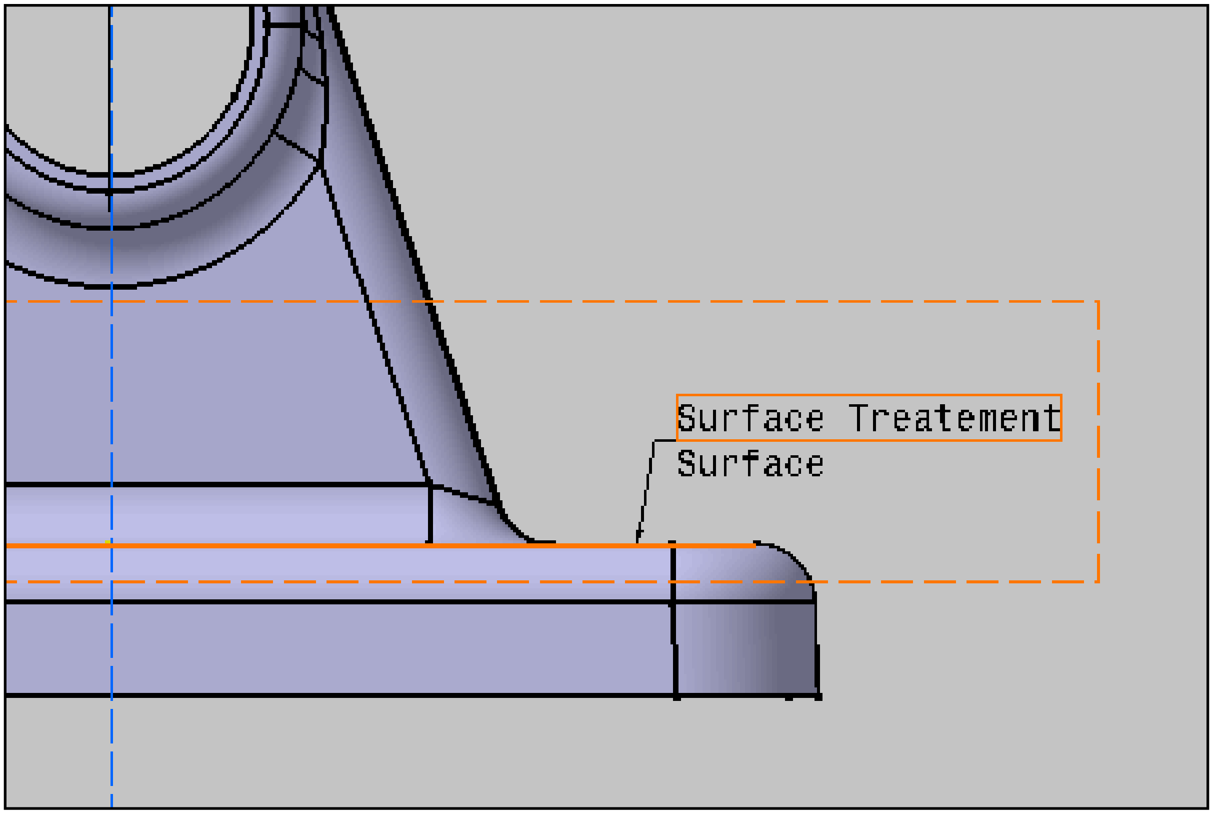
The **Positioning** dialog box appears. **Text.1** is the active reference annotation.



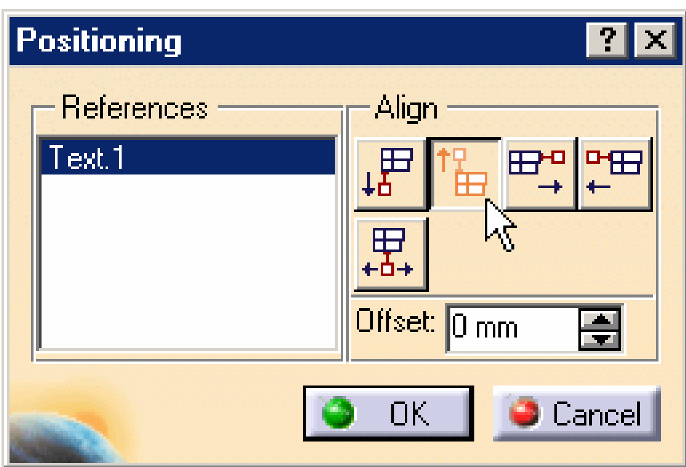
There are six settings:

- **Align Bottom** the selected annotation according to the reference.
- **Align Top** the selected annotation according to the reference.
- **Align Right** the selected annotation according to the reference.
- **Align Left** the selected annotation according to the reference.
- **Center Horizontally** the selected annotation according to the reference.
- Set the **Offset** between the selected annotation and the reference.

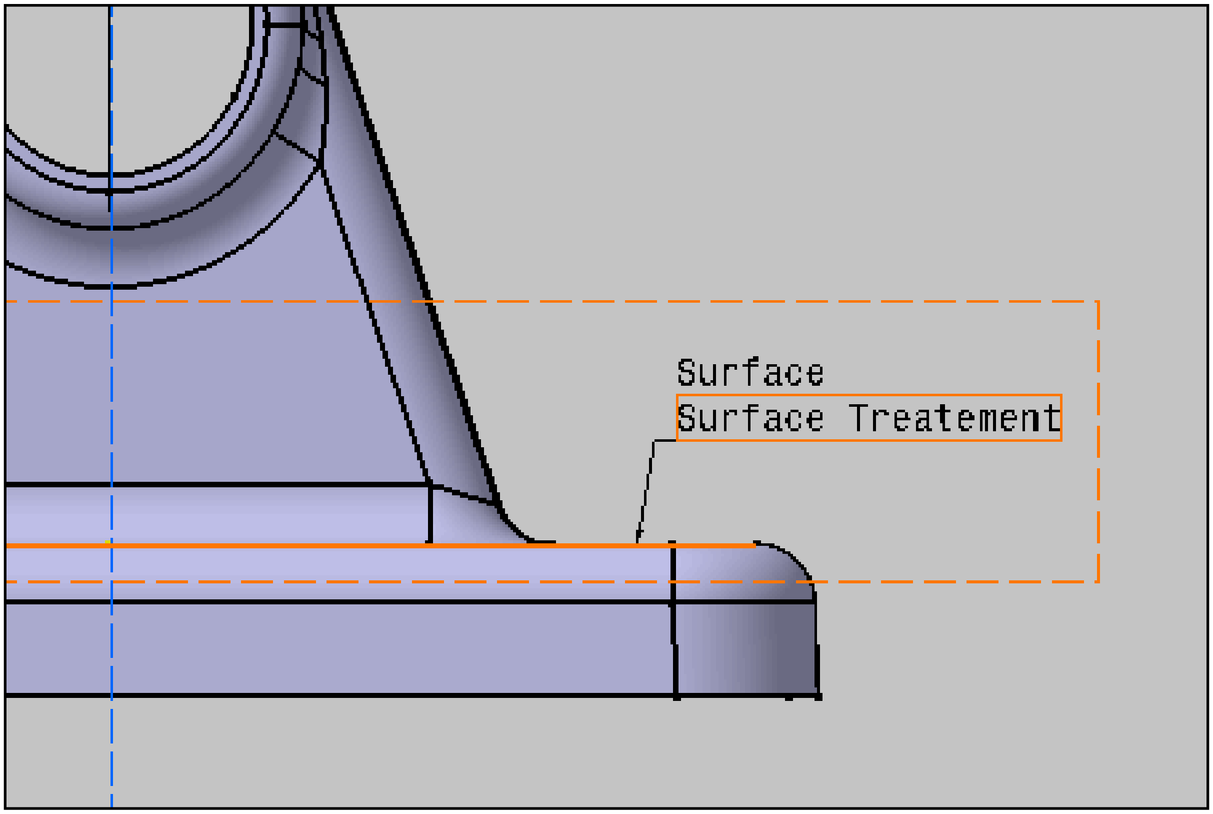
Annotations are grouped according to the dialog box settings: **Align Bottom** and 0mm **O**. **Text.1** reference annotation is orange framed.



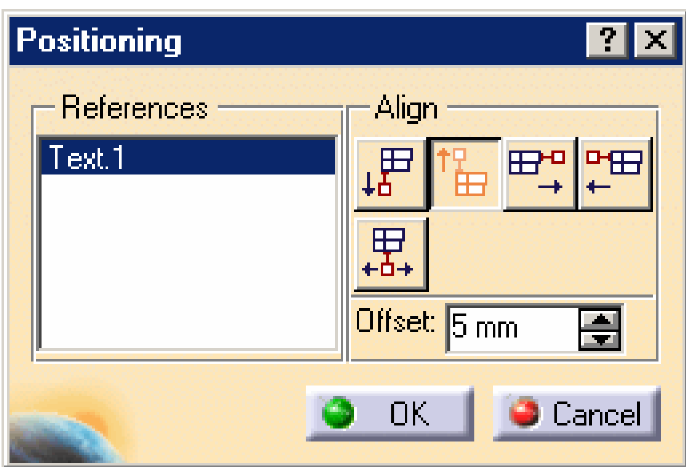
4. Click the **Align Top** icon in the **Positioning** dialog box.



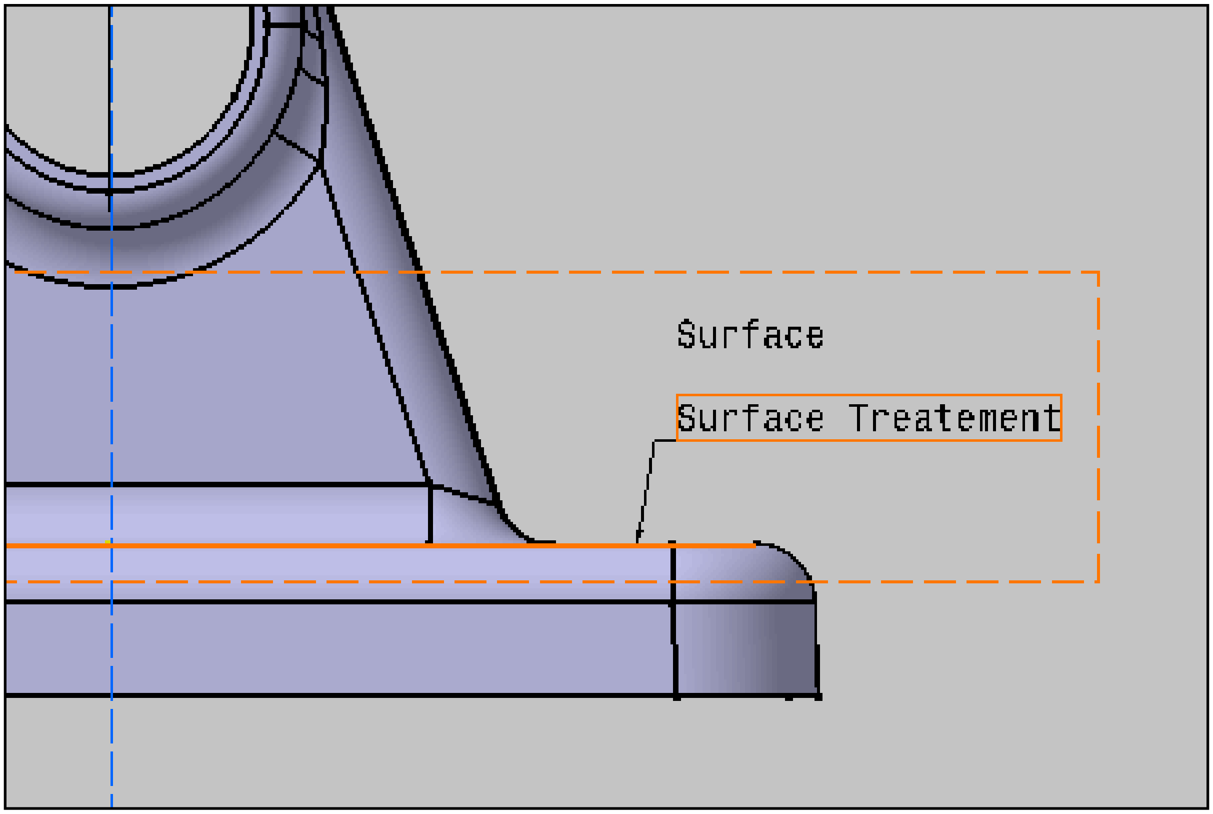
Surface (**Text.2**) is aligned to the top of **Surface Treatment** (**Text.1**).



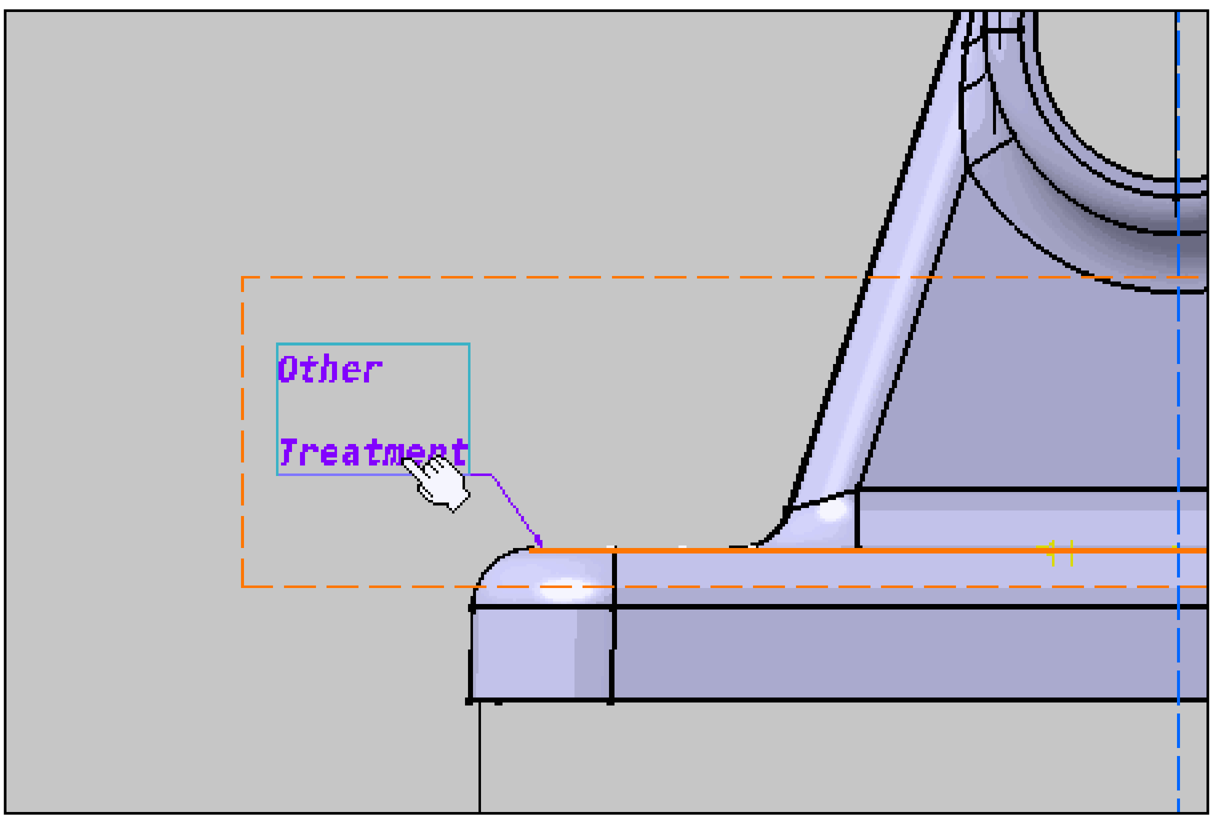
5. Set the **Offset** to 5mm in the **Positioning** dialog box.



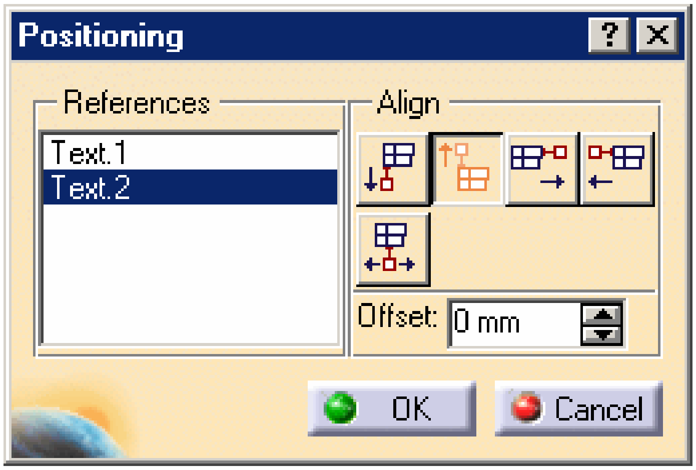
Surface (**Text.2**) is aligned to the top of **Surface Treatment** (**Text.1**).



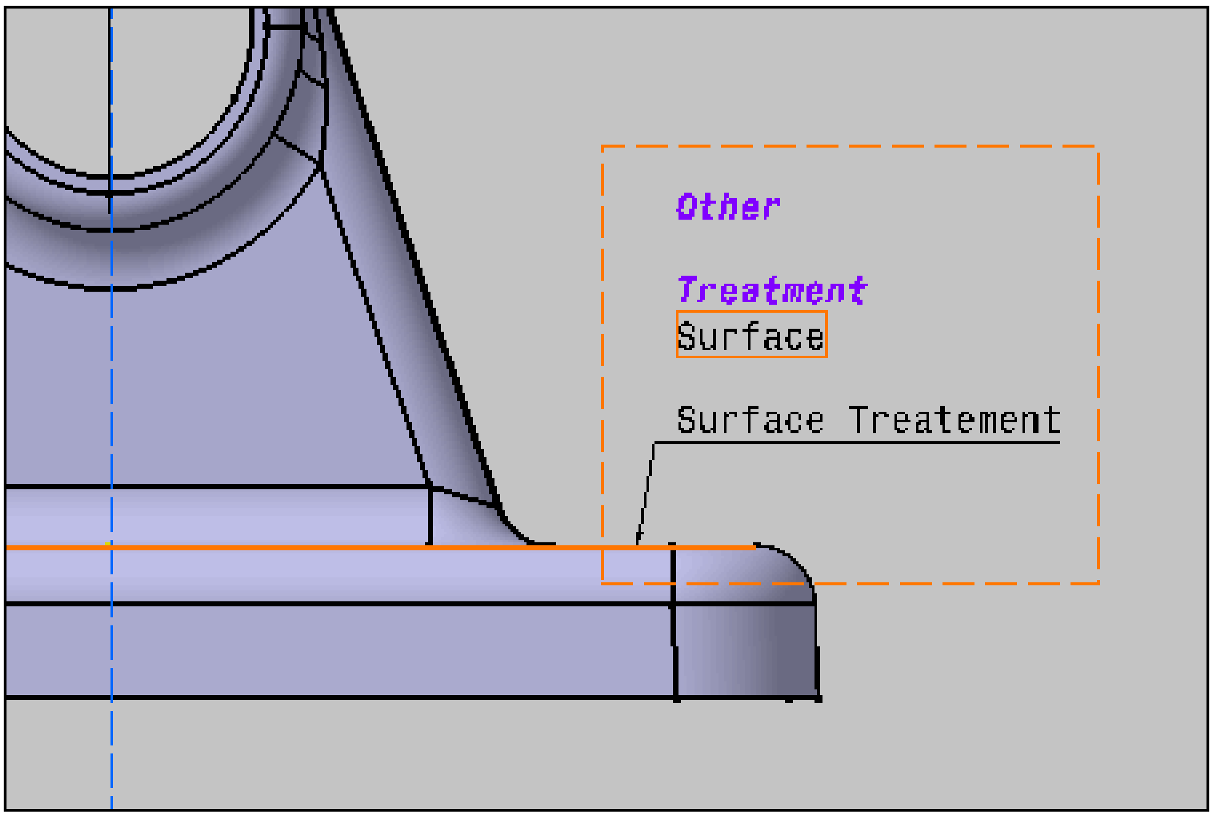
6. Select **Other Surface** (**Text.3**) as the new slave annotation.



Text.2 the previous slave annotation becomes the new active reference annotation in the **Positioning** dialog box.



Text.2 reference annotation is orange framed. **Text.3** annotation is the grouped annotation.

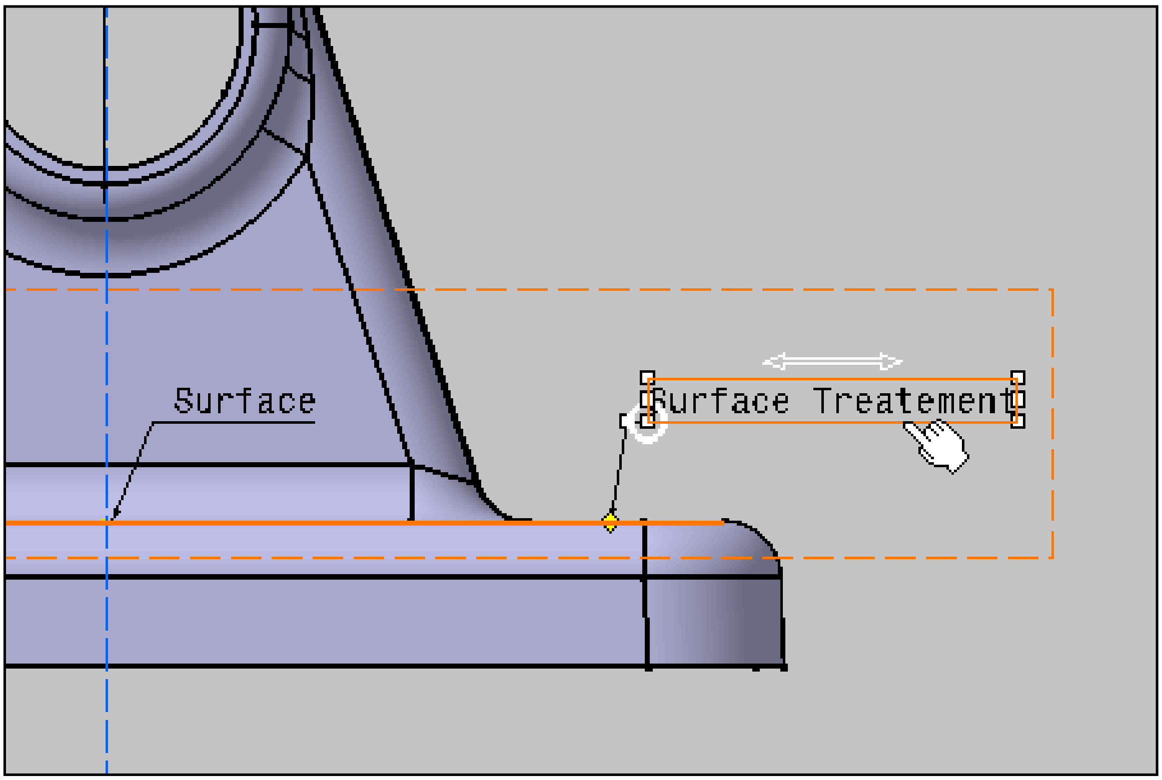


Making the Position of a Text Associative

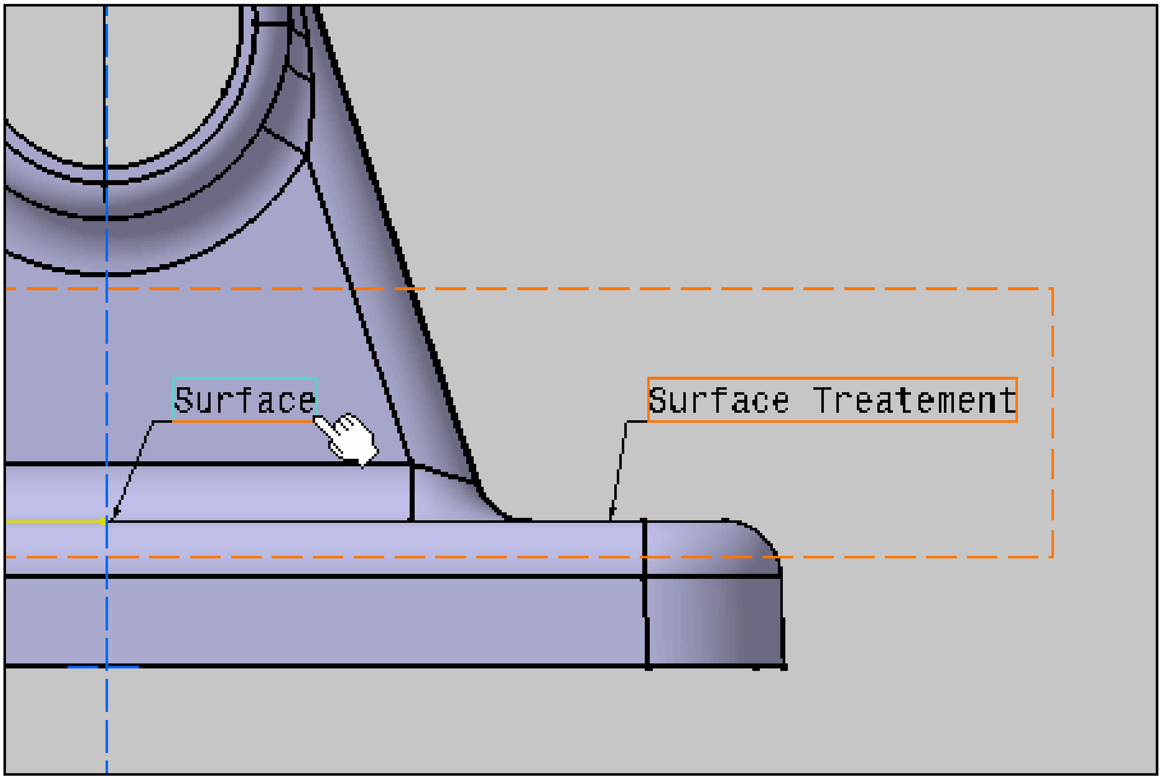
This task shows you how to set a positional link between a text and another element.
This allows you to move several annotations in only one interaction.

Open the Annotations_Part_04.CATPart document.

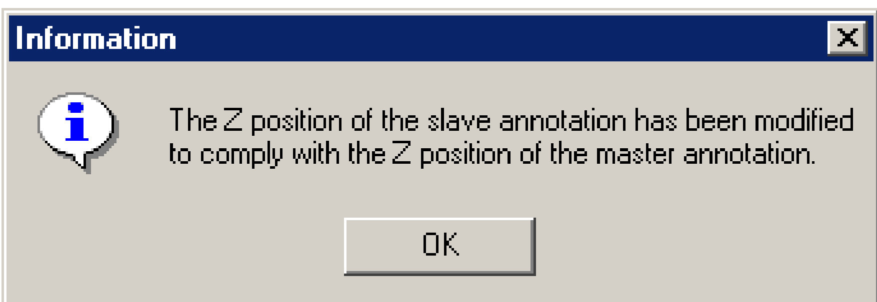
1. Right-click the slave text (text itself, frame or leader) and select the **Annotation Links -> Create Positional Link** contextual menu.



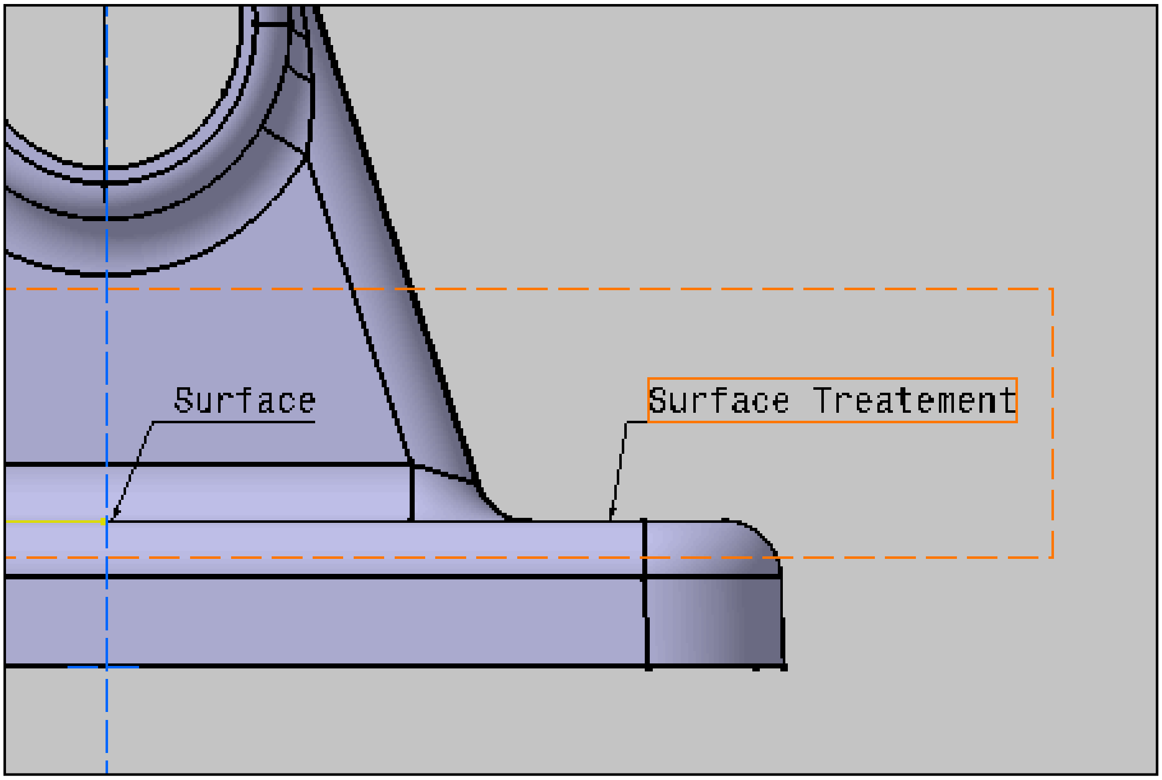
2. Select the master text (text itself, frame or leader).



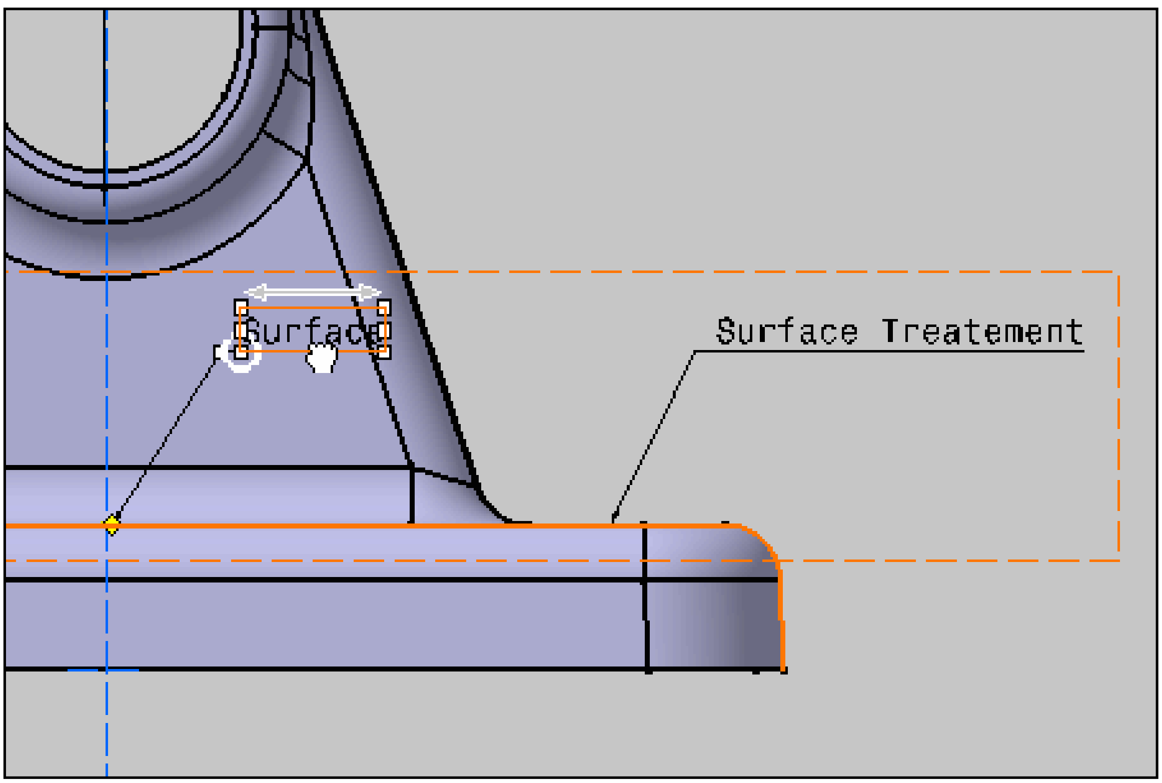
An information popup appears to warn you that the slave text is now at the master elevation.



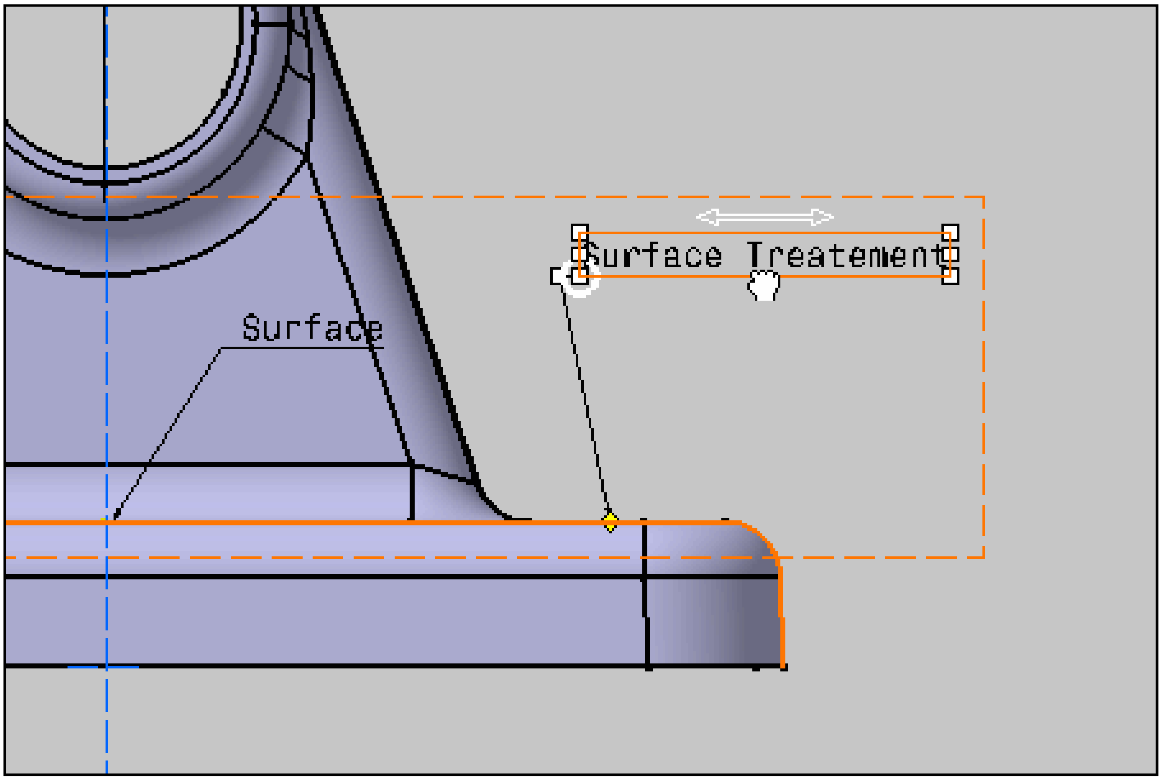
Master and slaves texts must belong to the same active view and associated geometrical element.



3. Move the master text: both texts are moving and their distance remains the same.

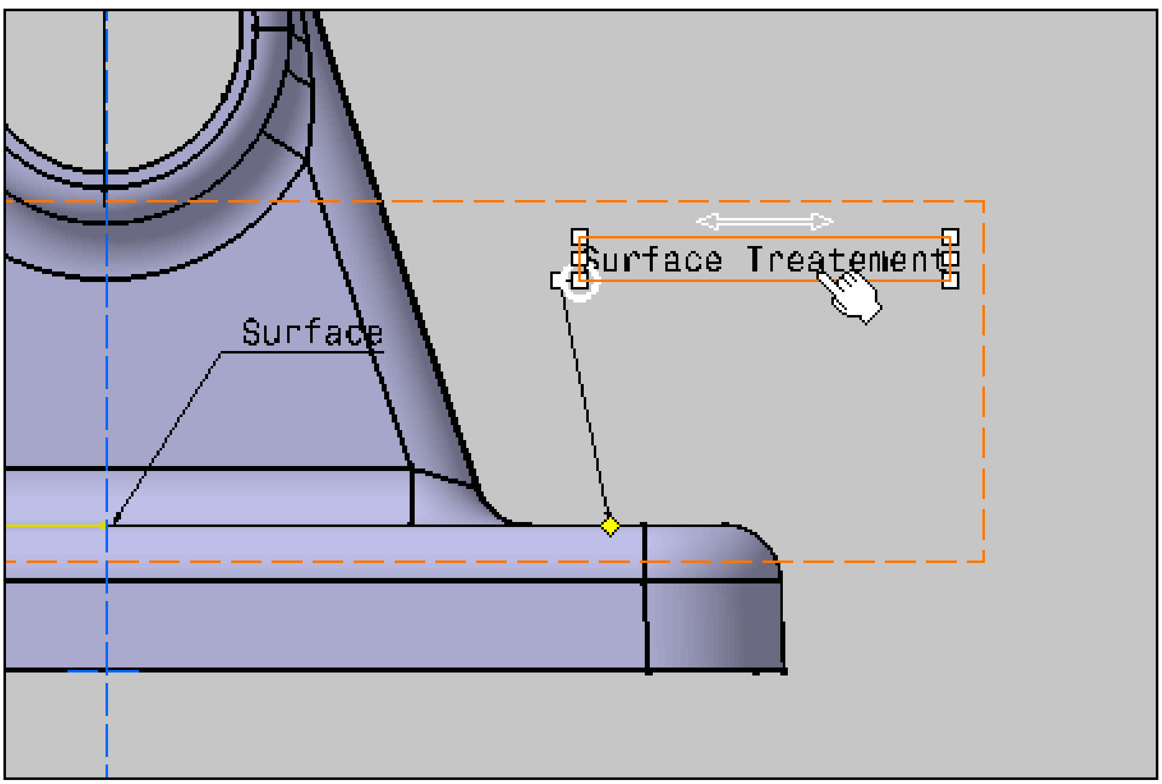


4. Now, if you move the slave text you selected, only this annotation is moved.

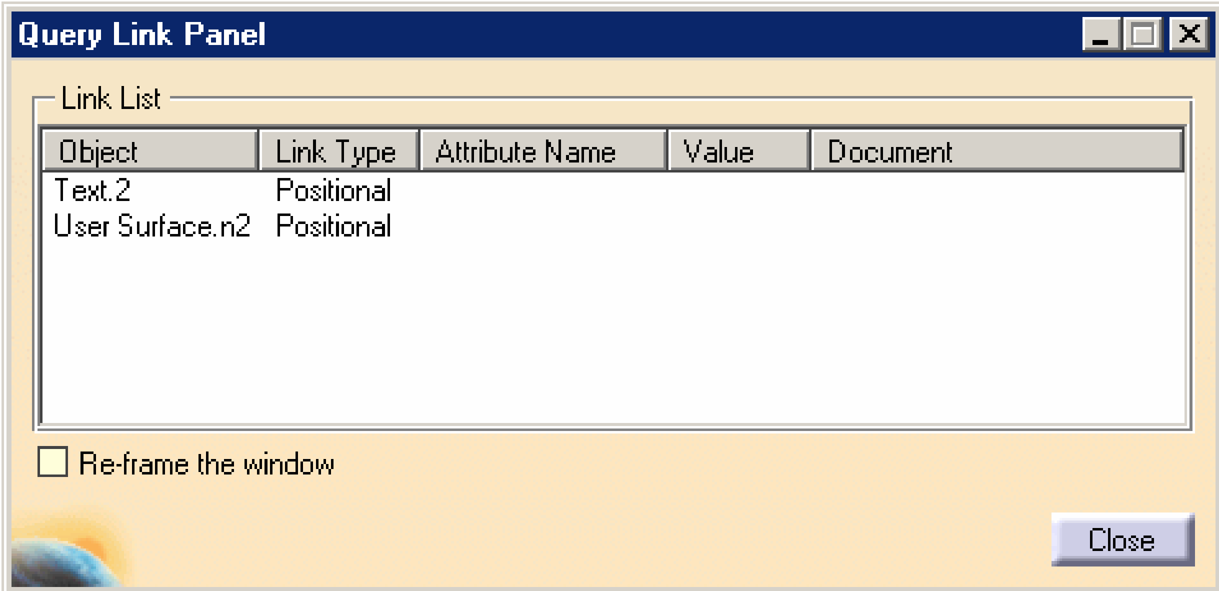


To delete the associativity, right-click the slave text and select the **Annotation Links -> Delete Positional Link** contextual menu.

5. Right-click the slave text and select the **Annotation Links -> Query Object Links** contextual menu.



The **Query Object Panel** dialog box appears.



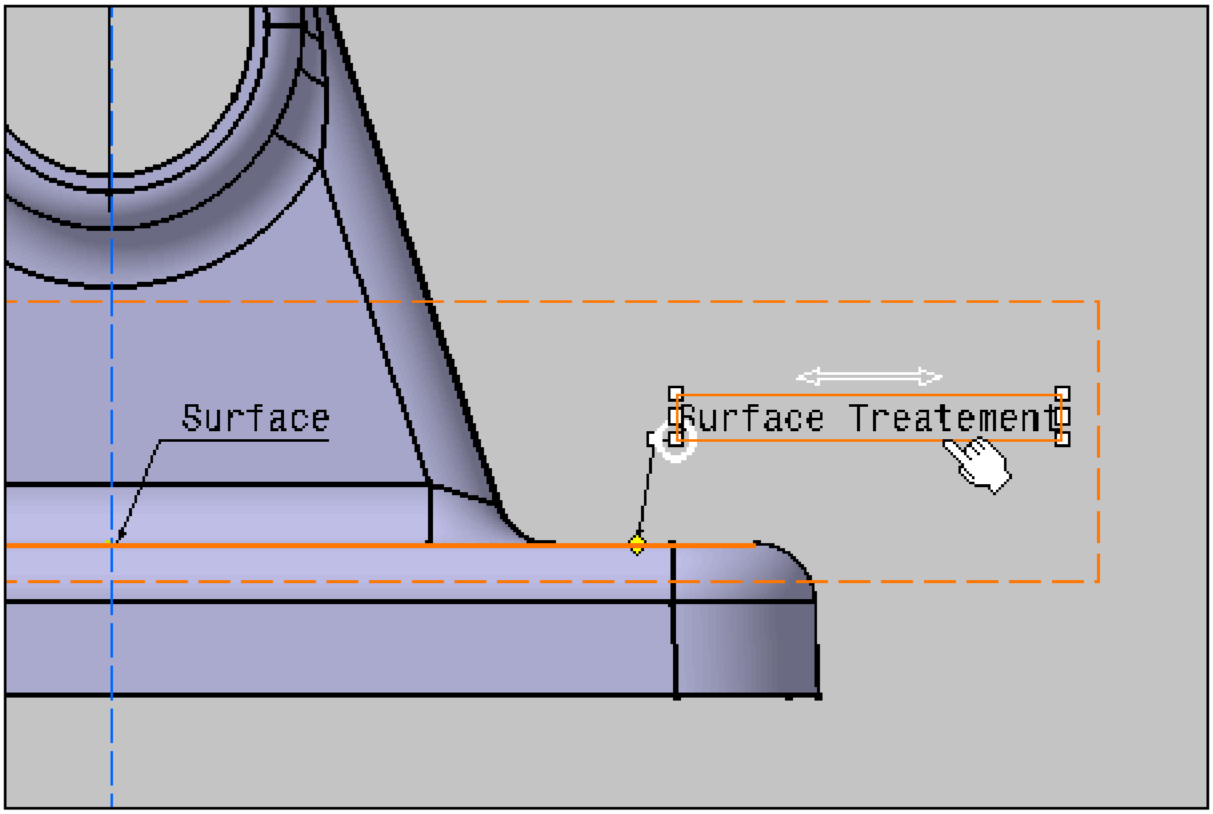
It show that a positional link has been created between this annotation and **Text.2** annotation.

Making the Orientation of a Text Associative

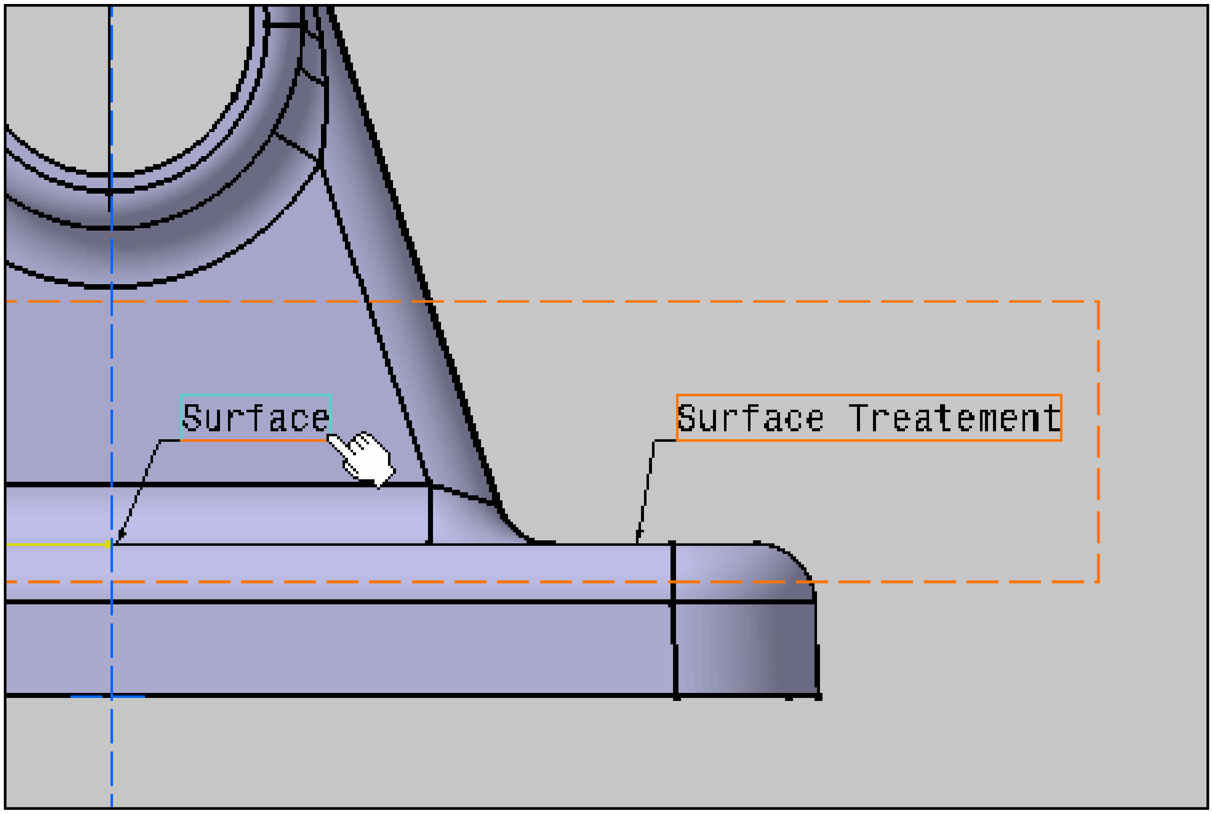
This task shows you how to set a orientation link between a text and another element.
This allows you to rotate several annotations in only one interaction.

Open the Annotations_Part_04.CATPart CATPart document.

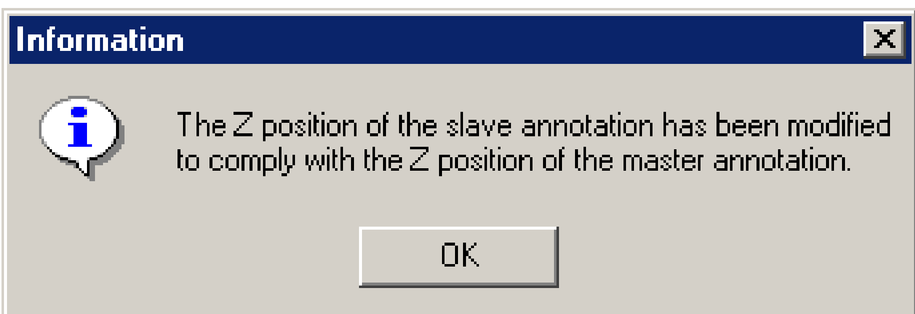
1. Right-click the slave text (text itself, frame or leader) and select the **Annotation Links -> Create Orientation Link** contextual menu.



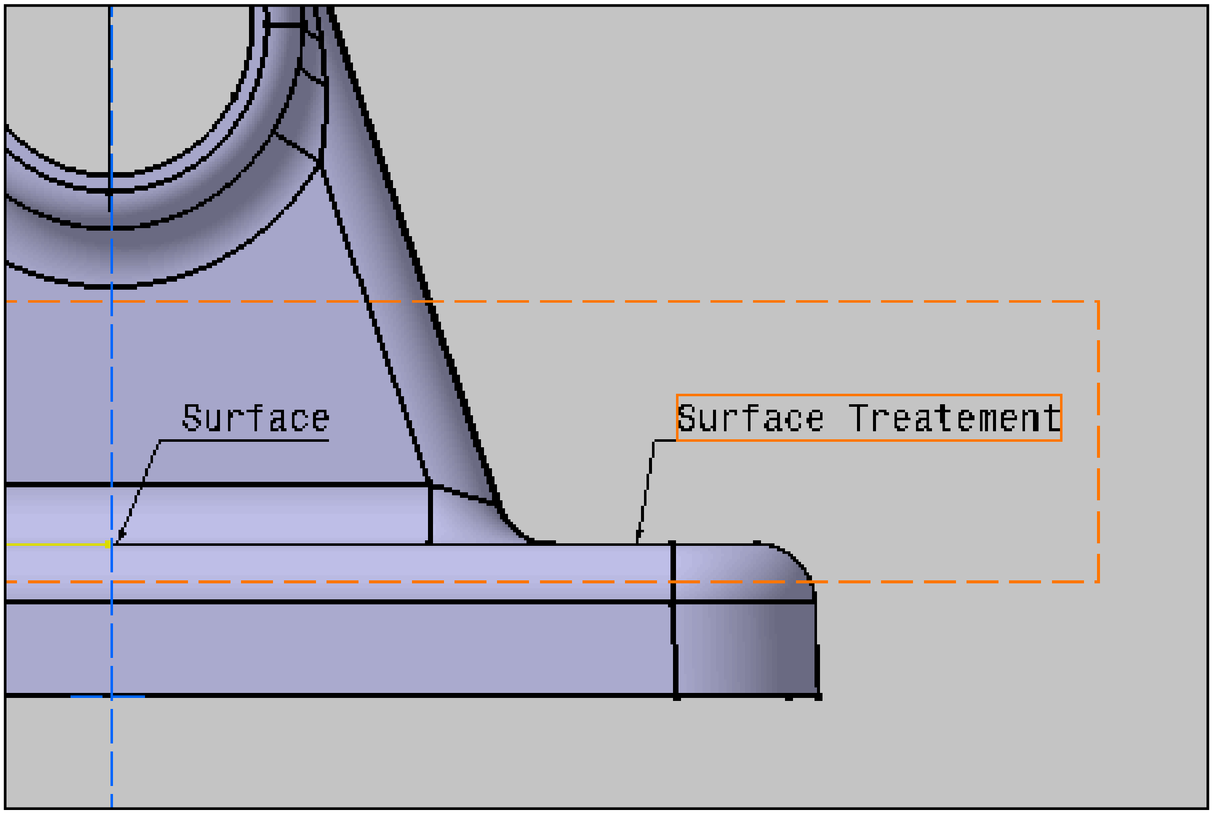
2. Select the master text (text itself, frame or leader).

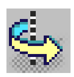


An information popup appears to warn you that the slave text is now at the master elevation.

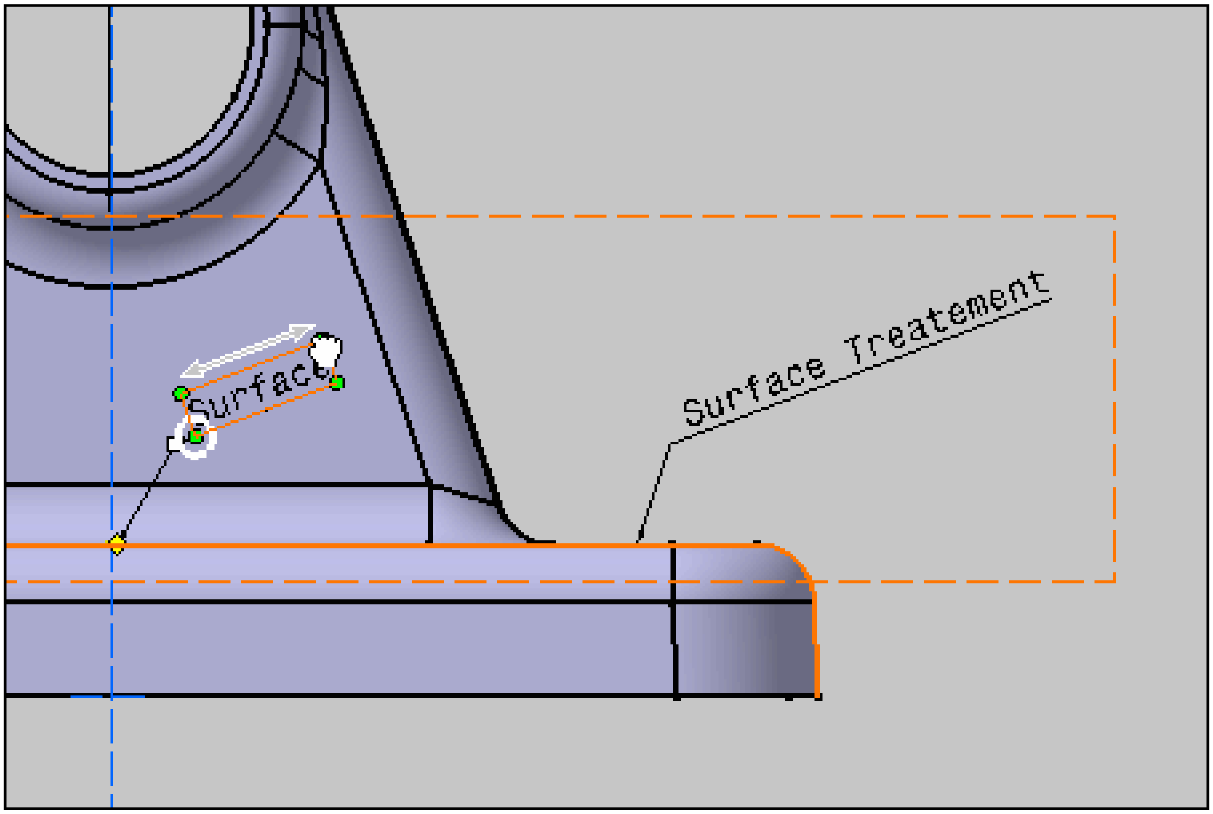


Master and slaves texts must belong to the same active view and associated geometrical element.

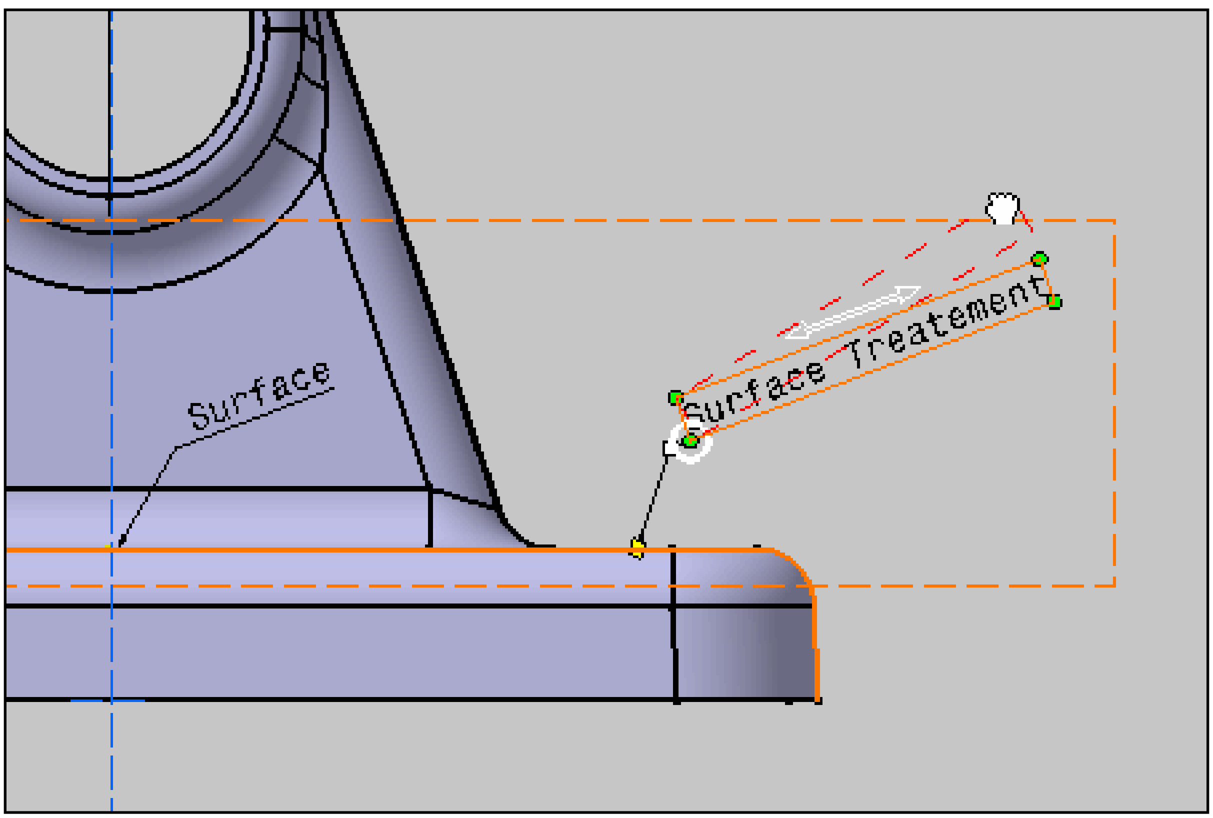


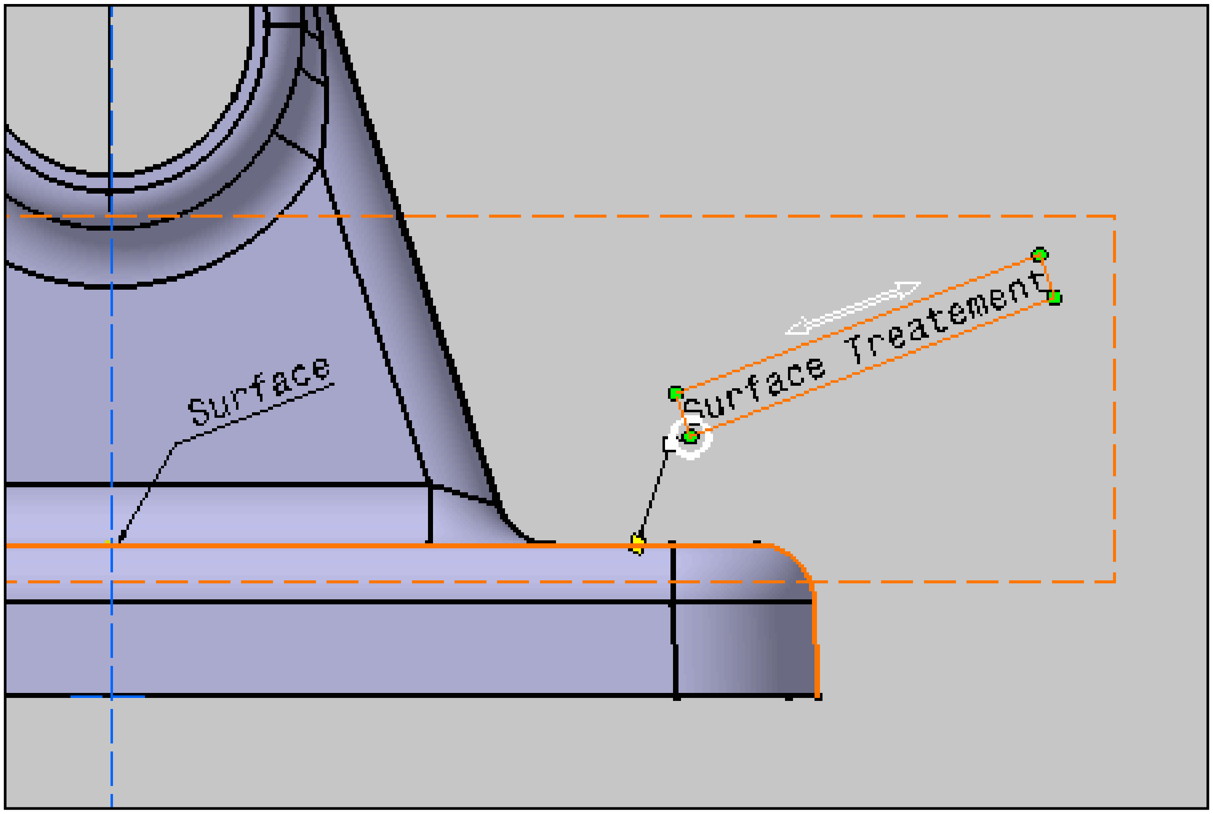
3. Select the **Free rotation** icon: 

4. Rotate the master text: both texts are rotating with the same angle.



5. Now, if you rotate the slave text you selected, this annotation is not rotated.





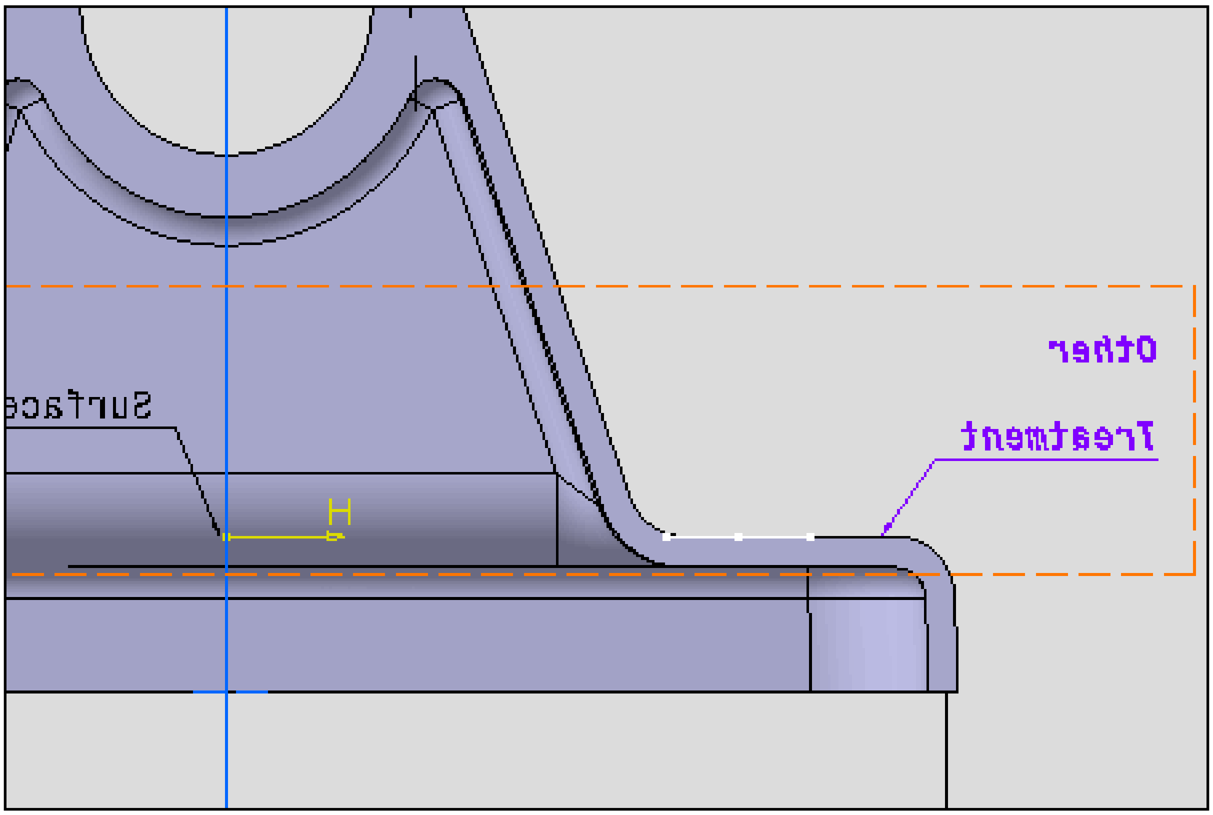
To delete the associativity, right-click the slave text and select the **Annotation Links -> Delete Orientation Link** contextual menu.

Mirroring Annotations

This task shows you how to mirror reversed annotation relative to the screen view.

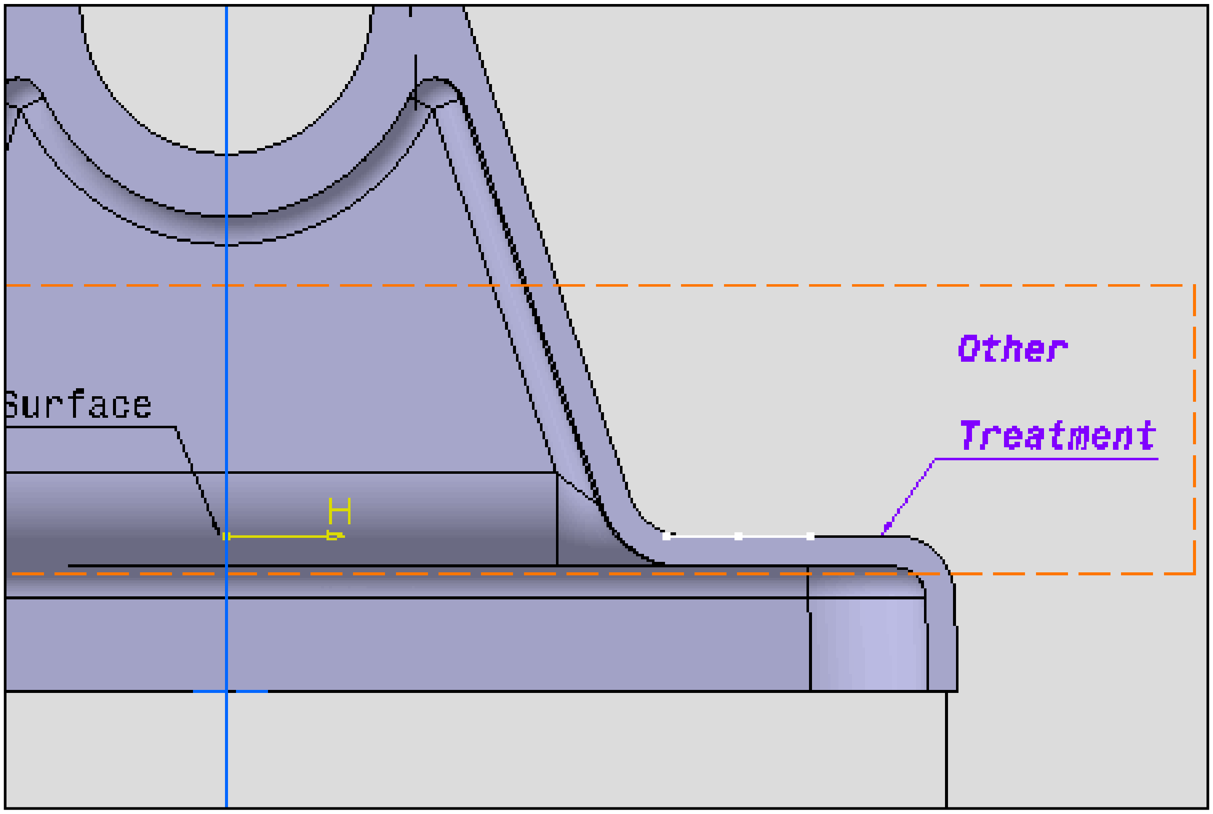
Open the Annotations_Product_04.CATProduct document.

1. Turn the part to show annotations reversed.



2. Click the **Mirror Annotations** icon

All reversed annotations are mirrored.

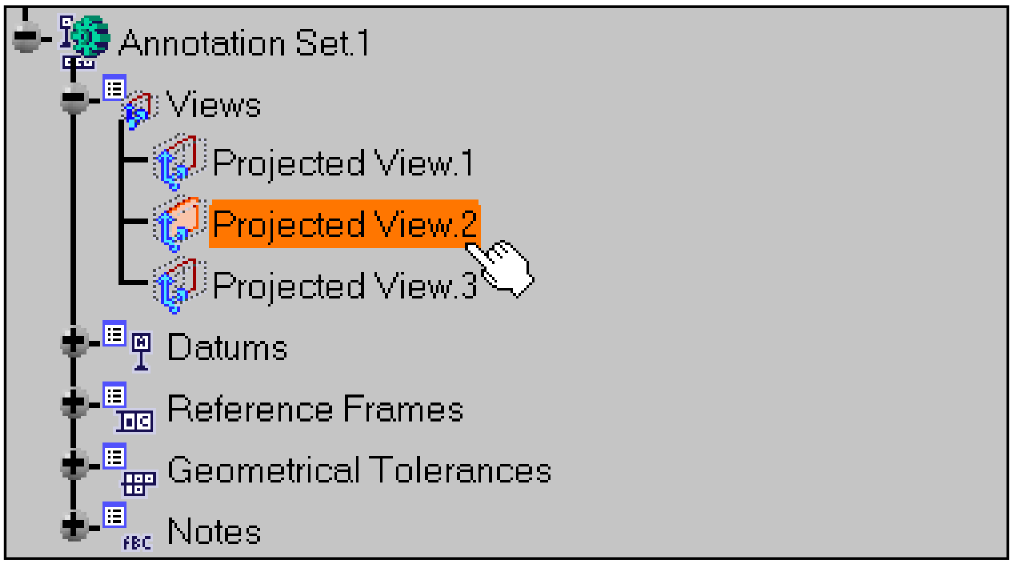

Surface
H
Other
Treatment

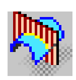
Clipping Annotations Plane

This task shows you how to clip a part according to the annotation plane in relation to its normal.

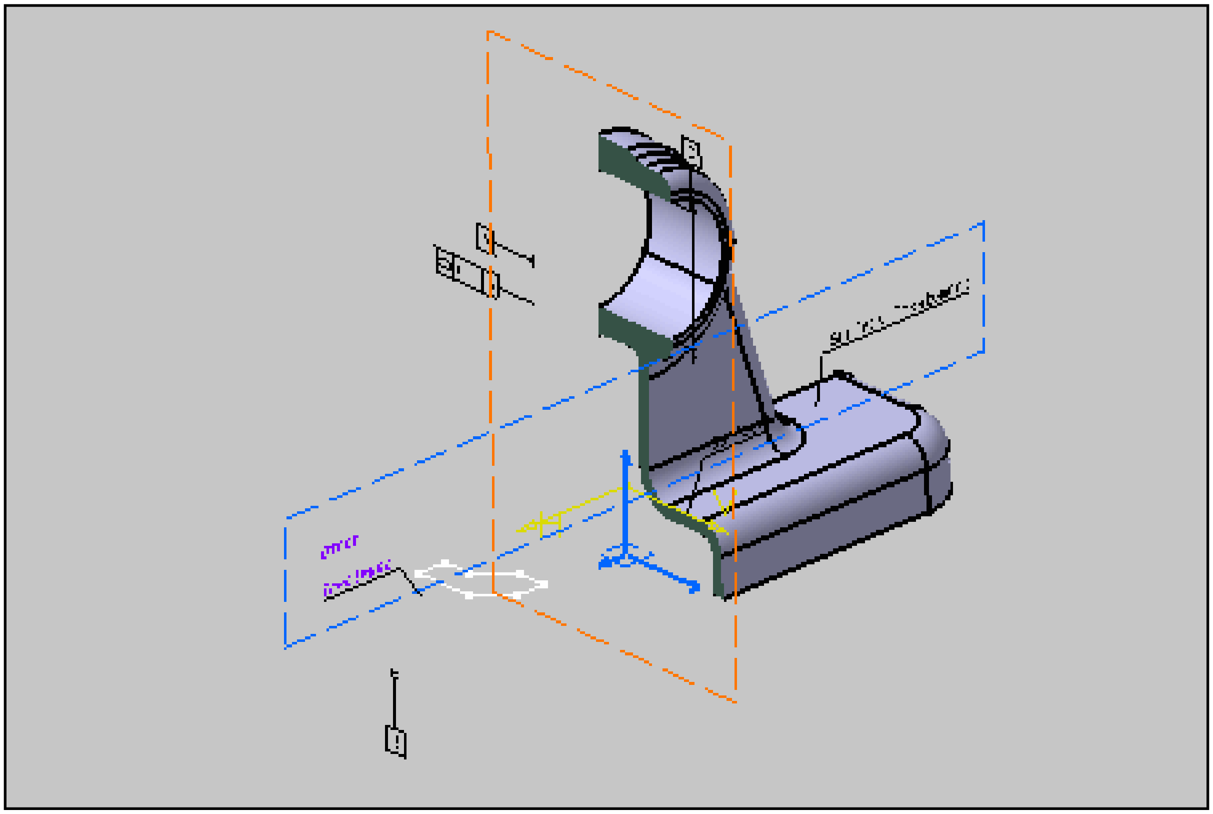
Open the Annotations_Part_04.CATPart document.

1. Activate the **Projected View.2** annotation plane.



2. Click the **Clipping Plane** icon 

The part is clipped according to the annotation plane in relation to its normal.

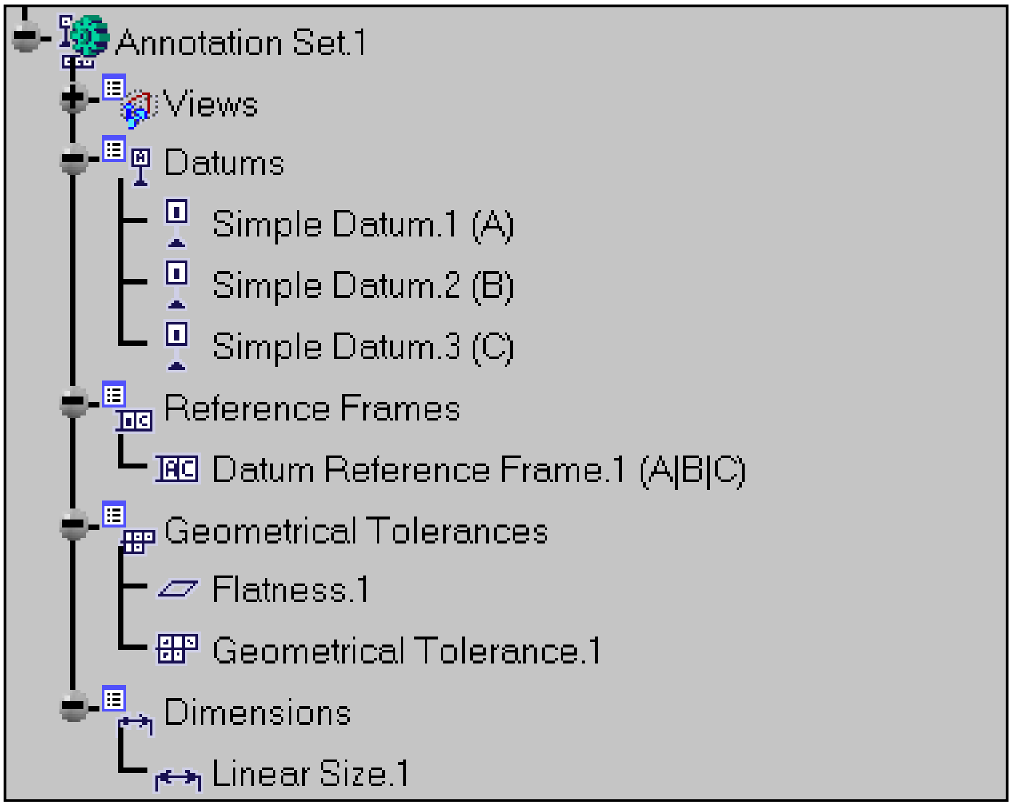


Marking Non-semantic Annotations

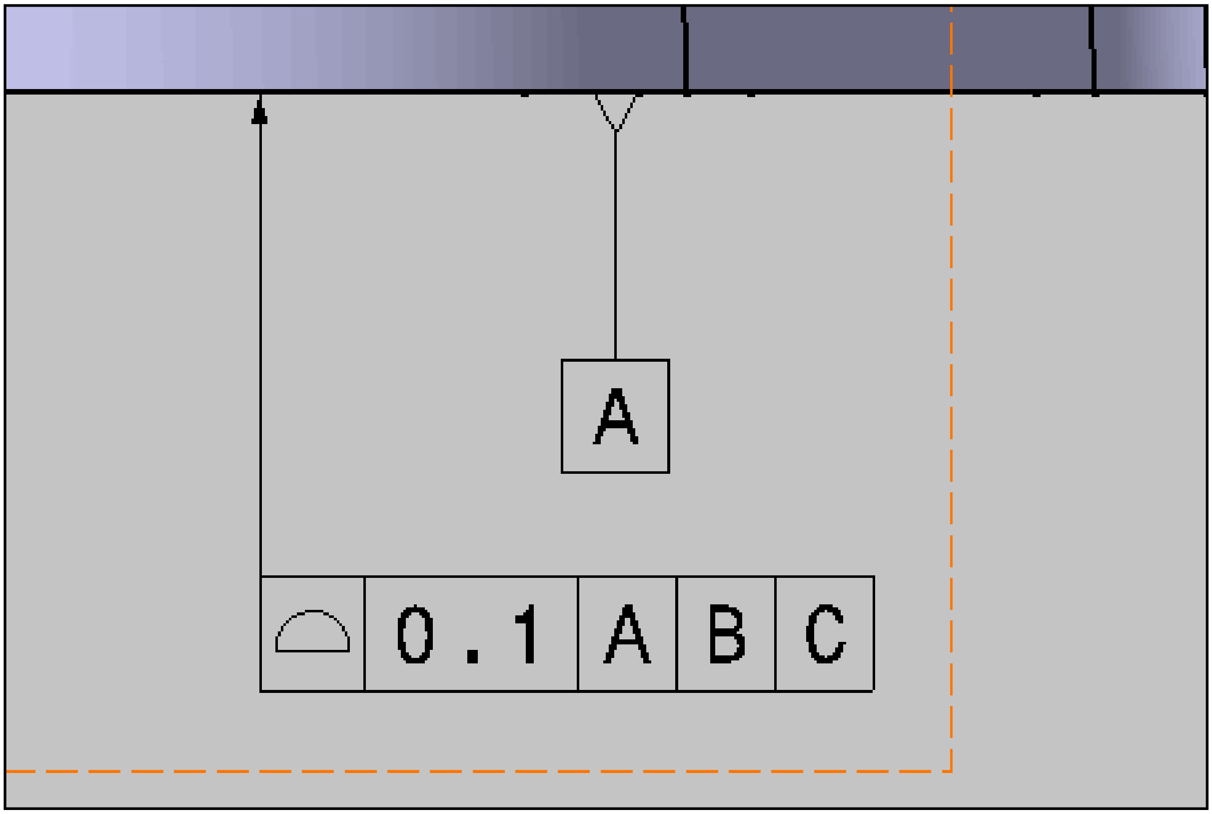
This task shows you how to mark non-semantic annotations with a wavy red line in the specification tree and geometry.
It allows you to graphically distinguish non-semantic annotations (datum, datum targets, geometric tolerances, dimensions) from semantic one, see Concepts.

Open the Tolerancing_Annotations_03 CATPart document.

The specification tree looks like this:

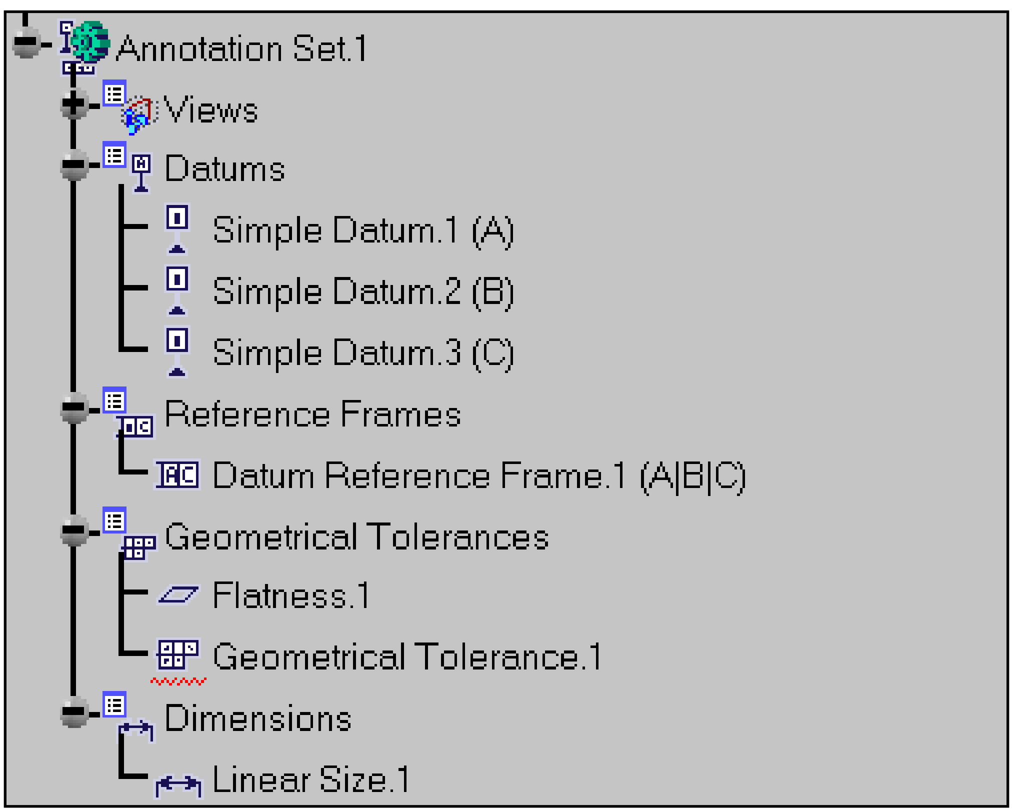


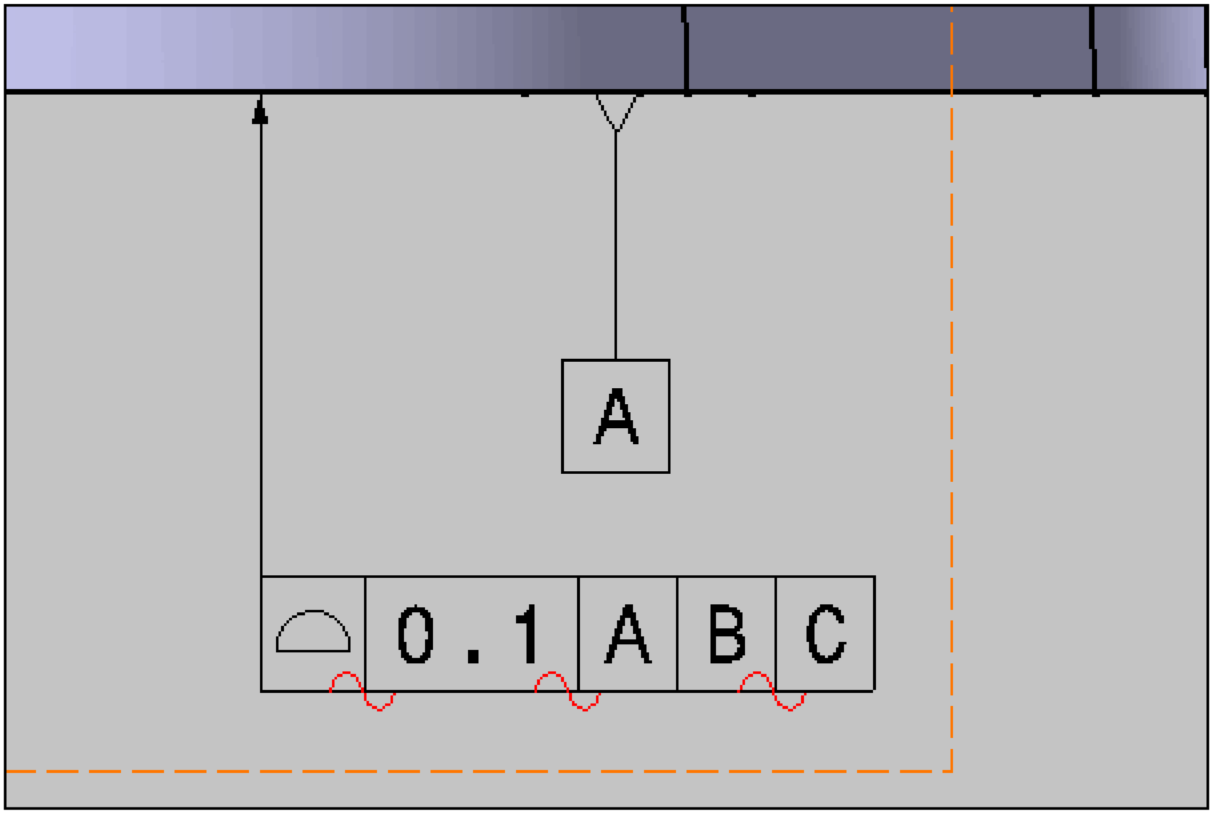
The geometry looks like this:



1. Check the **Mark non-semantic annotation** option. See Display setting.

The **Geometrical Tolerance.1** annotation is a non-semantic annotation and so it is marked in the specification tree and the geometry:




A
0.1
A
B
C

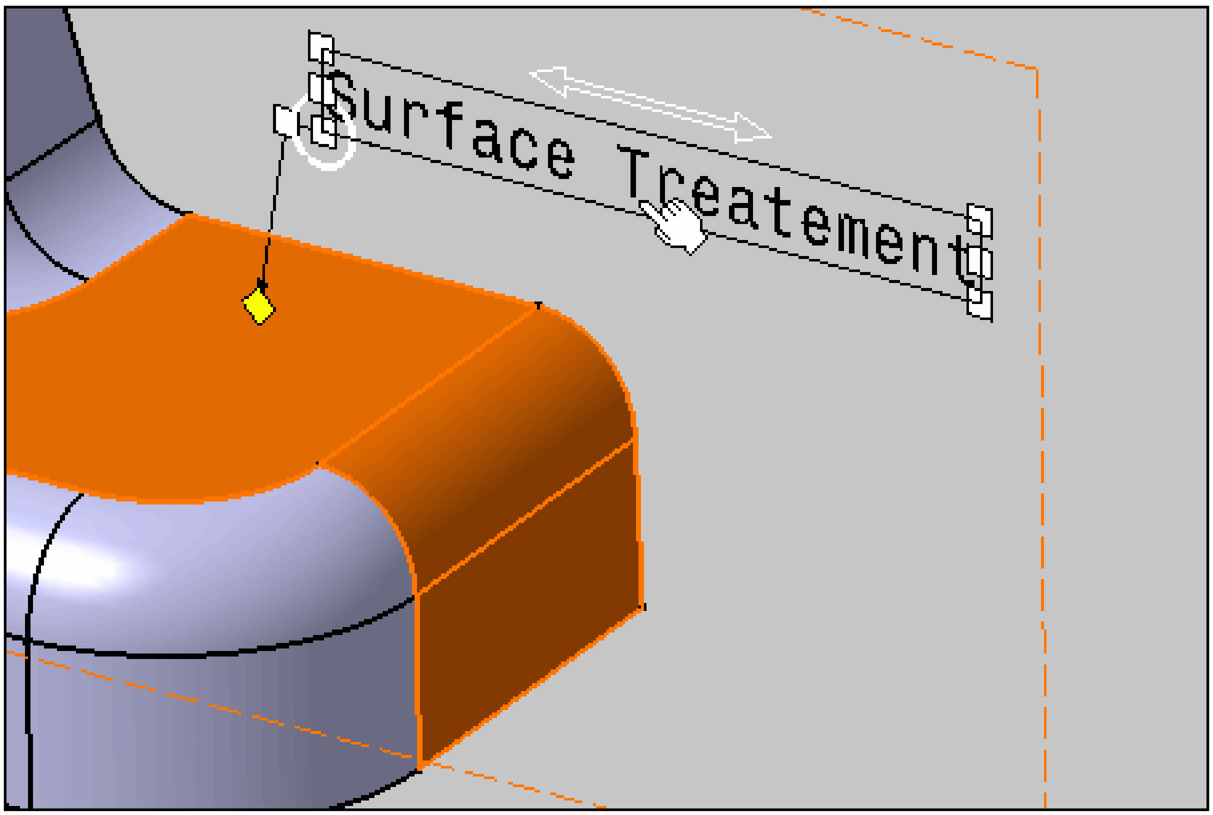
Setting Annotation Parallel to the Screen

This task shows you how to set an annotation text parallel to the screen.
The operating mode described here applies to text, flagnote and note object attribute too.

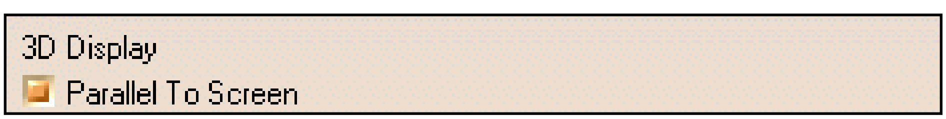
Open the Annotations_Part_04.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Right-click the annotation text and select **Properties** from the contextual menu, and click the **Display** tab.

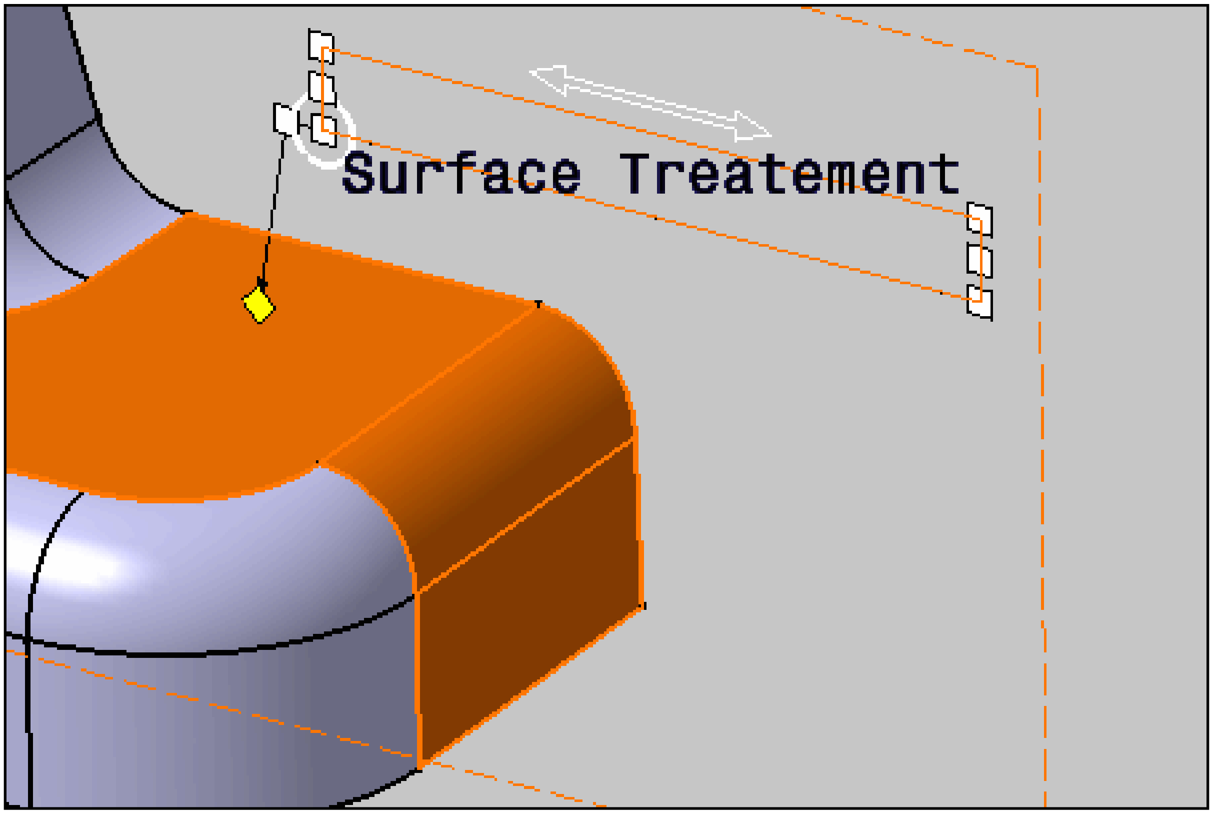


2. Check **Parallel to Screen** option.



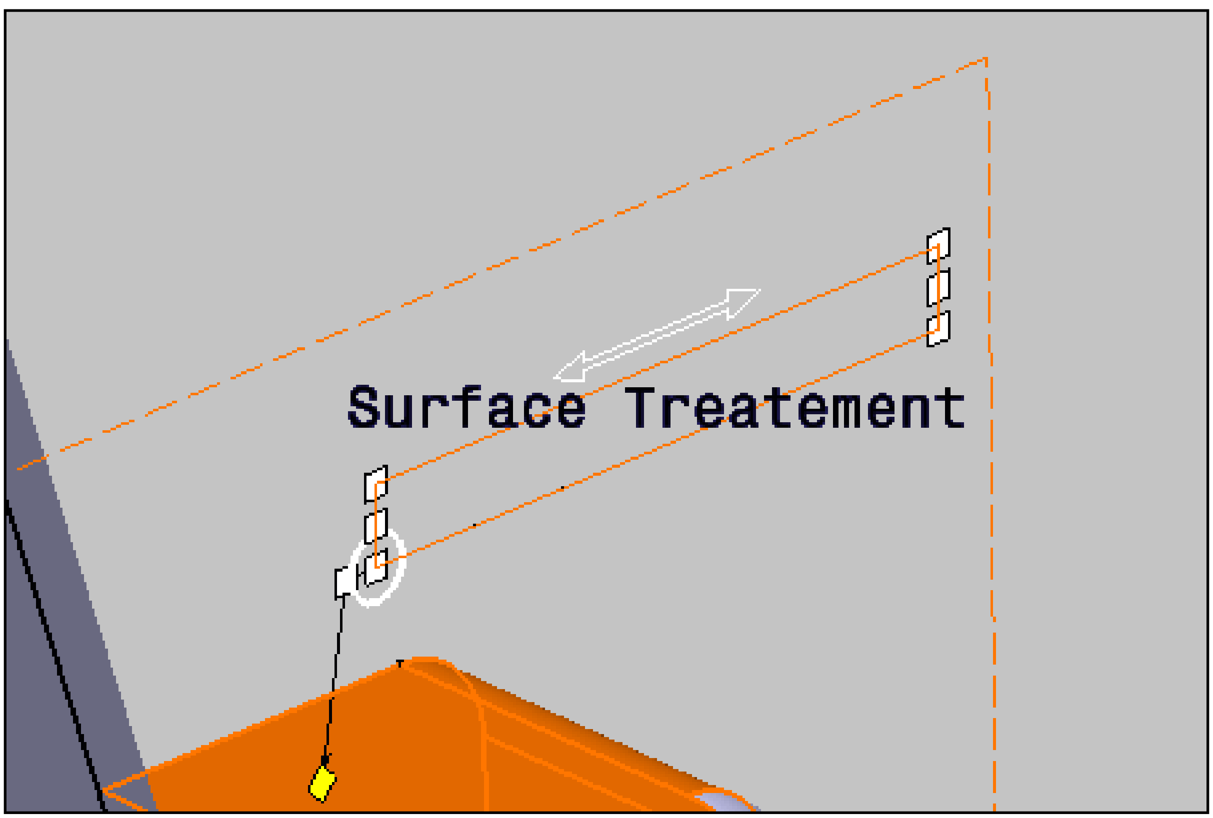
3. Click the **OK**.

The text is now parallel to the screen.



4. Move and rotate the part..

The text is always parallel to the screen and its size is constant for any point of view or zoom value.

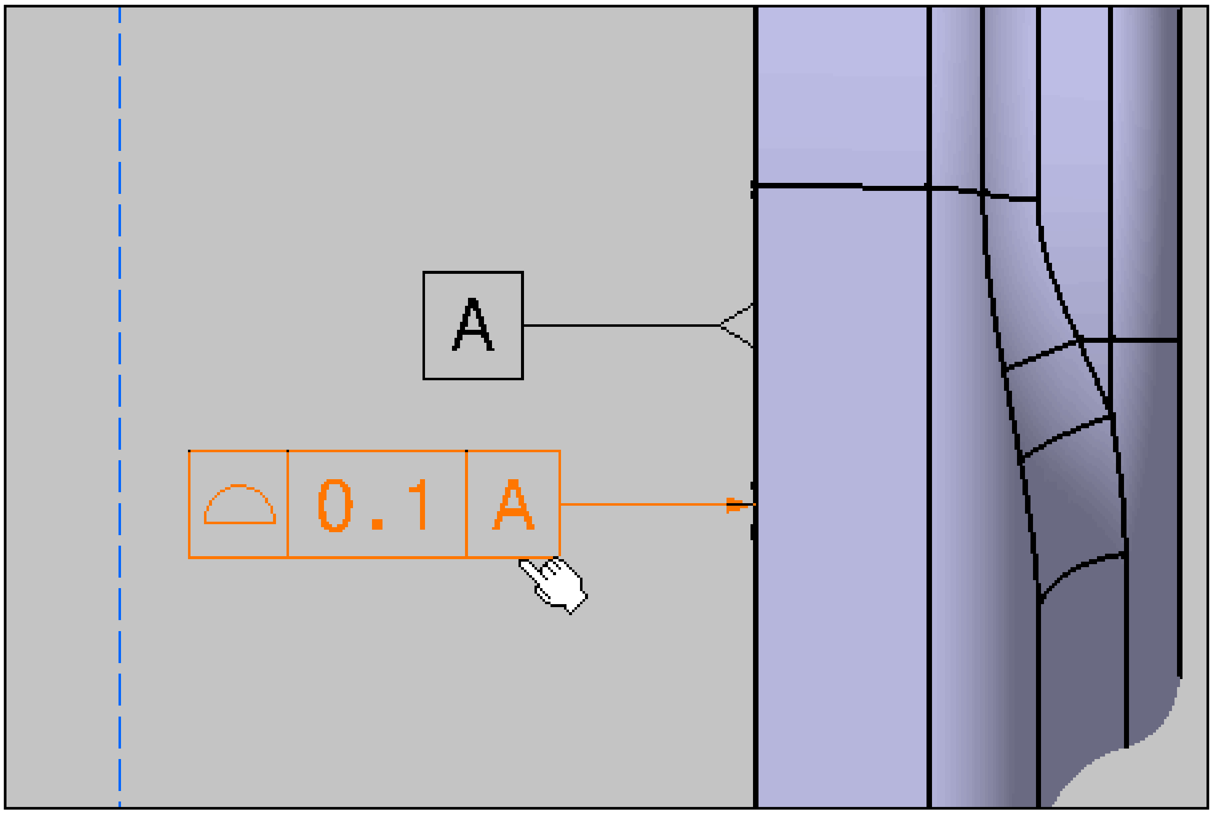


Replacing a Datum Reference Frame

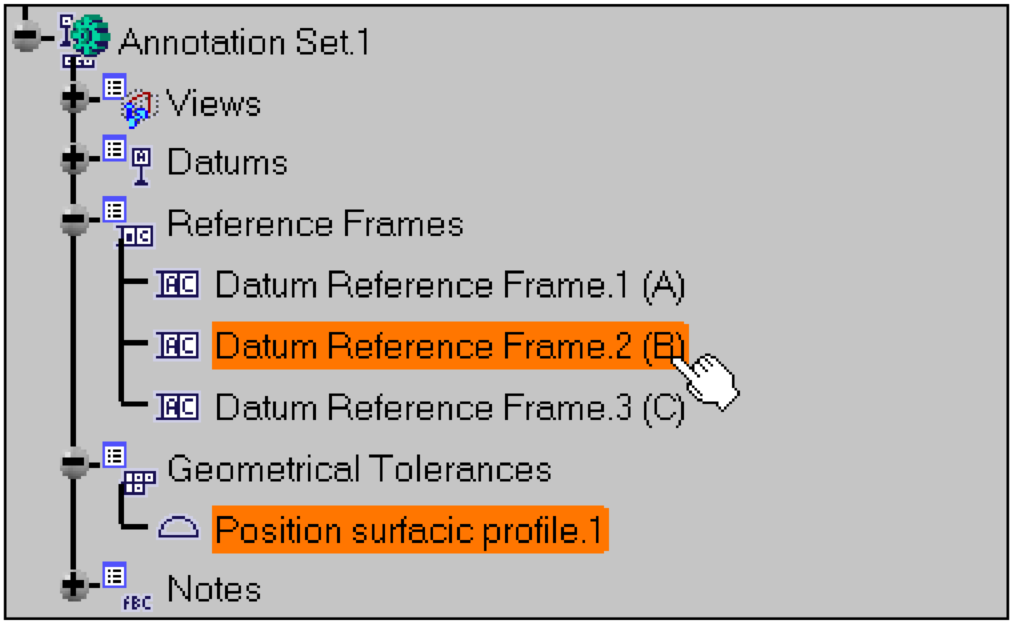
This task shows you how to replace a datum reference frame in an annotation.

Open the Annotations_Part_04.CATPart document.

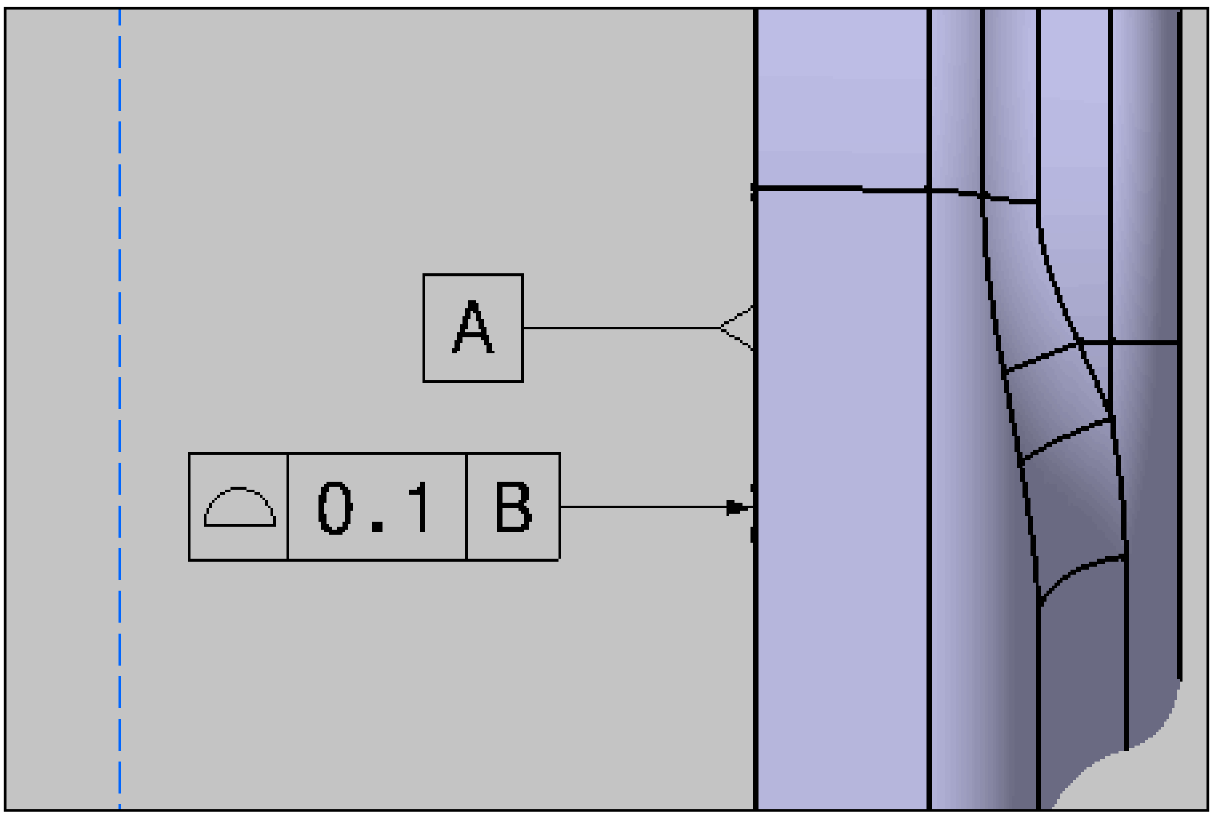
1. Right-click the annotation and select the **Replace Datum Reference Frame** contextual command.



2. Select the **Datum Reference Frame.2 (B)** in the specification tree.



The datum reference frame is replaced.

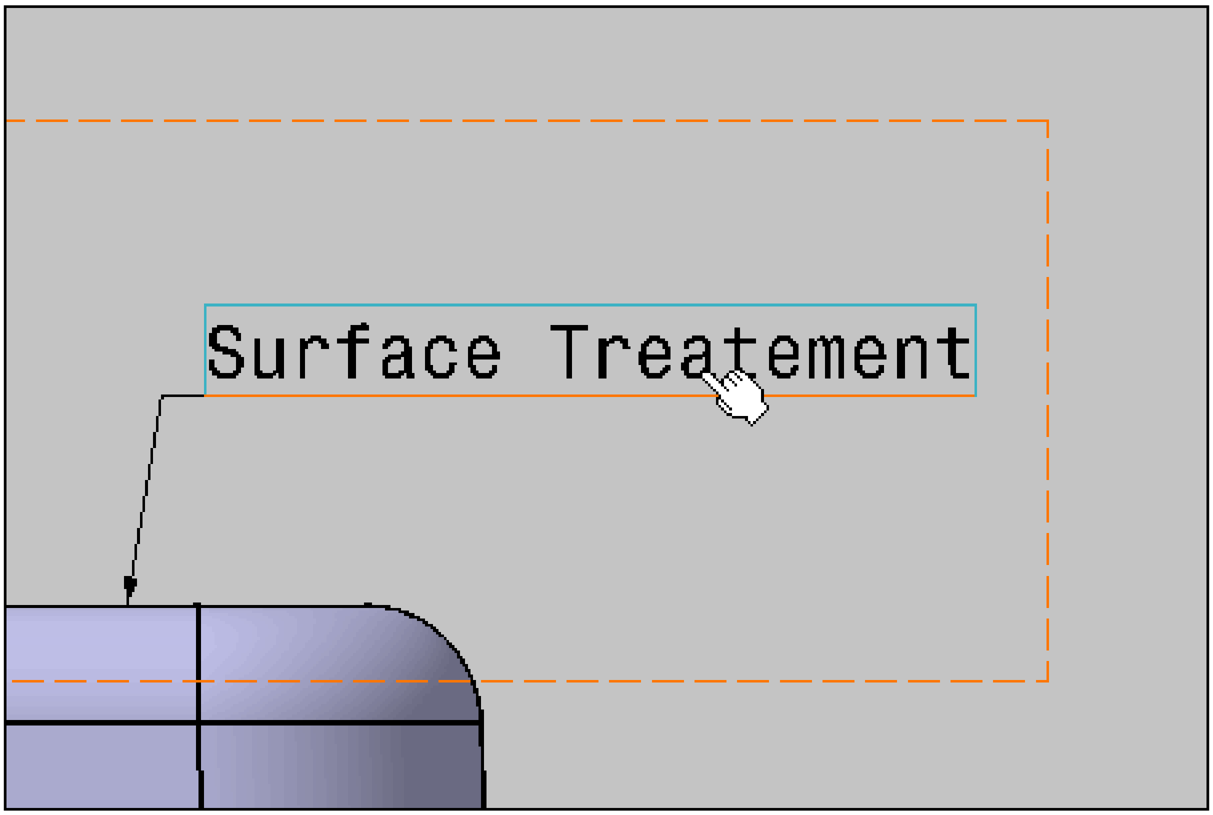


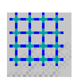
Using a 3D Grid

This task shows you how to use the 3D grid to position the annotations of an annotation view.
See Display settings to customize the grid.

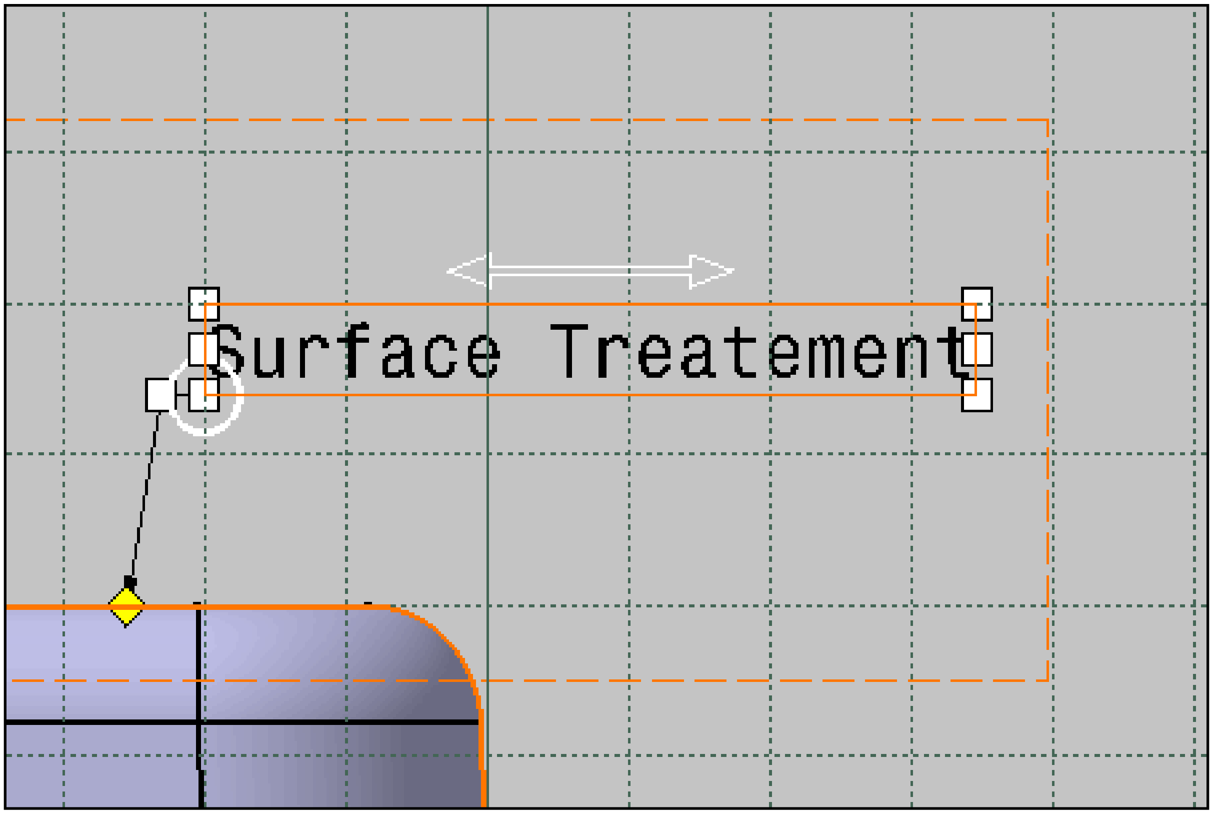
Open the Annotations_Part_04.CATPart document.

1. Select the annotation.

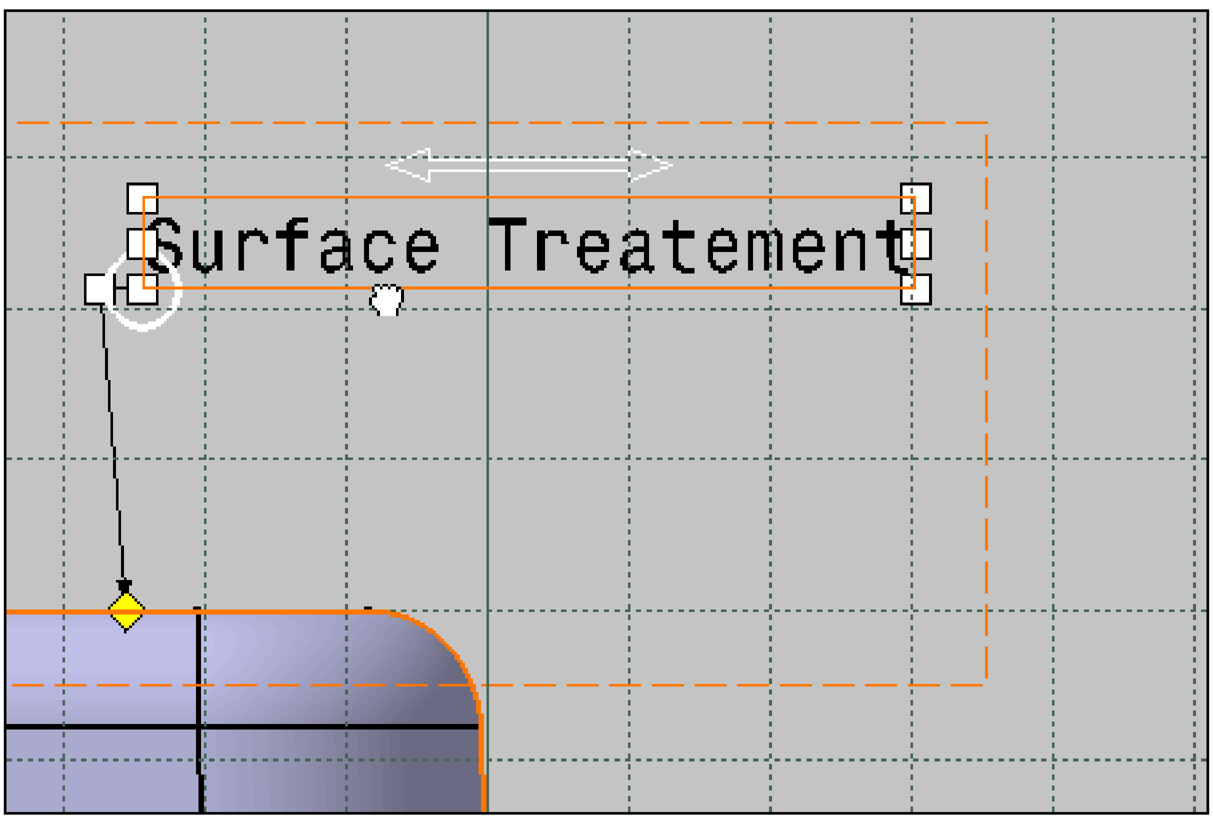


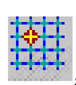
2. Click the **Display 3D Grid** icon: 

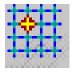
The 3D grid appears.



3. Drag anywhere the annotation.

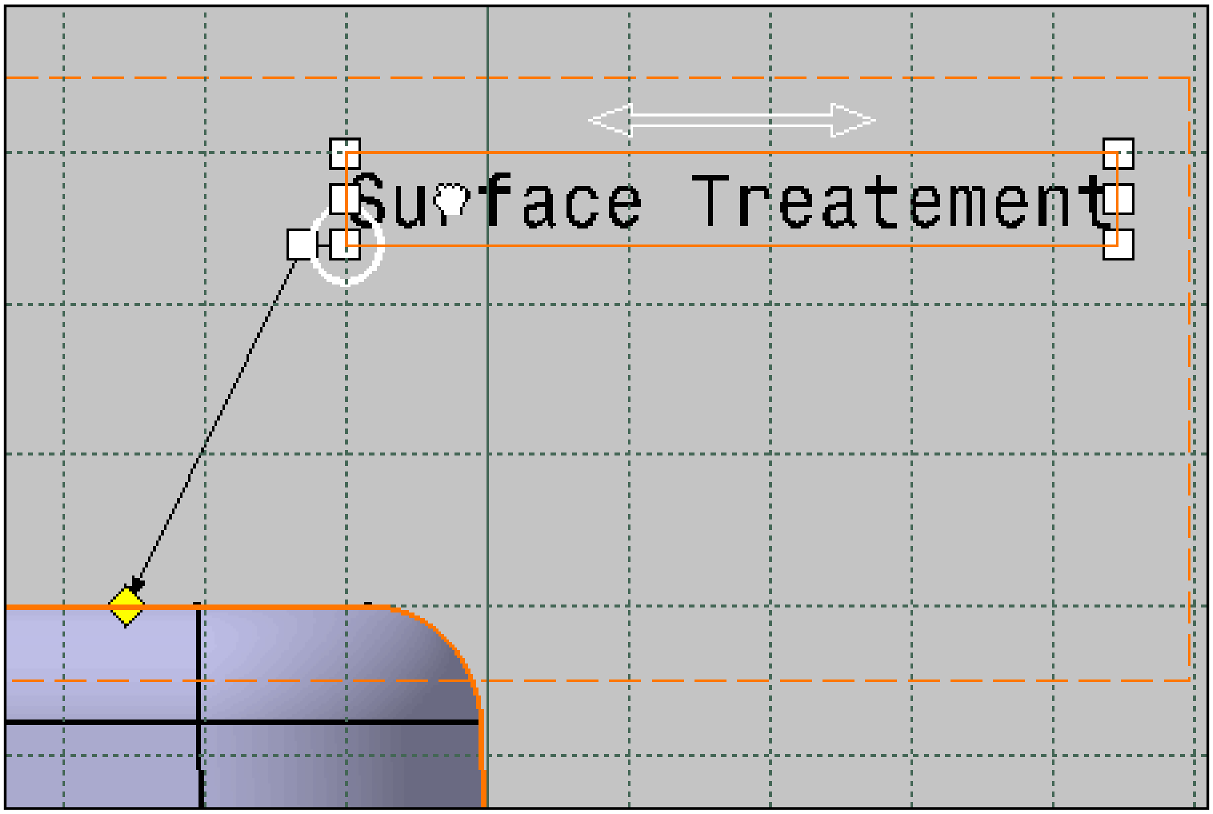


While the **Snap to Point** icon is deactivated , the annotation may be moved without any connection to the grid.

4. Click the **Snap to Point** icon: 

5. Drag anywhere the annotation.

The annotation snaps to each point of the grid according to its anchor point. See Text Properties Toolbar.



You may temporarily reverse the Snap to Point status pressing the **Shift** key.

Managing Annotation Leaders

Adding Leaders and Using Breakpoints

This task shows you how to add:

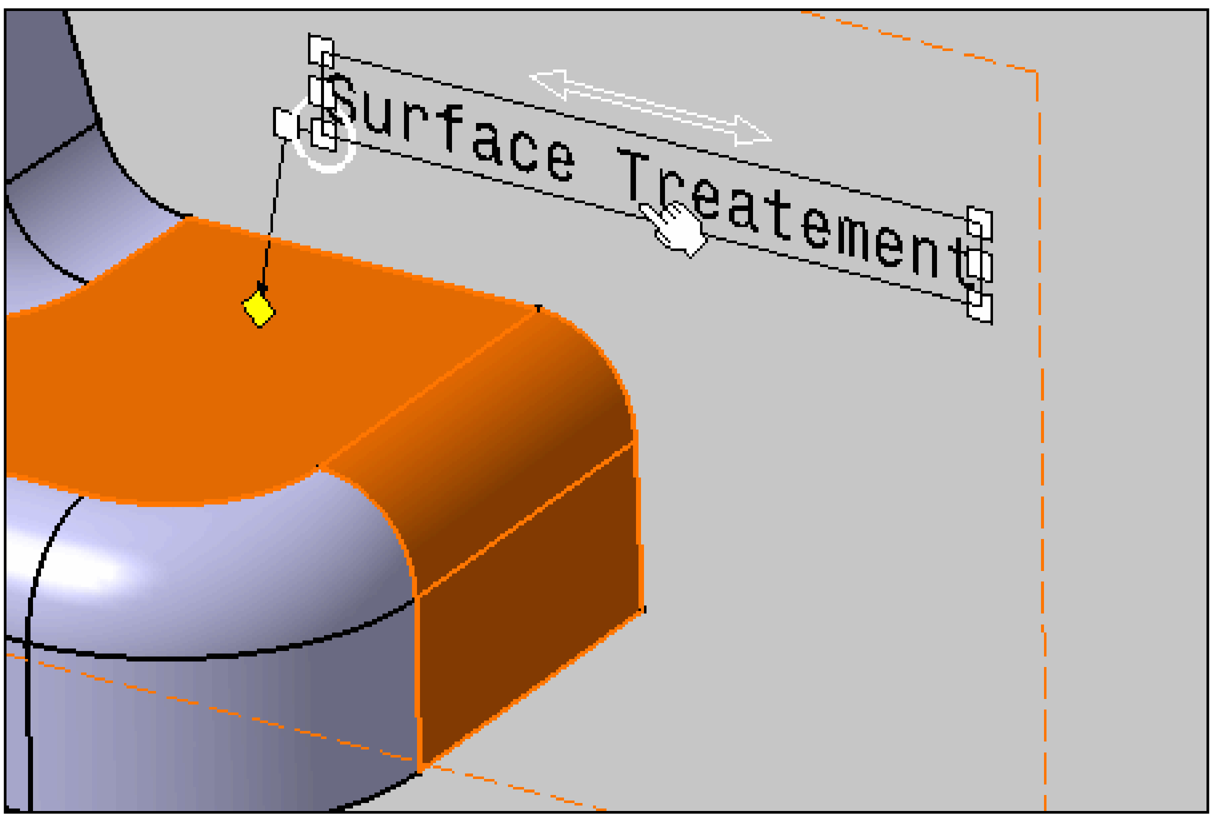
- A leader to an annotation.
- A breakpoint on the created leader.
- A leader from the created breakpoint.

You can add a leader to geometrical elements associated with the annotation only.

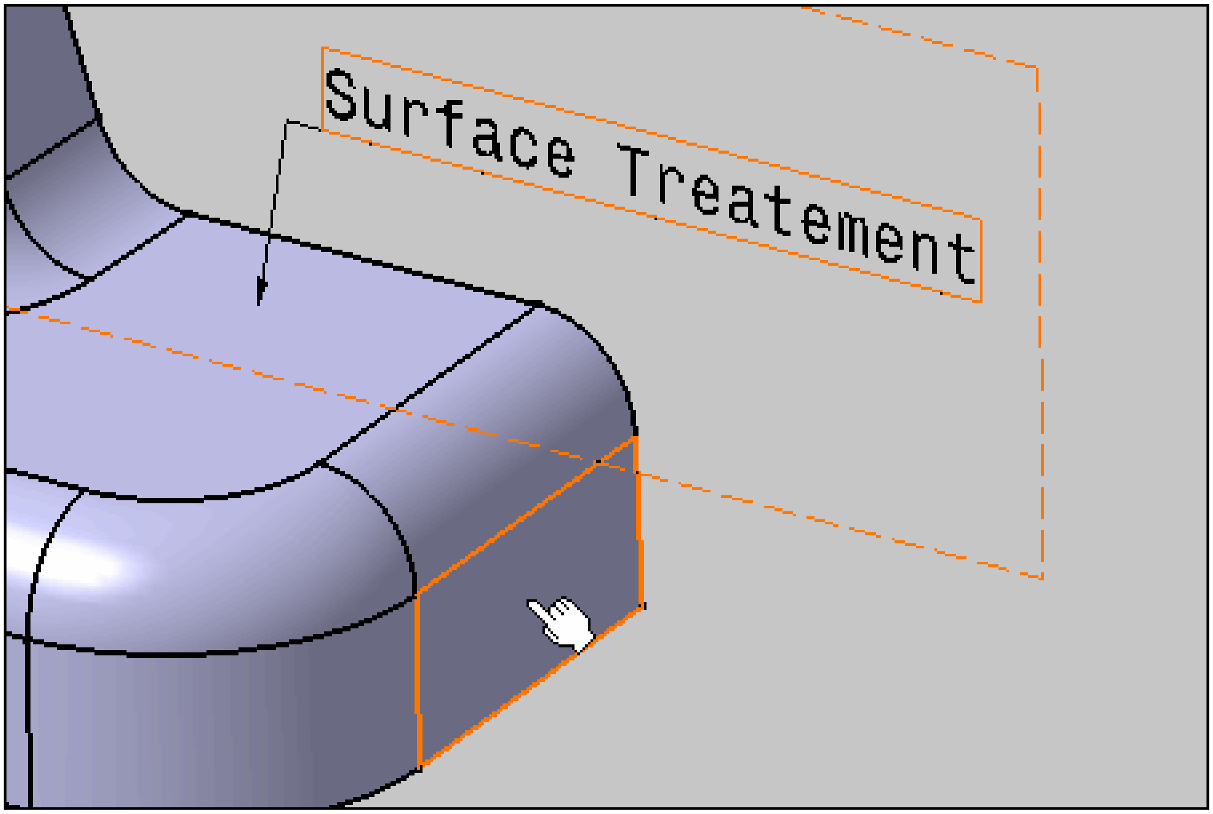
Open the Annotations_Part_04.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

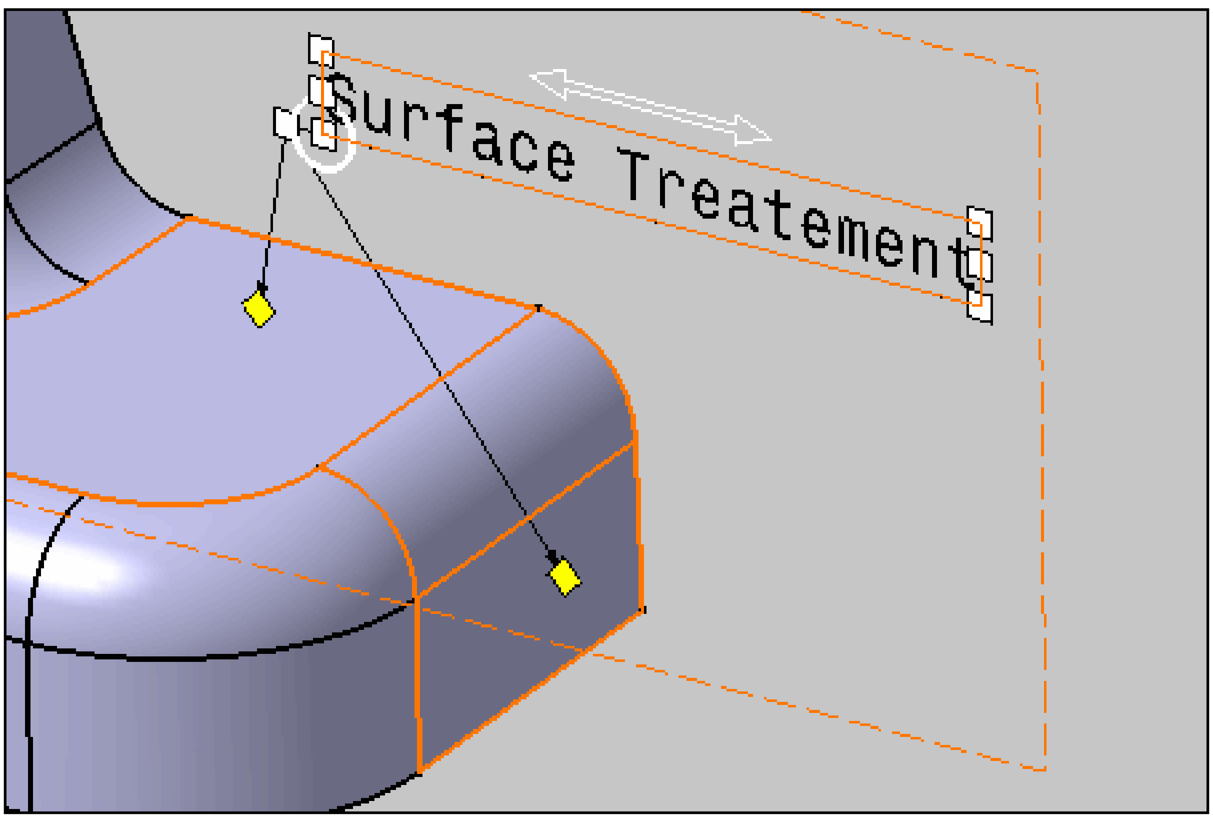
1. Right-click the annotation text and select **Add Leader** from the contextual menu.



2. Click the face as shown to begin the leader (arrow end).

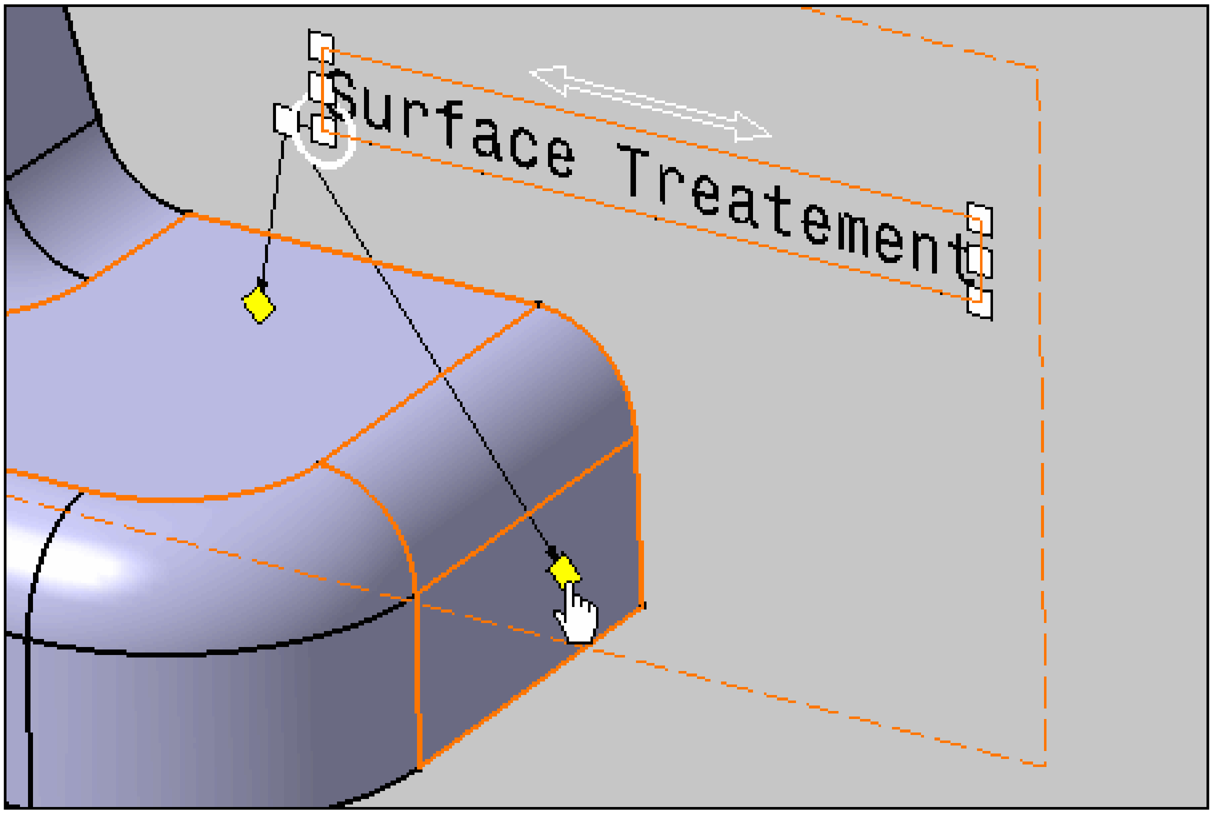


Then new leader appears.

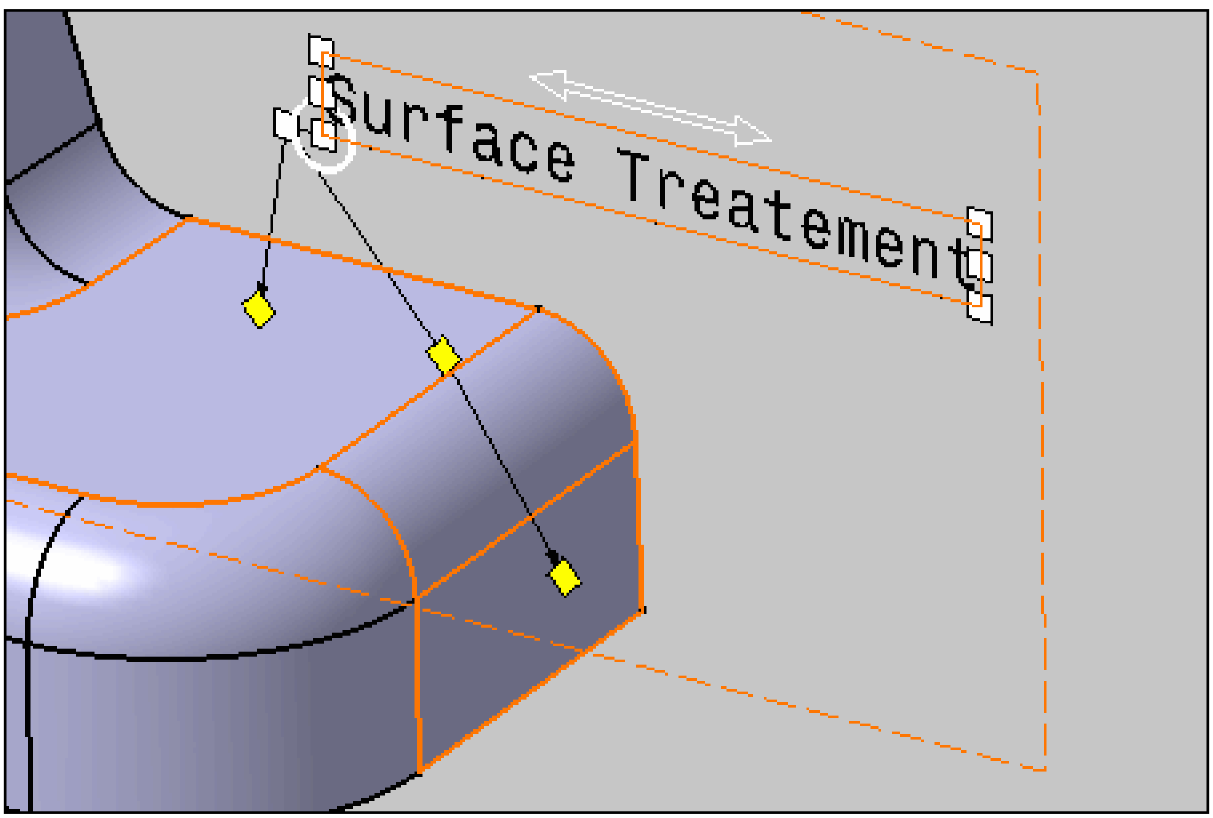


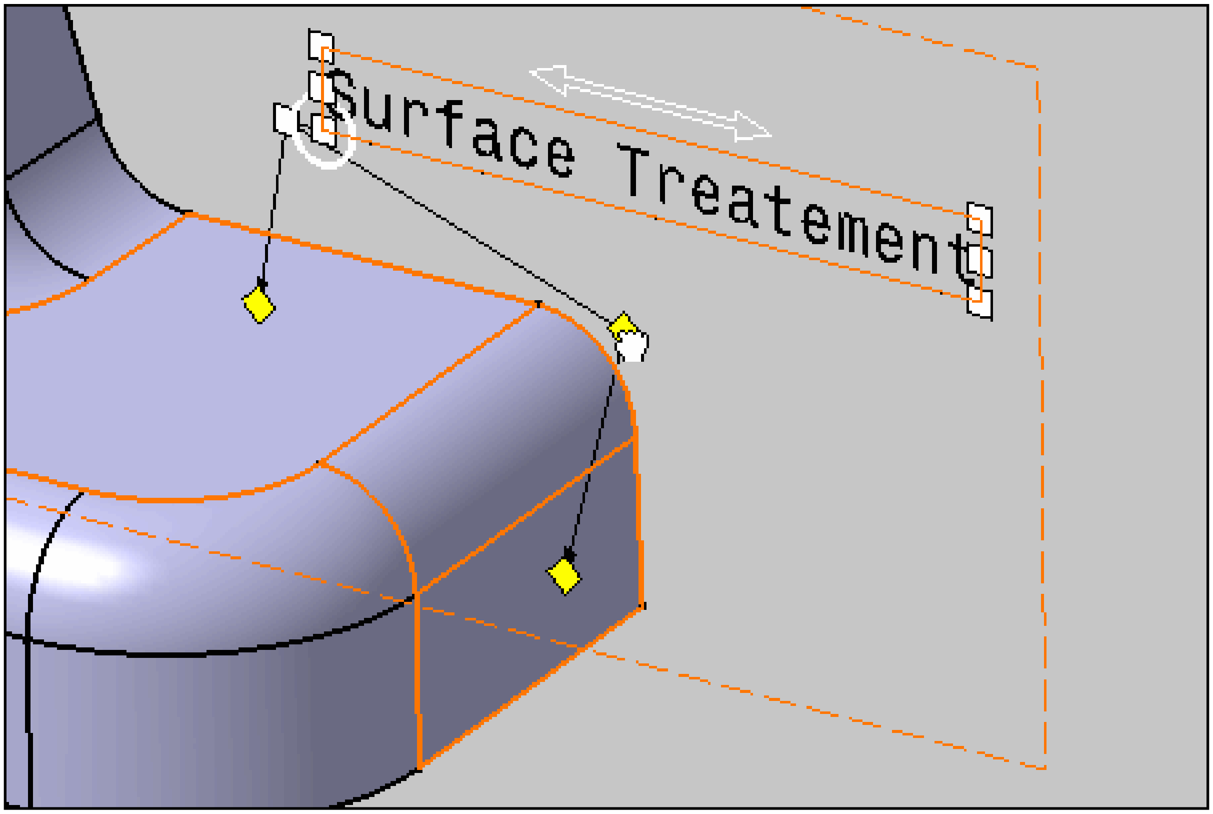
3. If needed, position the leader at the desired location by dragging it.

4. To add a breakpoint, select the manipulator at the extremity of the arrow end and select the **Add a Breakpoint** from the contextual command.



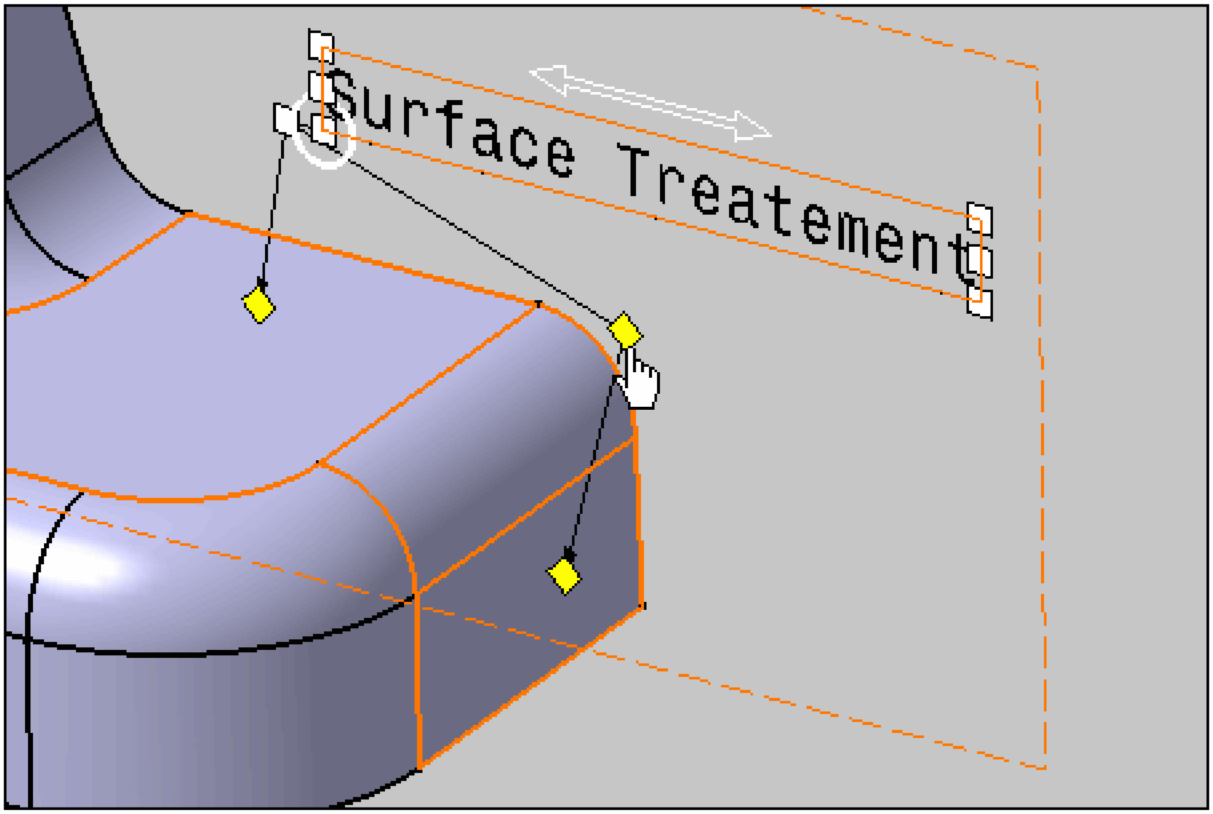
The breakpoint appears as yellow diamond. You can select it and drag the leader.



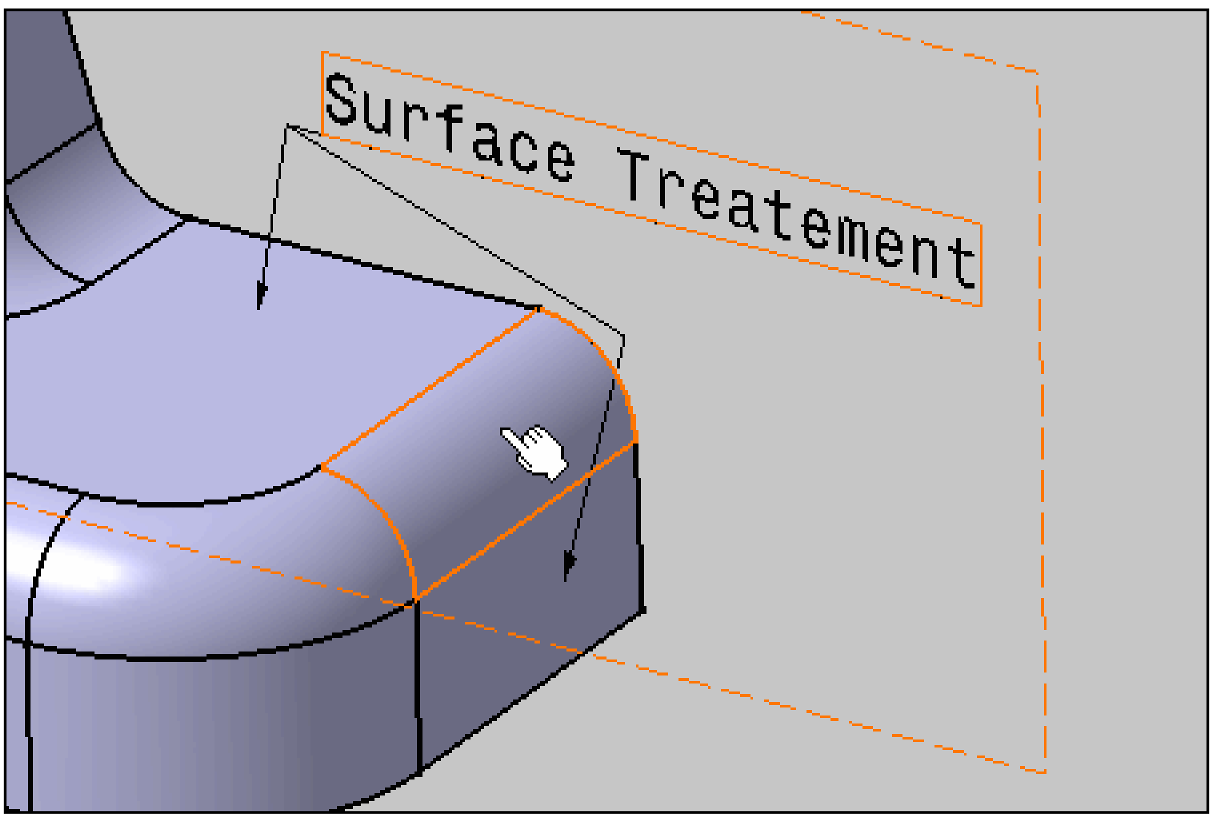


5. To add a leader from the breakpoint, select the breakpoint and select the **Add an Extremity** contextual command.

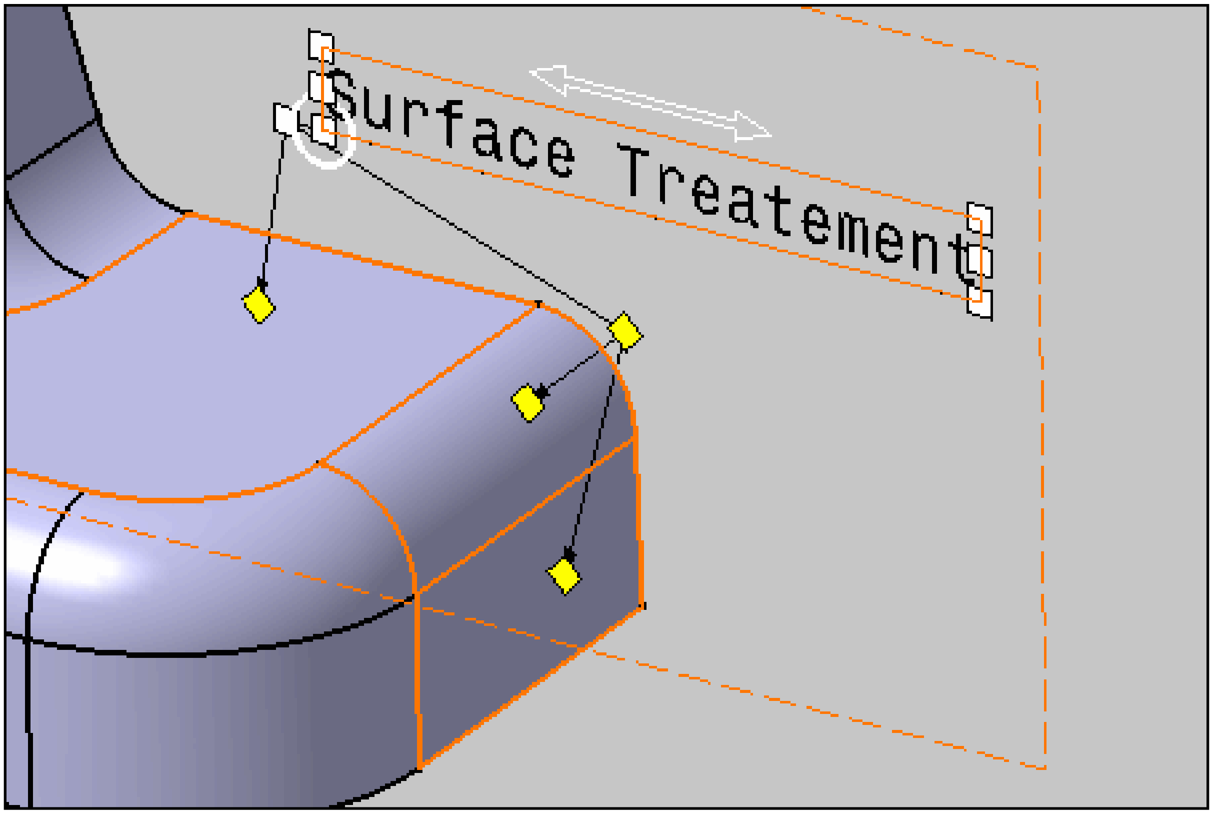
This command is only available for text and flag note annotations.



6. Click the face as shown to begin the leader (arrow end).



The leader appears.


Surface Treatement

Editing the Shape of an End Manipulator

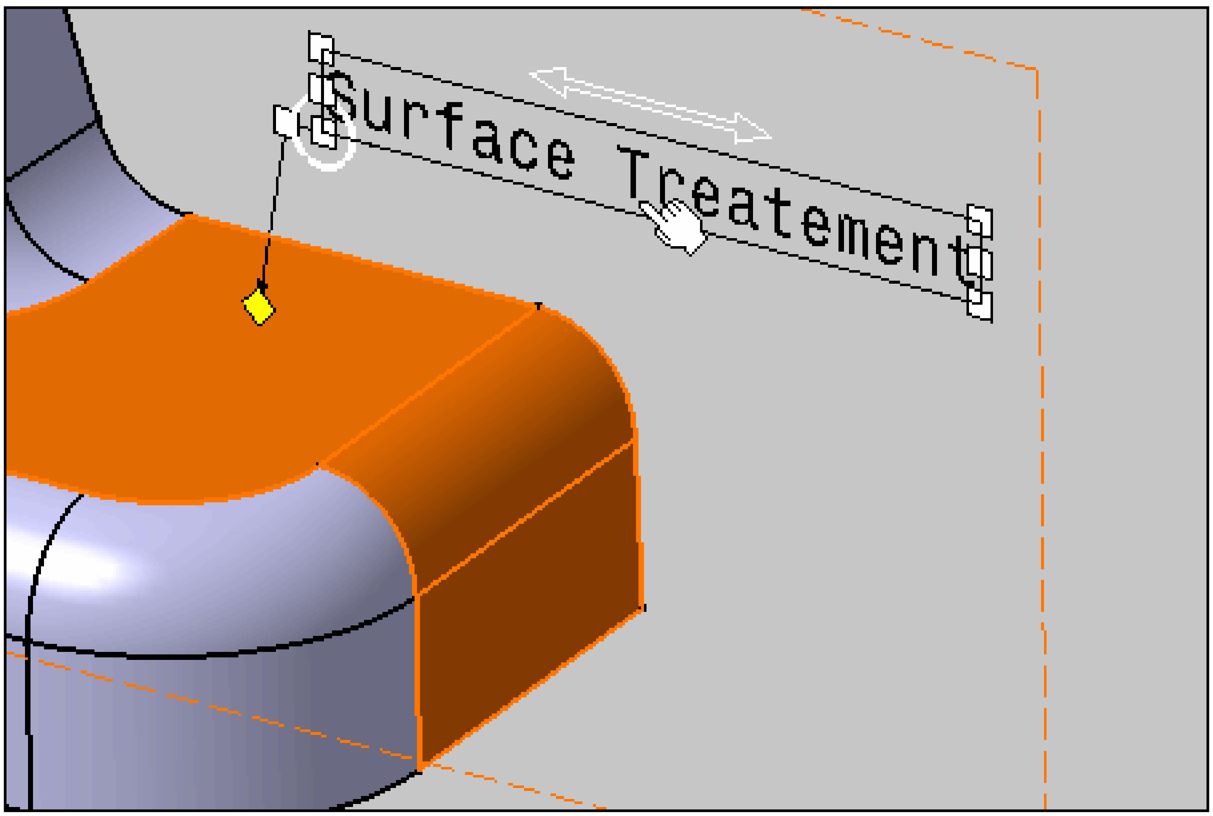
This task shows you how to edit the shape of an end manipulator of an annotation.

You can edit the shape of end manipulators only.

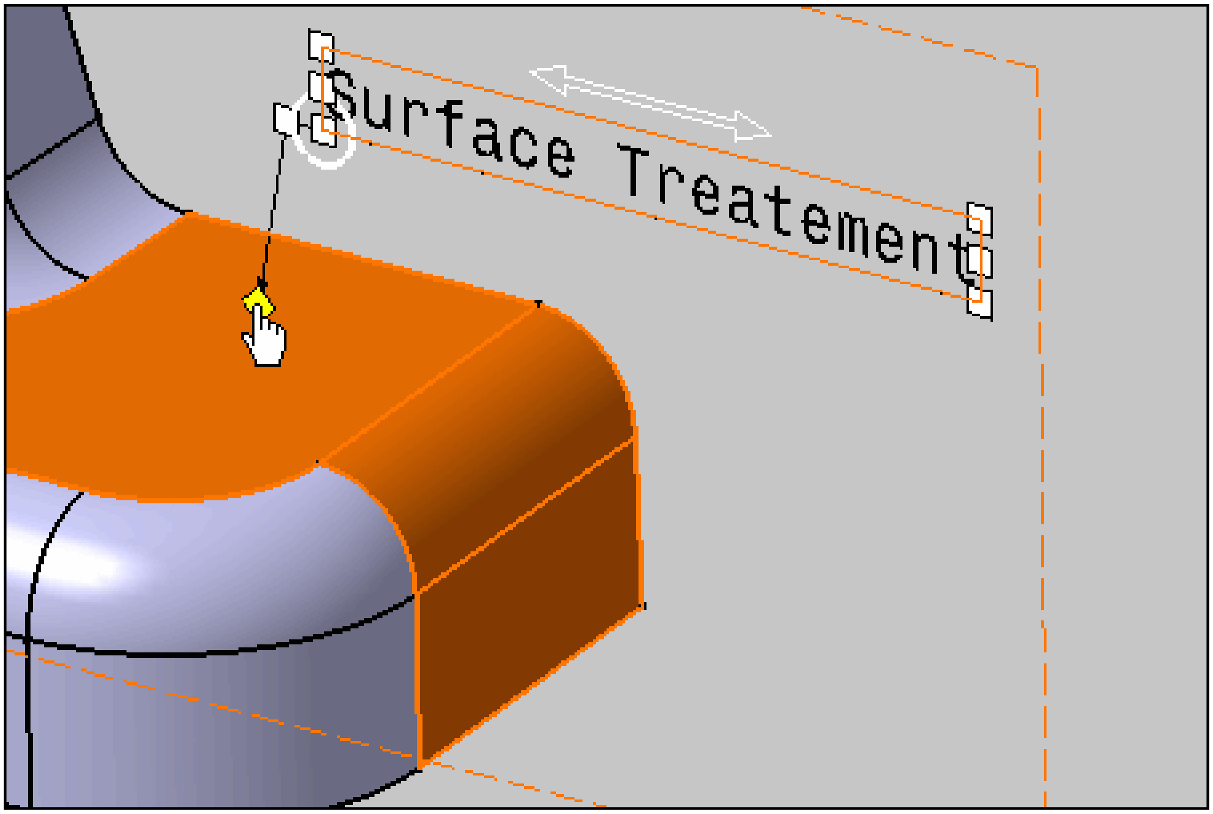
Open the Annotations_Part_04.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Click the annotation text.

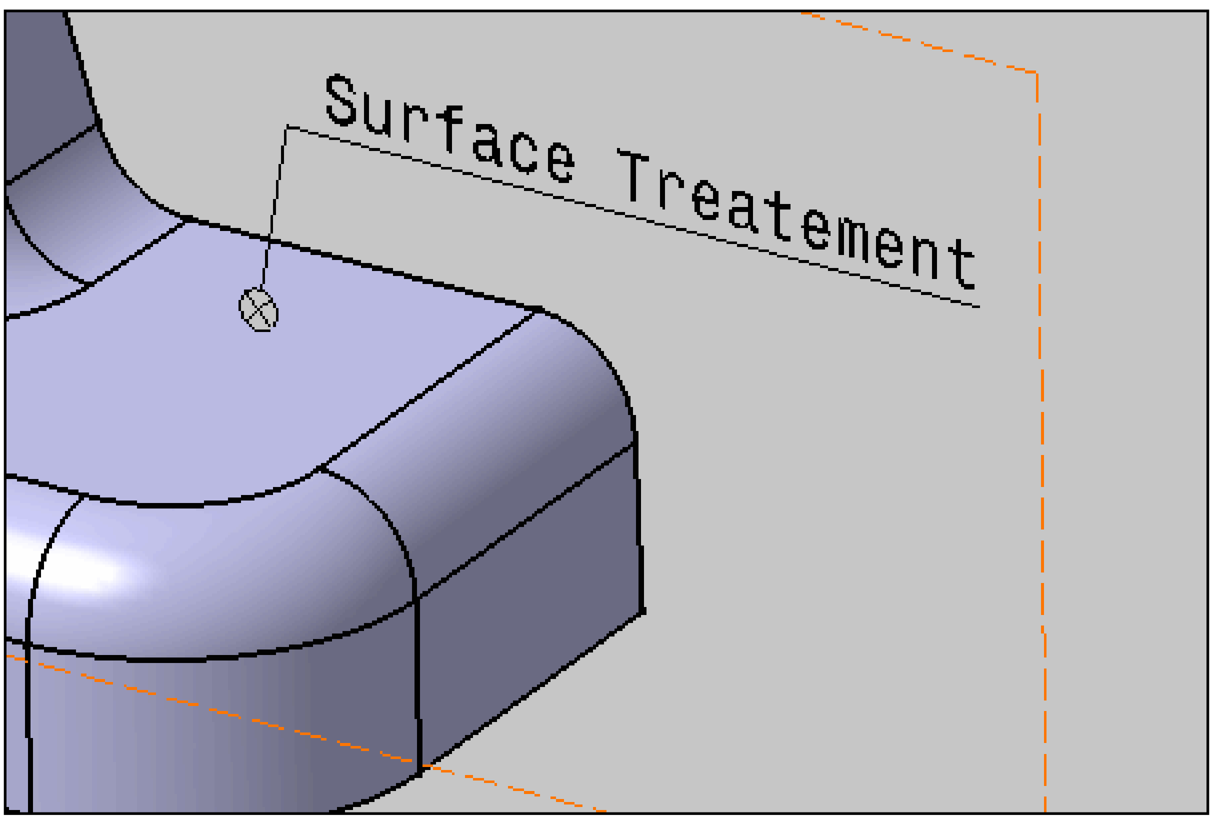


2. Right-click the end manipulator and select **Symbol Shape** from the contextual menu.



3. For instance, select the **Crossed Circle** shape and un-select the annotation.

You obtain this result.



Moving the End Manipulator of a Leader

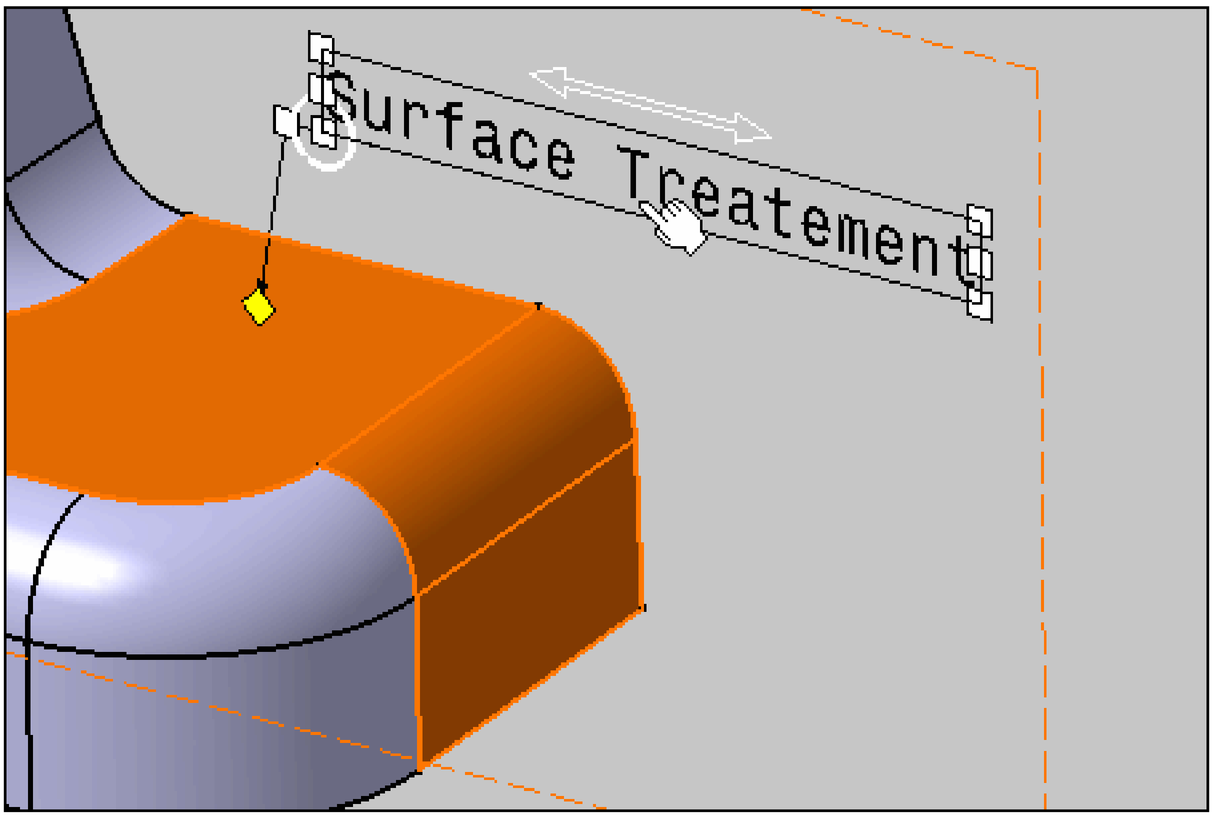
This task shows you how to move the end manipulator of a leader.

You can move a end manipulator of a leader on geometrical elements associated with the annotation only, a yellow line plots the route on them where the end manipulator is moveable.

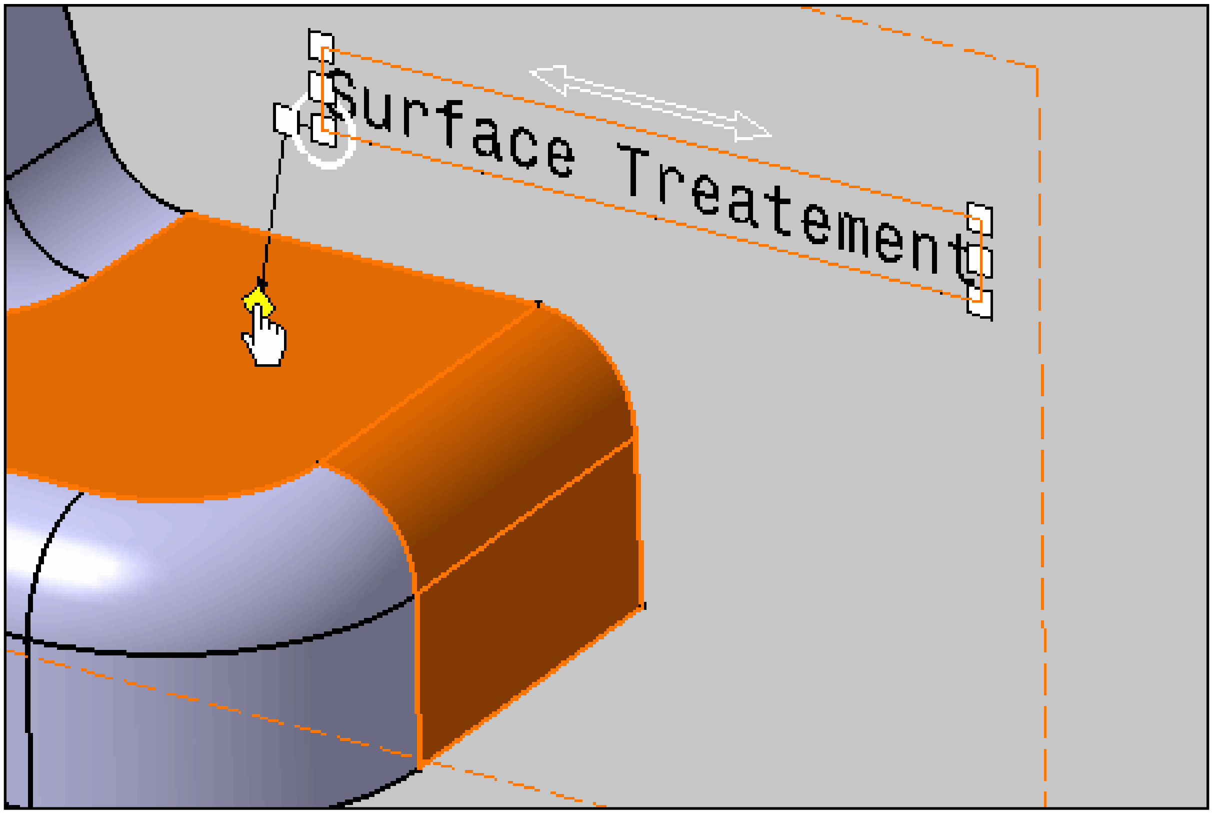
Open the Annotations_Part_04.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

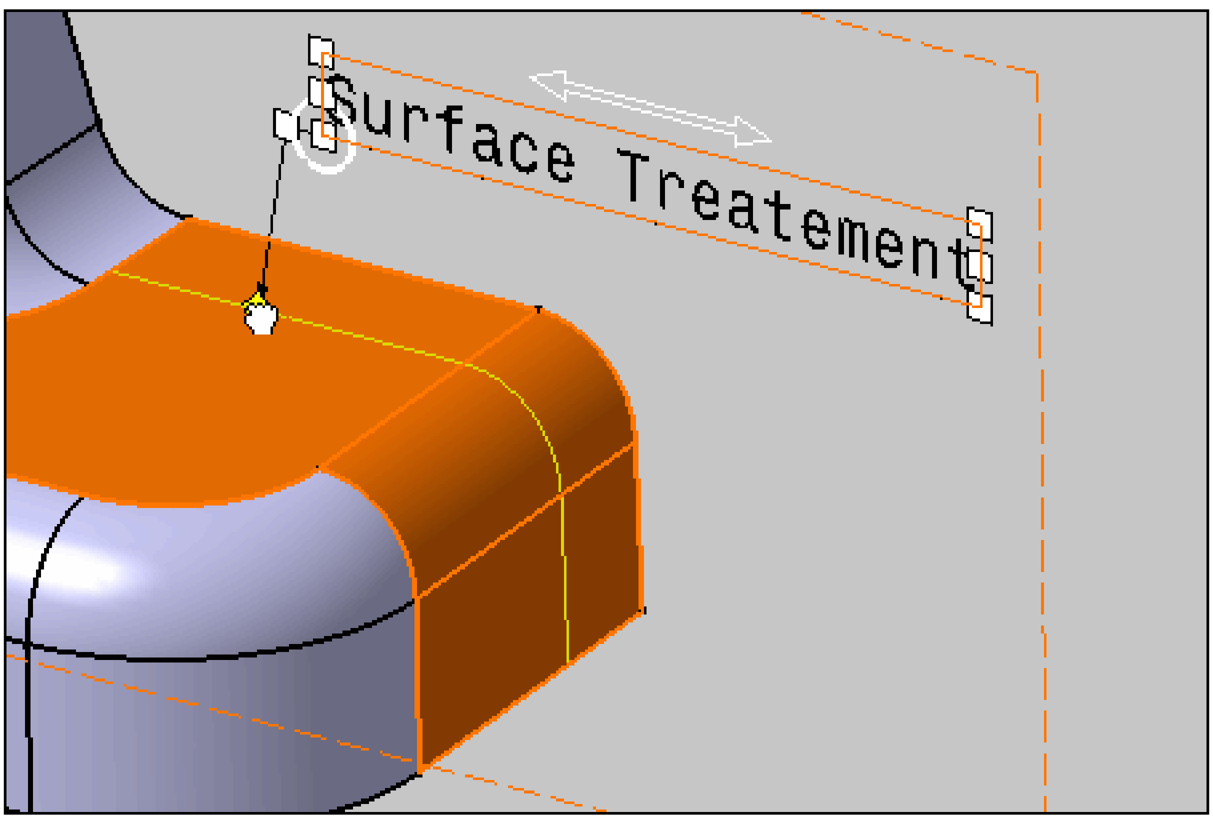
1. Click the annotation text.



2. Select the end manipulator to be moved.

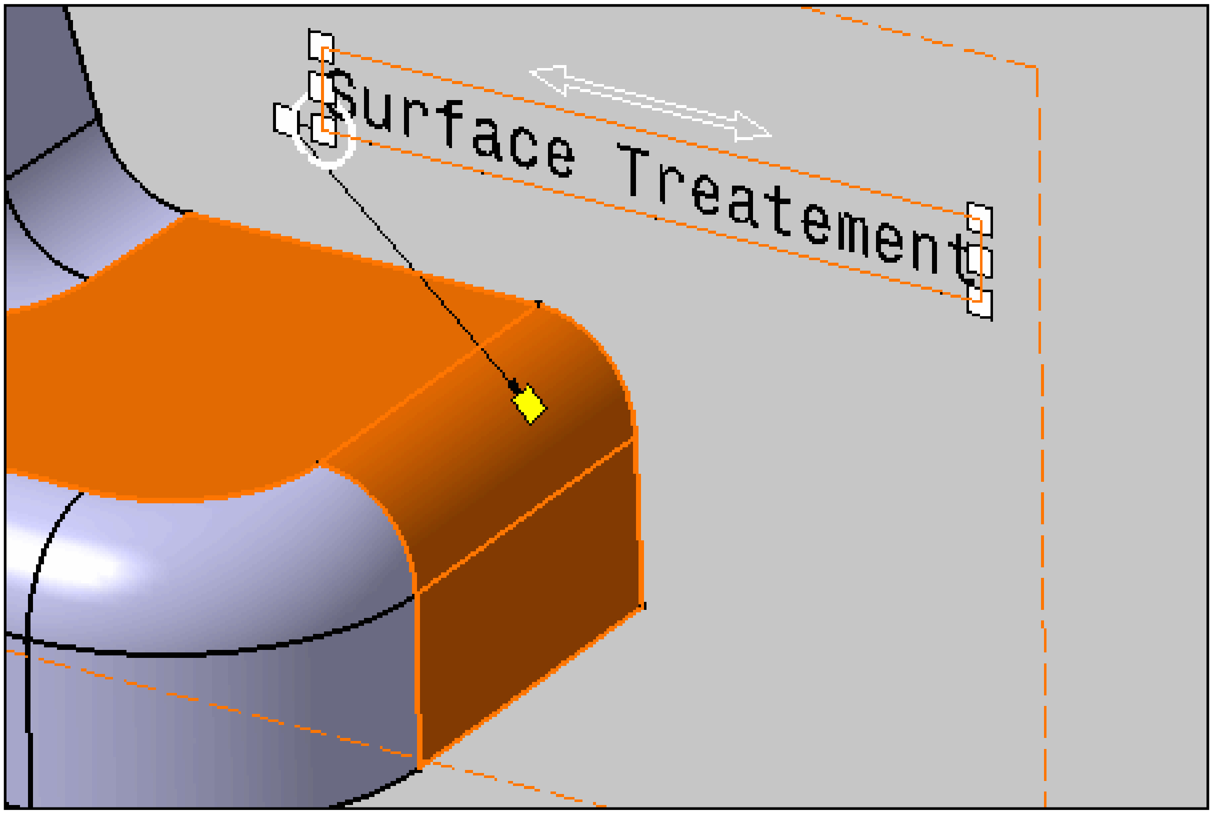


3. Drag the end manipulator along the yellow line.



4. Release the end manipulator.

You obtain this result.



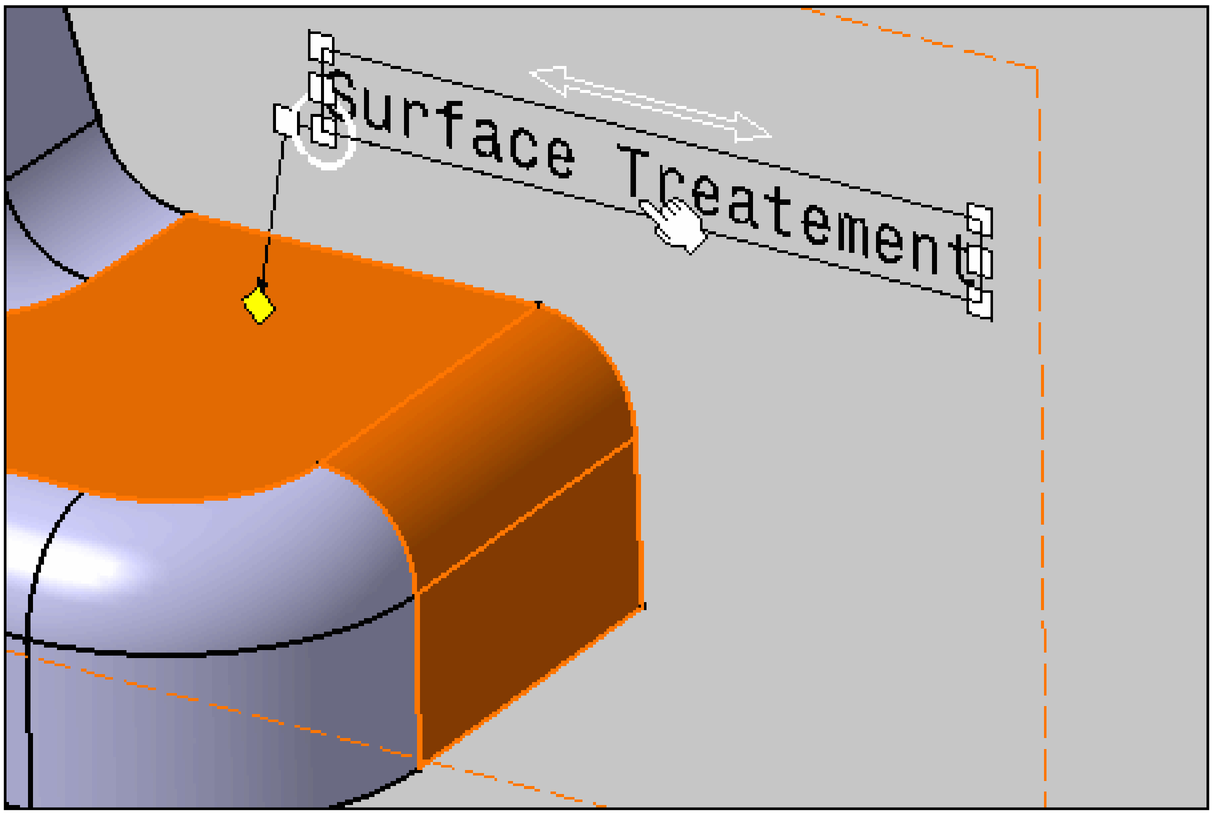
Adding the All Around Symbol

This task shows you how to add the All Around symbol to an annotation.

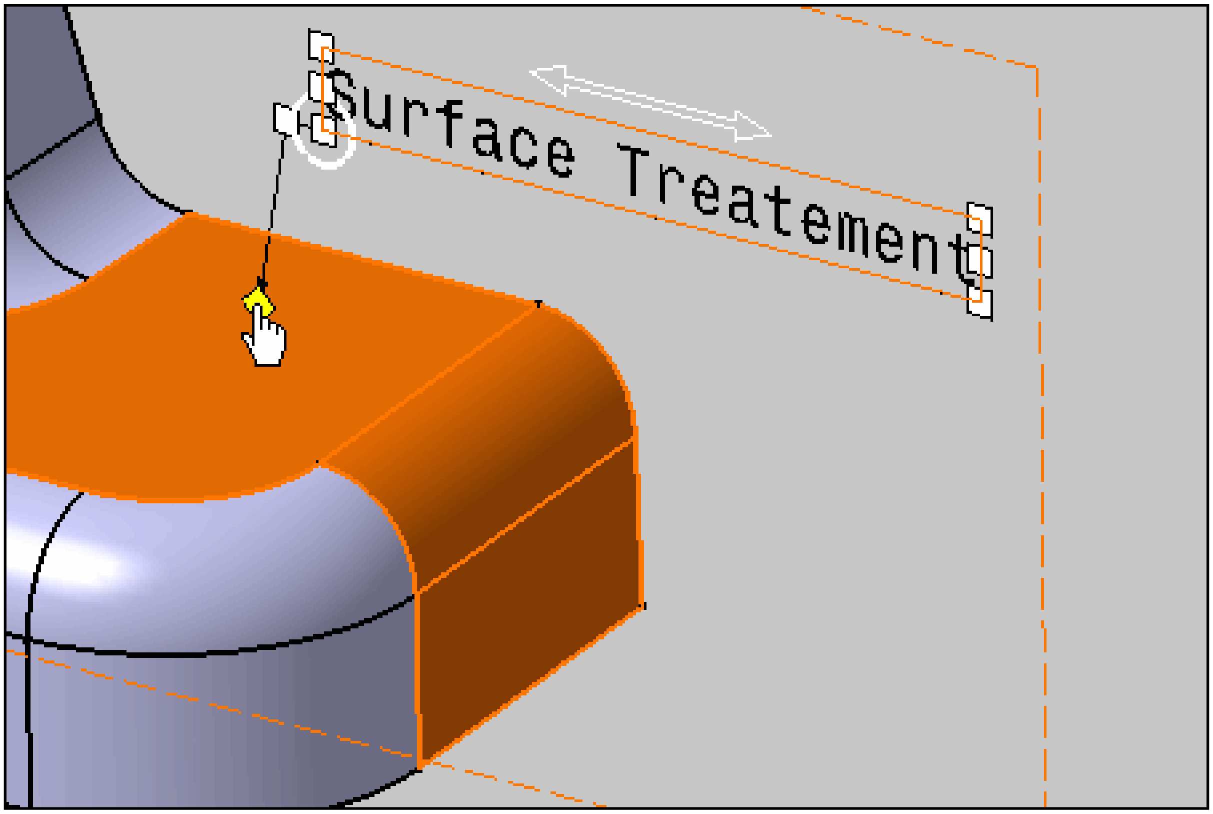
Open the Annotations_Part_04.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Click the annotation text.

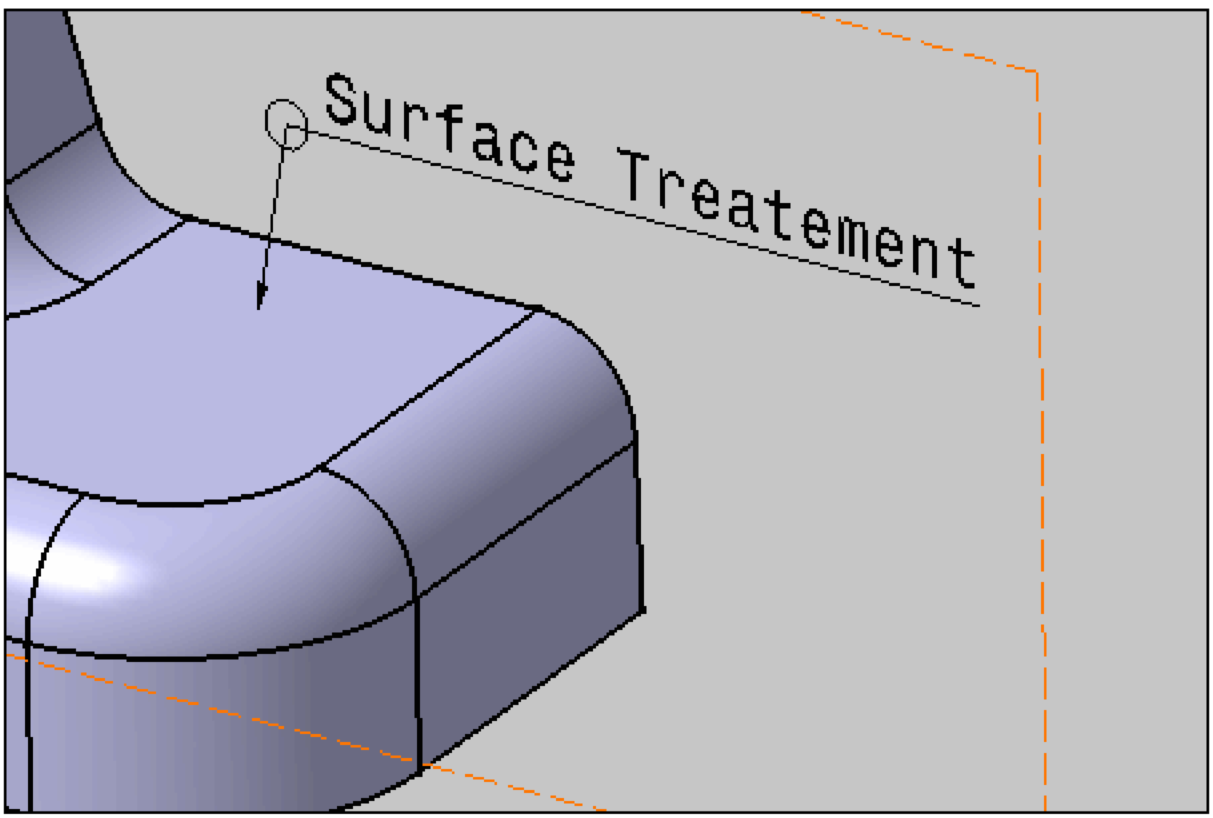


2. Right-click any manipulator and check **All Around** from the contextual menu.



3. Un-select the annotation.

You obtain this result.



Setting Perpendicular a Leader

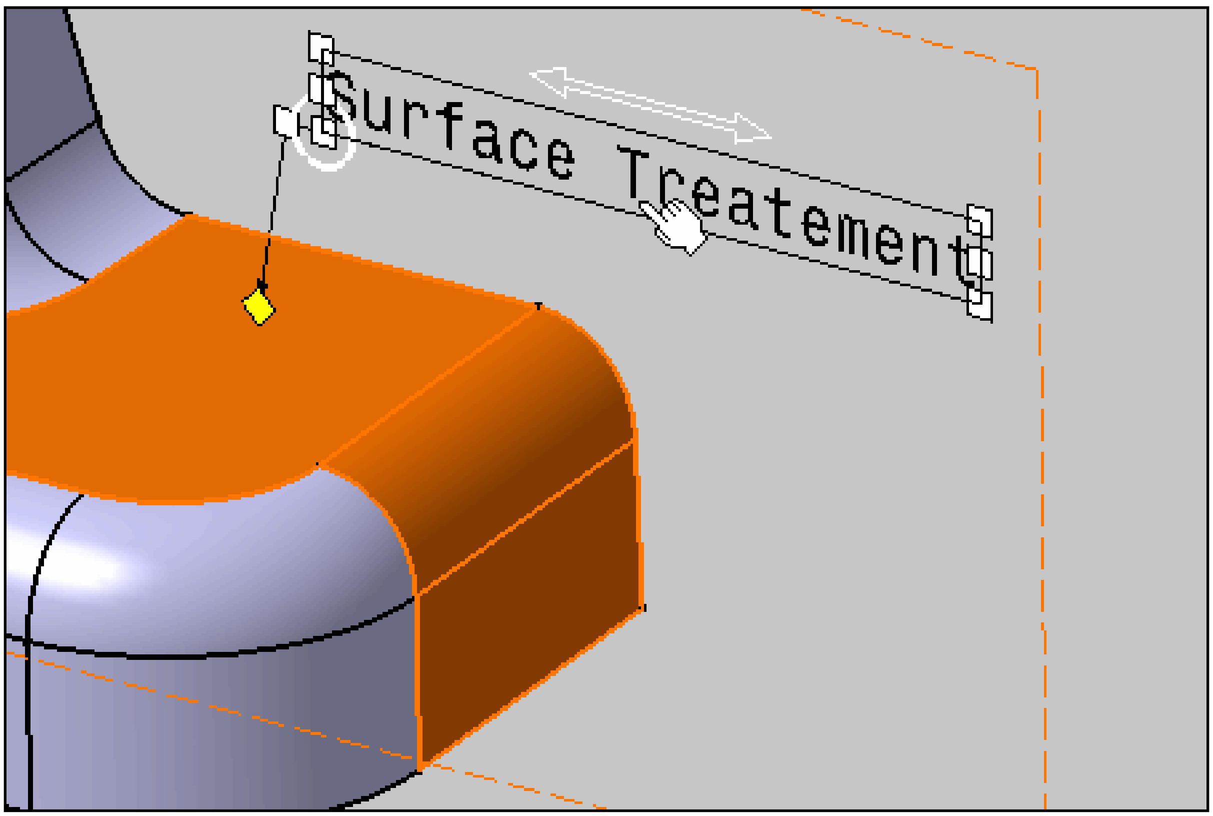
This task shows you how to set perpendicular an annotation leader to its associated geometrical element.

You can set the leader perpendicularity during the creation using the Leader associativity to the geometry option.

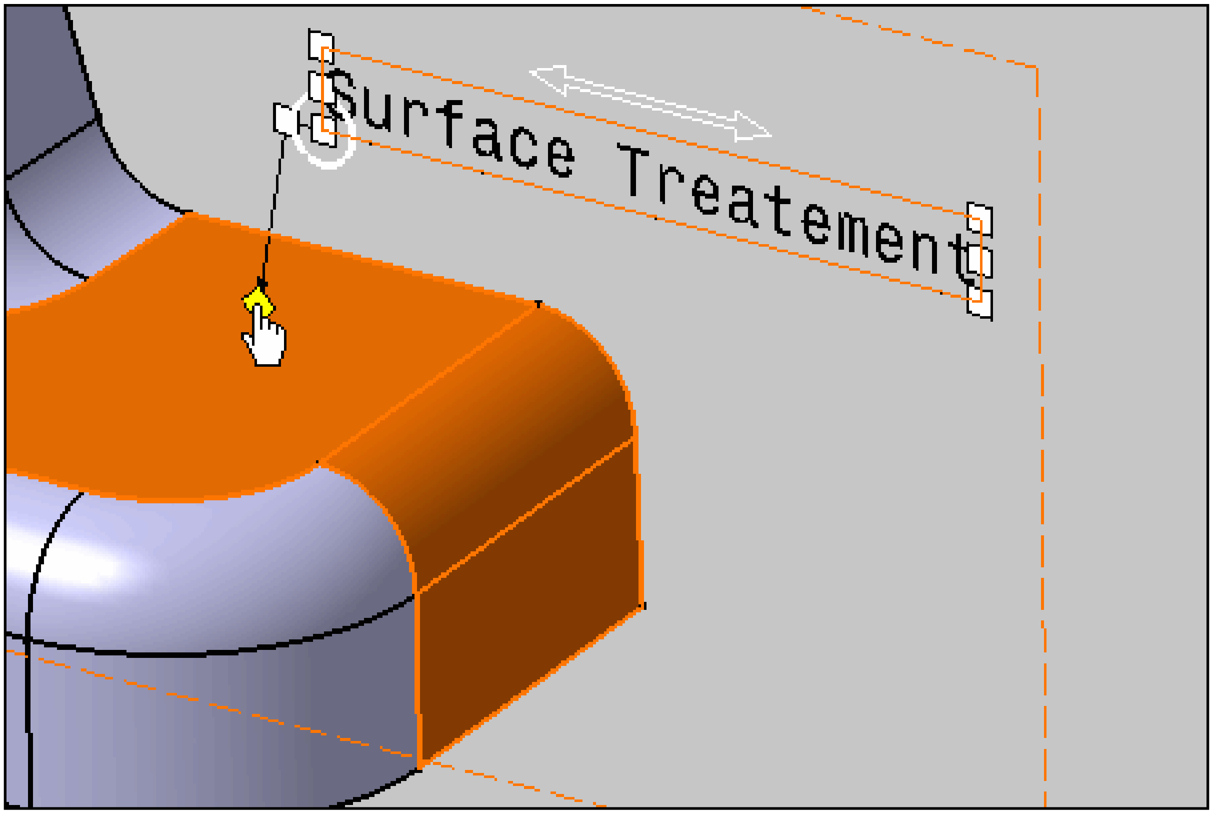
Open the Annotations_Part_04.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

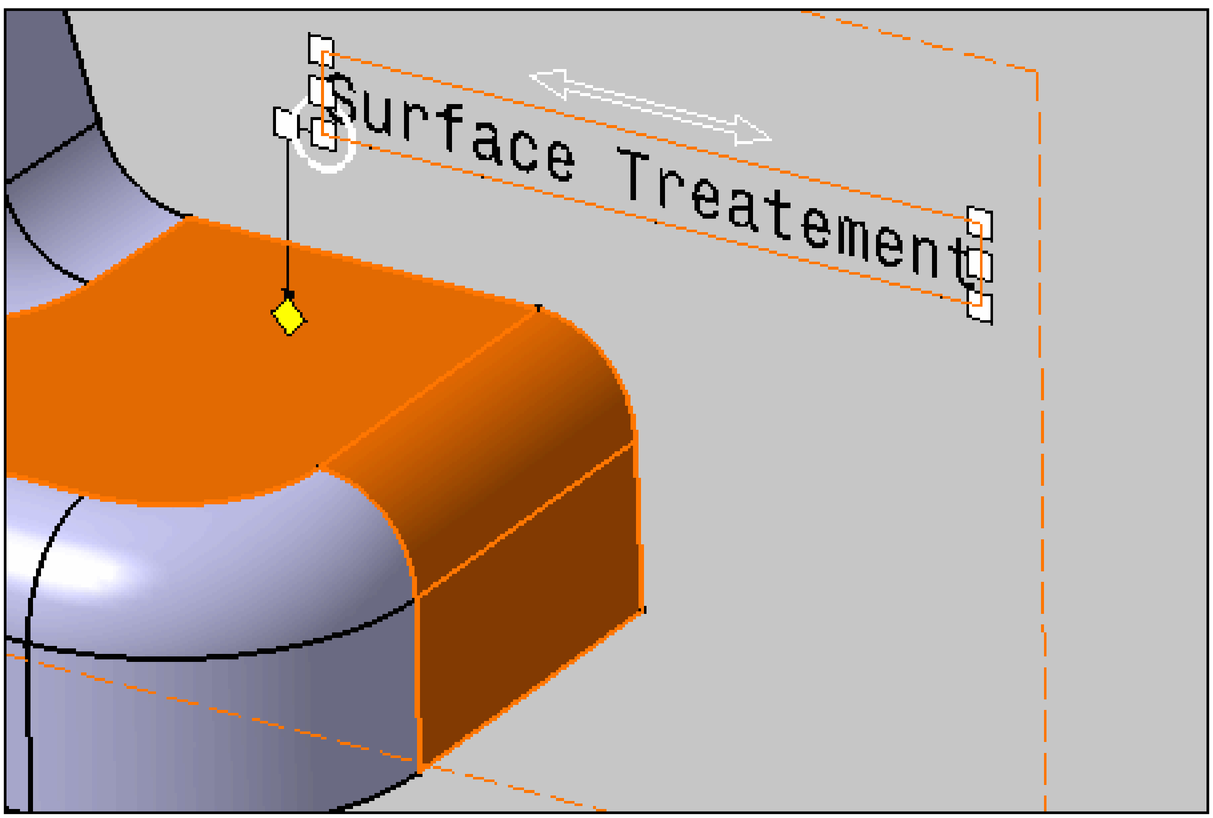
1. Click the annotation text.



2. Right-click the end manipulator and select **Switch to perpendicular leader** from the contextual menu.

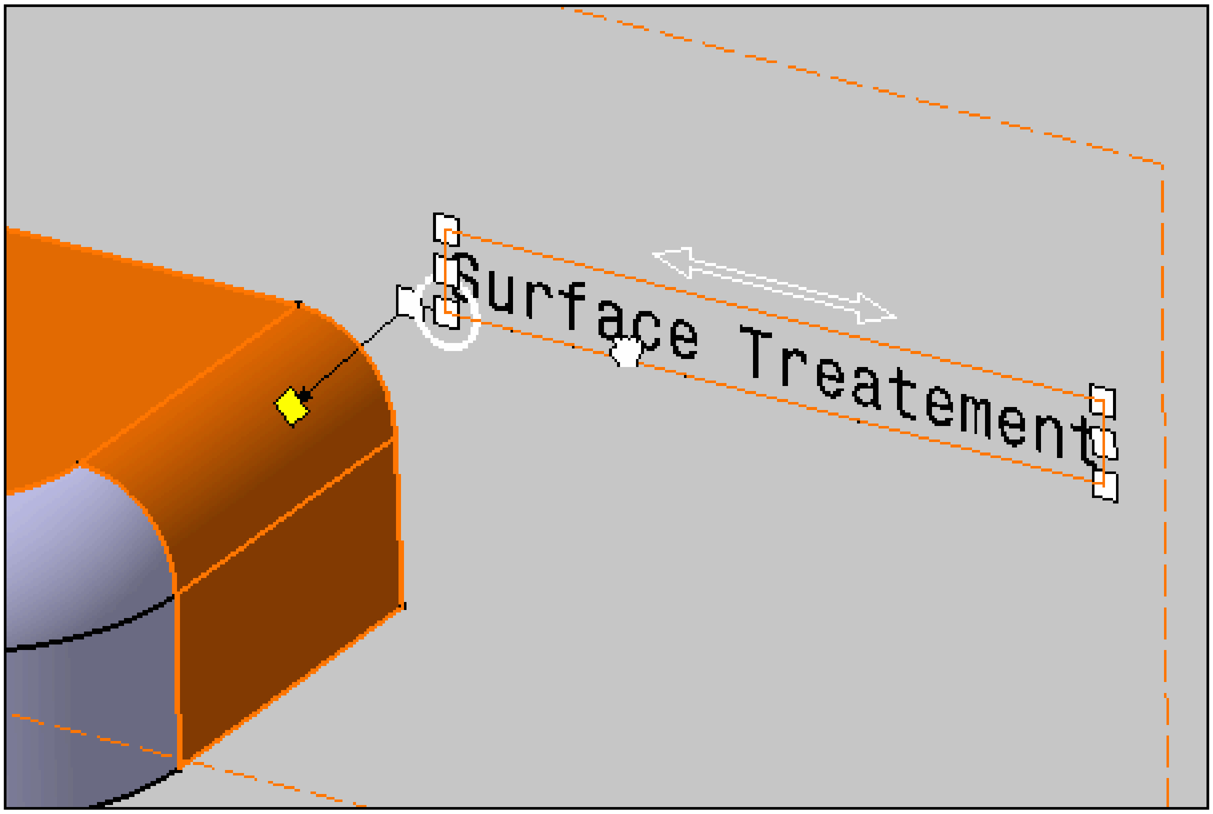


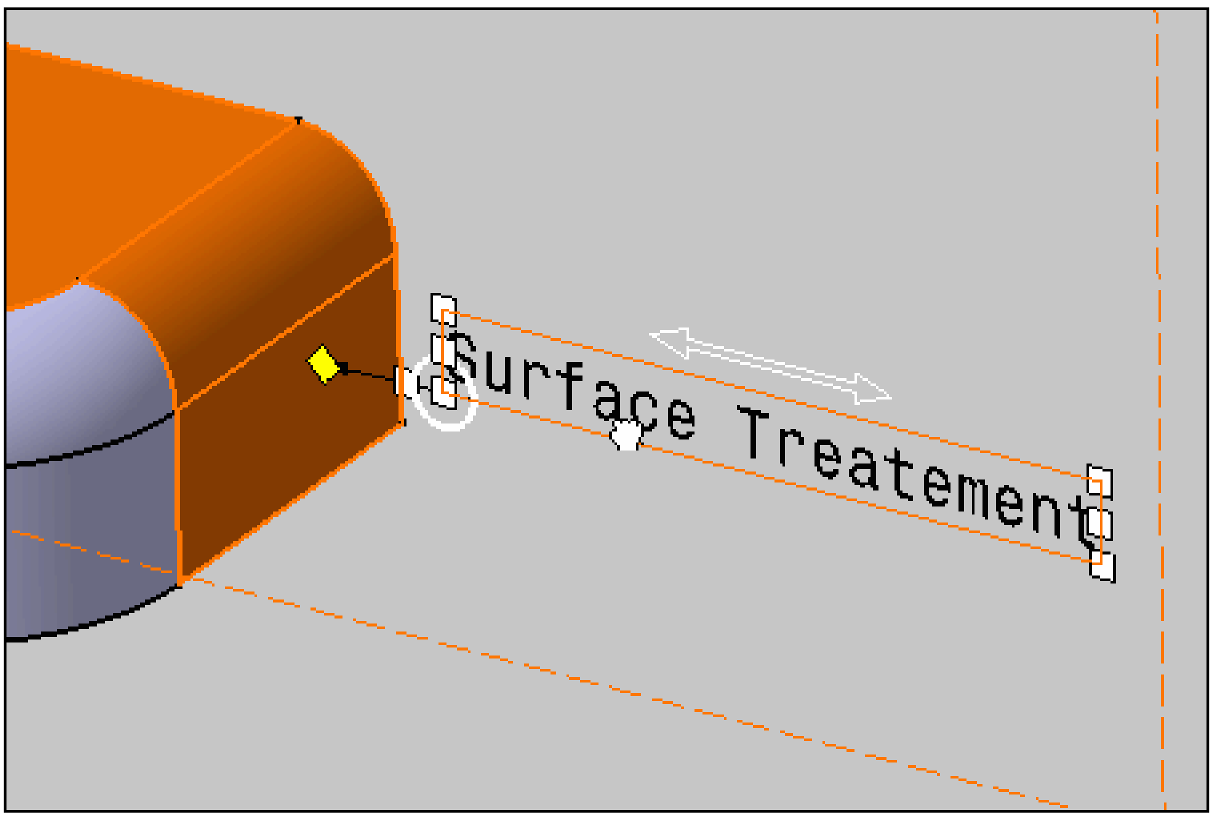
You obtain this result.



3. Drag the annotation.

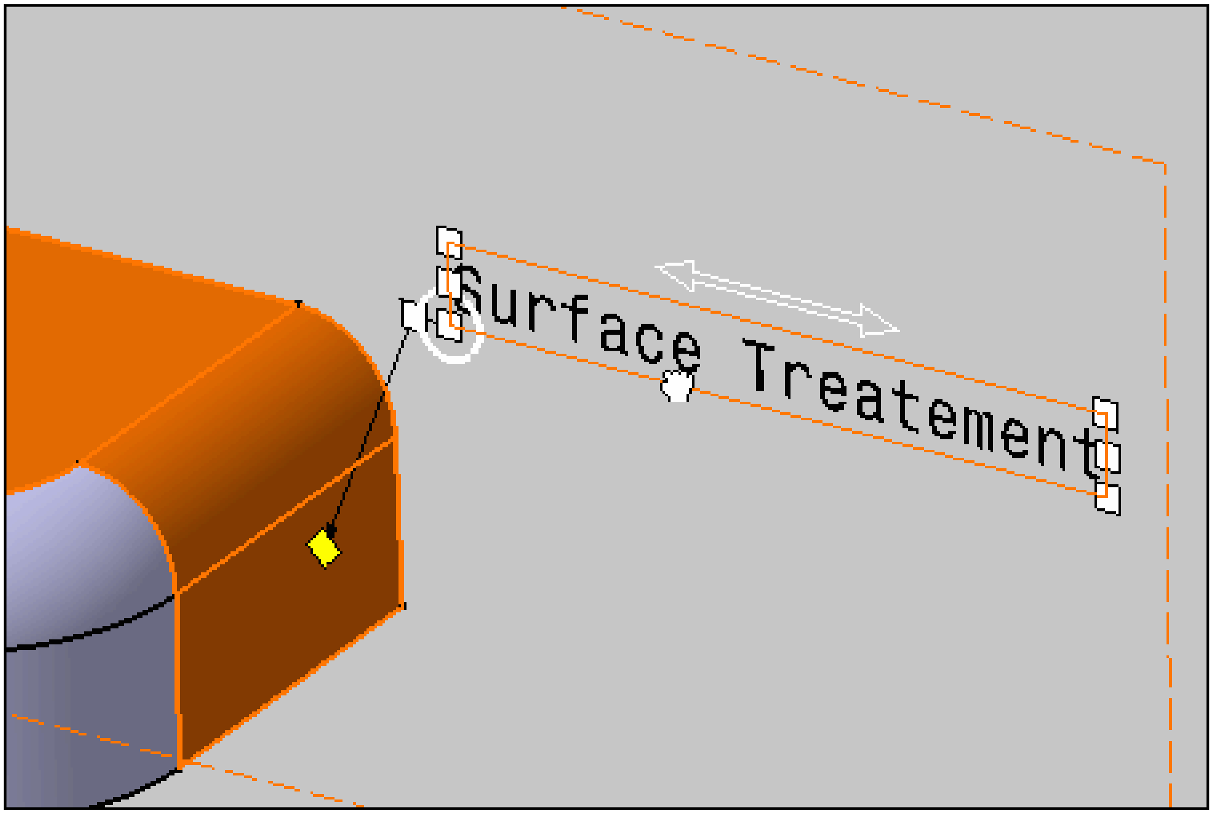
The annotation leader still perpendicular to any associated geometrical element with the annotation.





4. To cancel this behavior, right-click the end manipulator and select **Switch to orientation free leader** from the contextual menu.

The annotation leader orientation is free again.



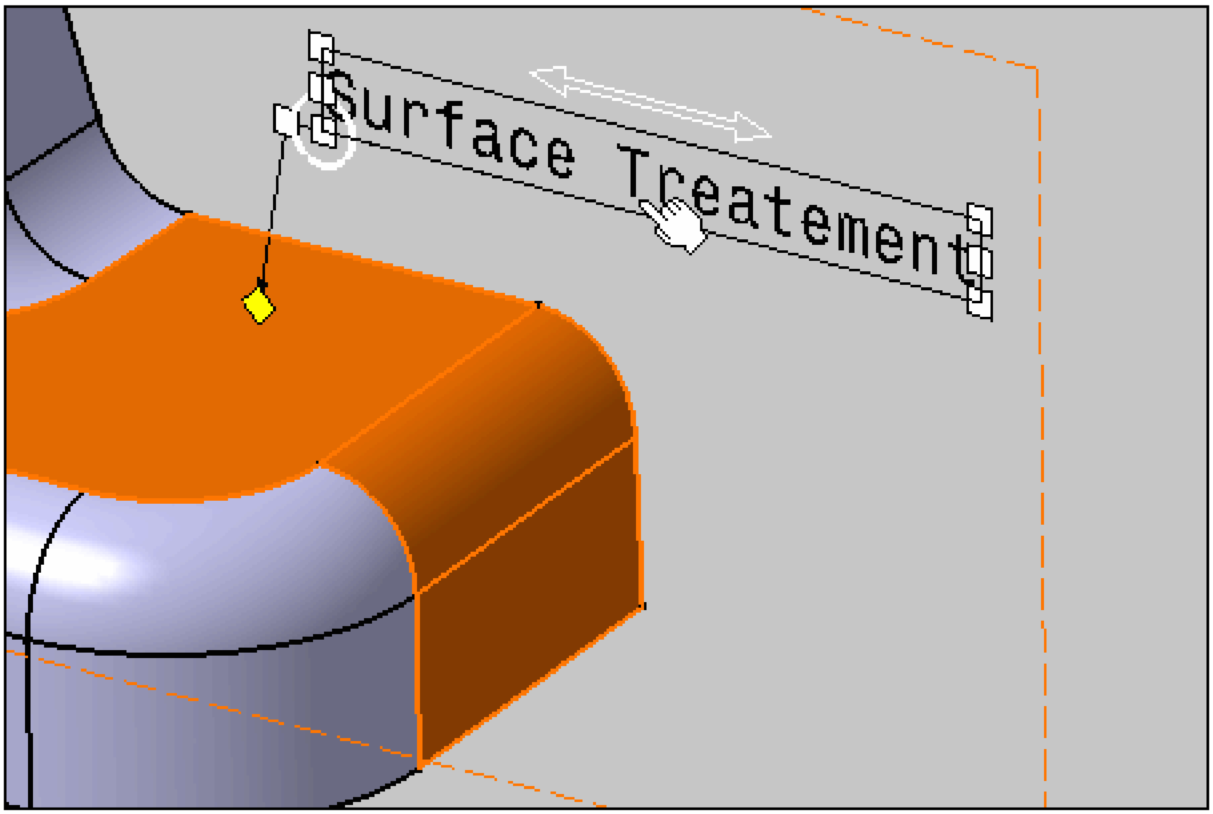
Adding an Interruption Leader

This task shows you how to add an interruption leader.

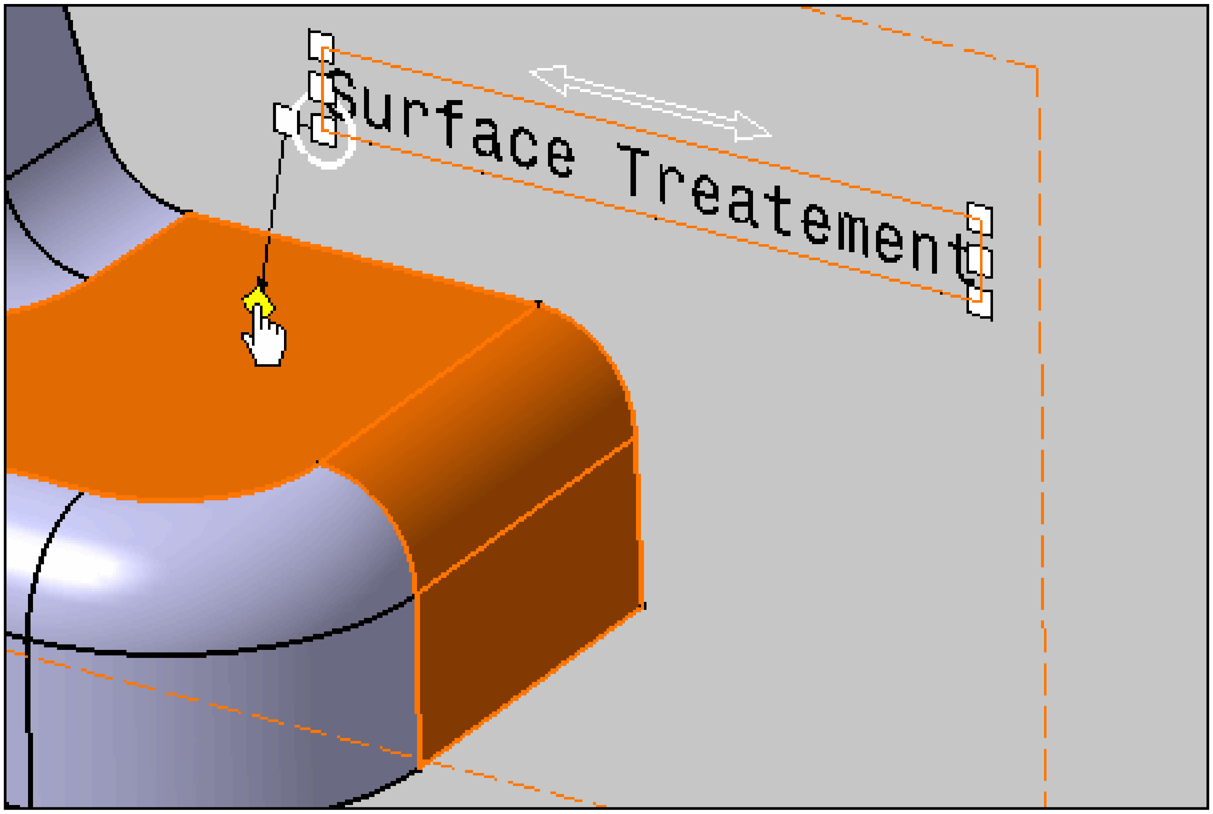
Open the Annotations_Part_04.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

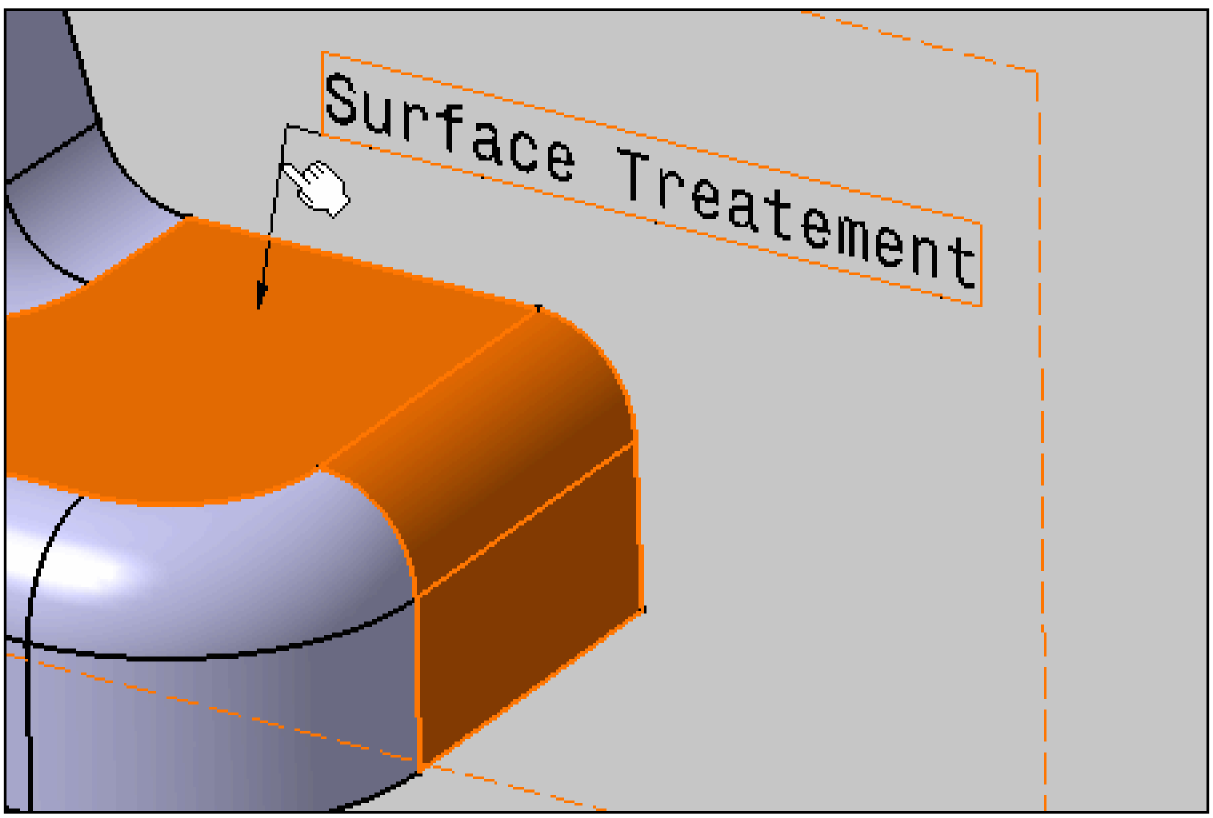
1. Click the annotation text.



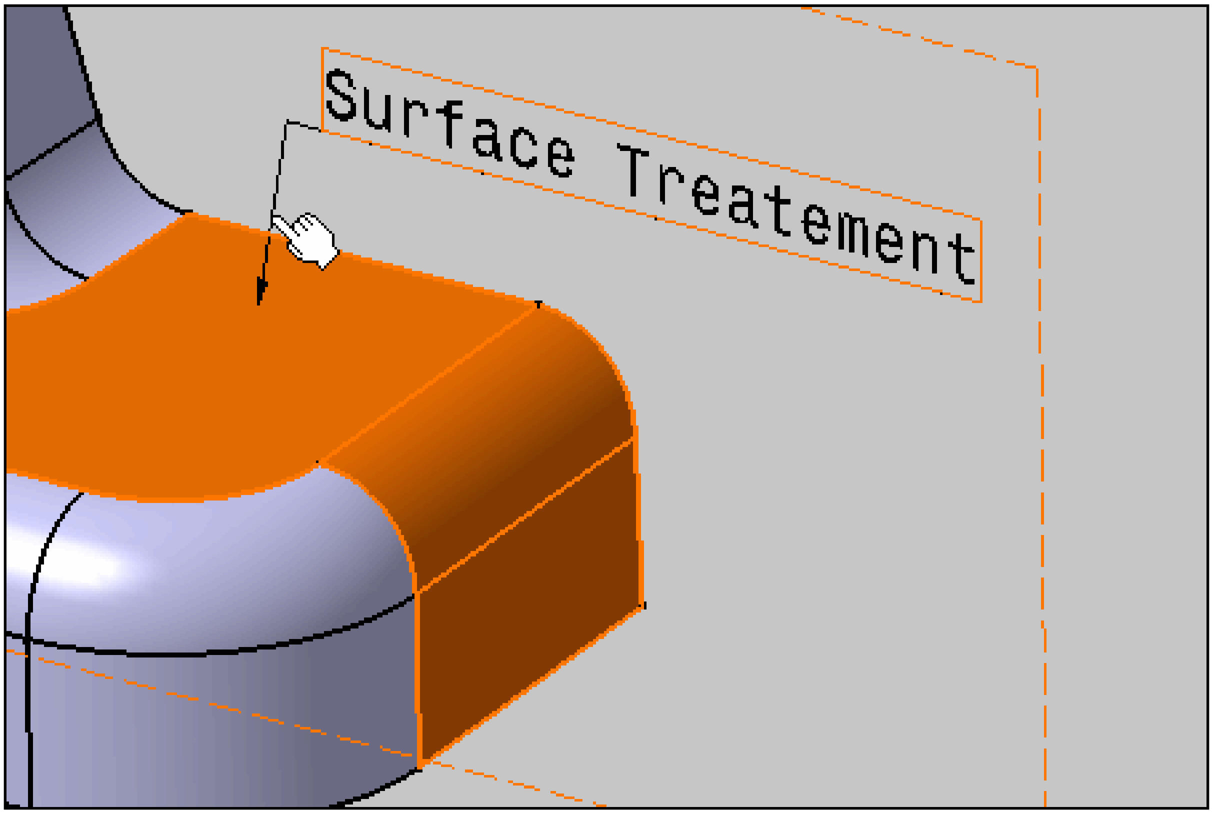
2. Right-click the end manipulator and select **Add an Interruption** from the contextual menu.



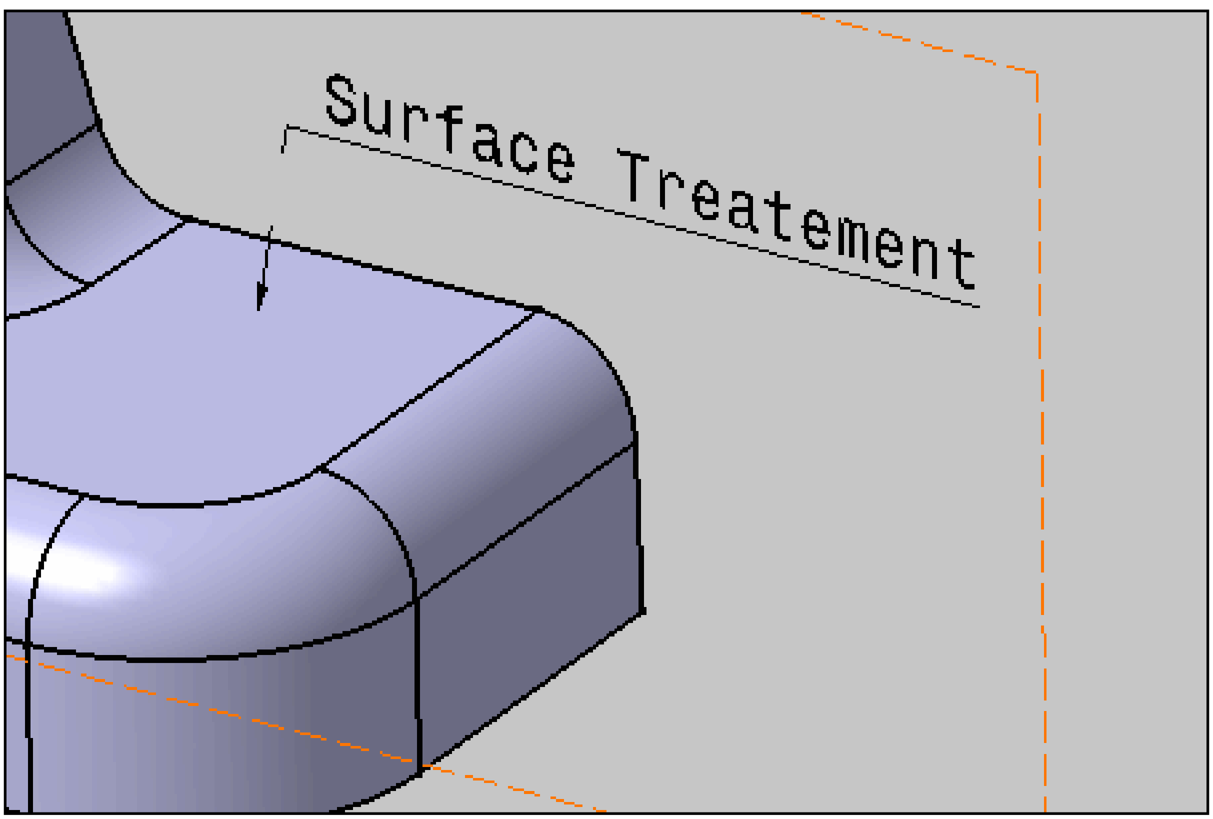
3. Indicate the start point of the interruption on the leader.



4. Indicate the end point of the interruption on the leader.



The interruption is created.



To remove the interruption, right-click the end manipulator and select **Remove Interruptions** from the contextual menu.

Managing Graphical Properties

Set Basic Graphical Properties: select the annotation then the desired options from the **Text Properties** toolbar.

Set Advanced Graphical Properties: select the annotation, the **Edit-> Properties** command and enter the parameters of your choice to edit the font and the text.

Set Graphical Properties as Default: select the annotation and right-click to select the **Set as Default** contextual command

Copy Graphical Properties: multi-select the textual annotations which graphic properties are to be modified, click this icon, and select the text to be used as the graphic reference.

Setting Basic Graphical Properties

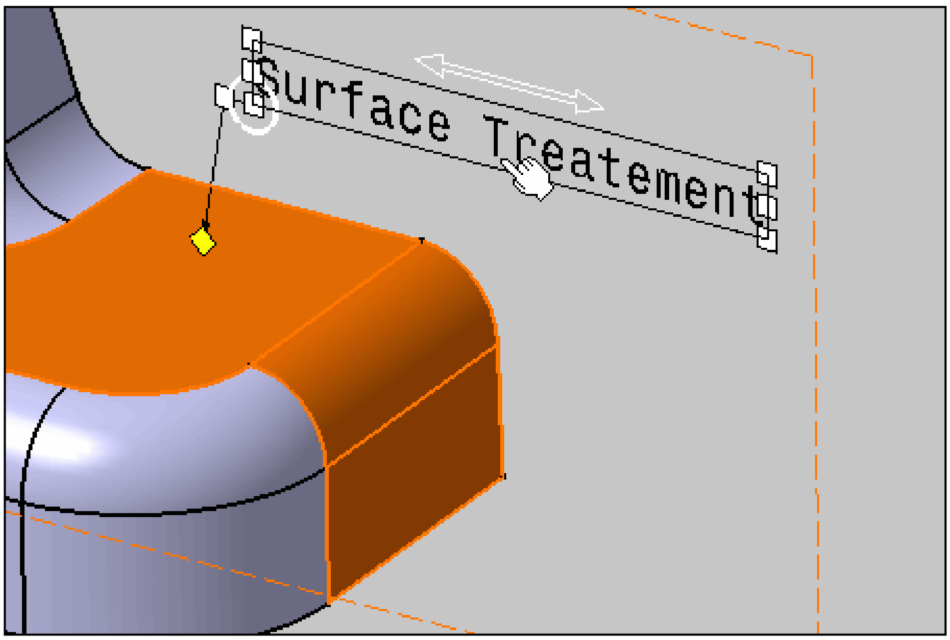
This task shows you how to set graphical properties from the **Text Properties** toolbar for a textual annotation.
Note that the operating mode described here is also valid for datum, datum targets and geometrical tolerances.

Open the Annotations_Part_04.CATPart document:

- Set the **Application Default Properties** or **User Default Properties** option in the **Style** toolbar if needed.

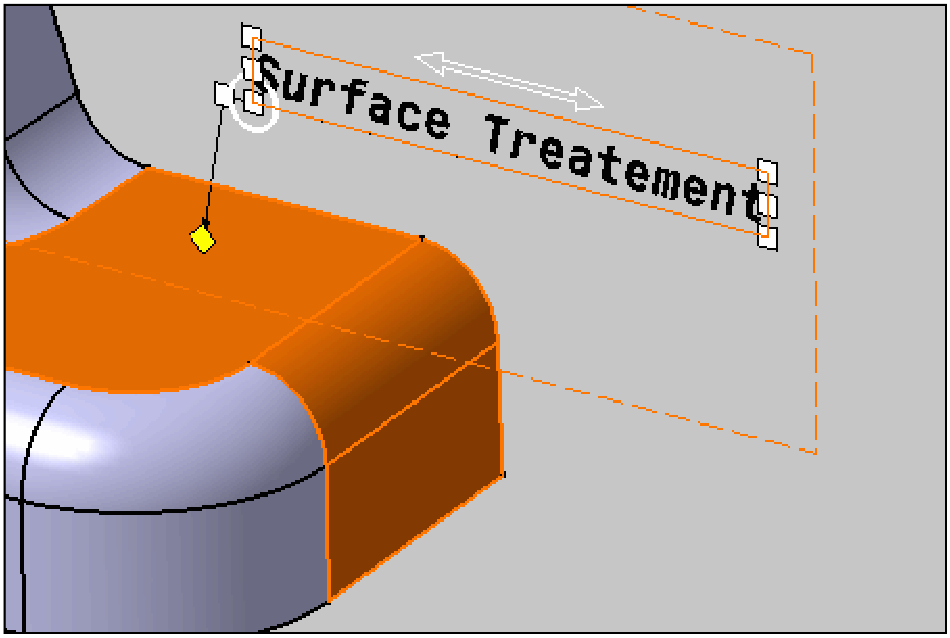
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Select the annotation text.



2. Select the **Bold** and **Italic** options from the **Text Properties** toolbar.

The text is modified accordingly.



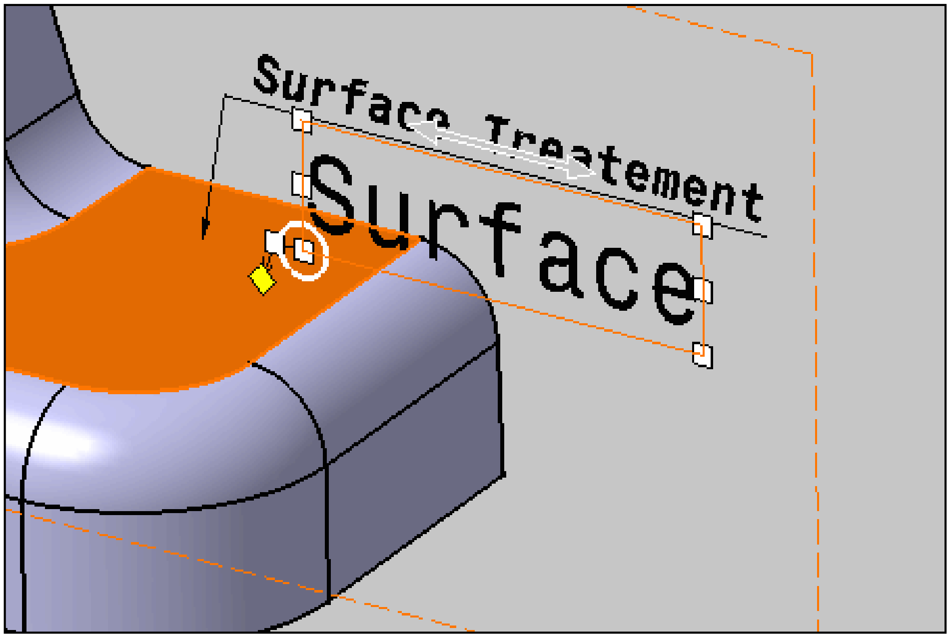
3. De-select the text.

The **Bold** and **Italic** options are de-selected too.

4. Set the **Font Size** to 7 mm from the **Text Properties** toolbar.

5. Create a new text and enter the following text: Surface (See Creating an Annotation Text).

The text is created accordingly.



Annotation texts inherit from pre-selected options in the **Text Properties** toolbar when created with the **Application Default Properties** or **User Default Properties** option selected in the **Style** toolbar.

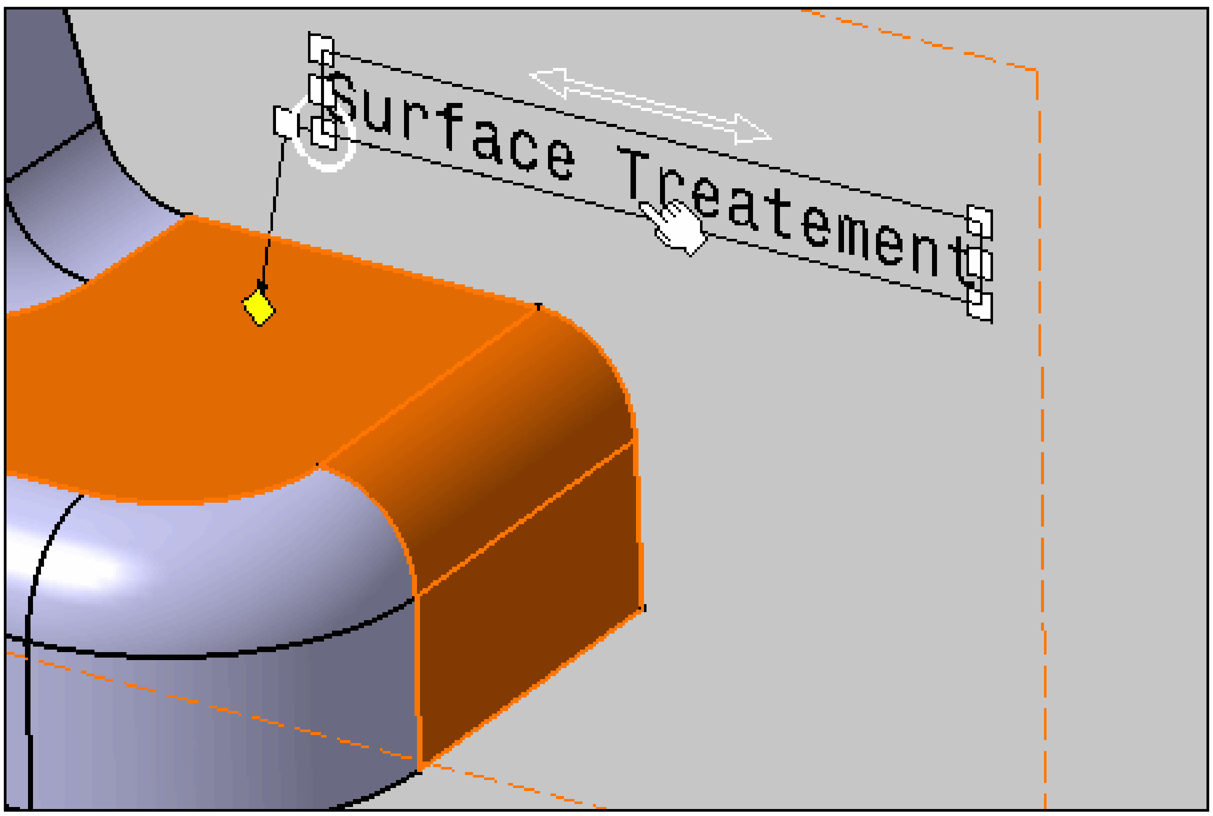
Setting Advanced Graphical Properties

This task shows you how to set graphical properties from the **Properties** dialog box for a textual annotation.
Note that the operating mode described here is valid for datum elements, datum targets and geometrical tolerances too.

Open the Annotations_Part_04.CATPart document:

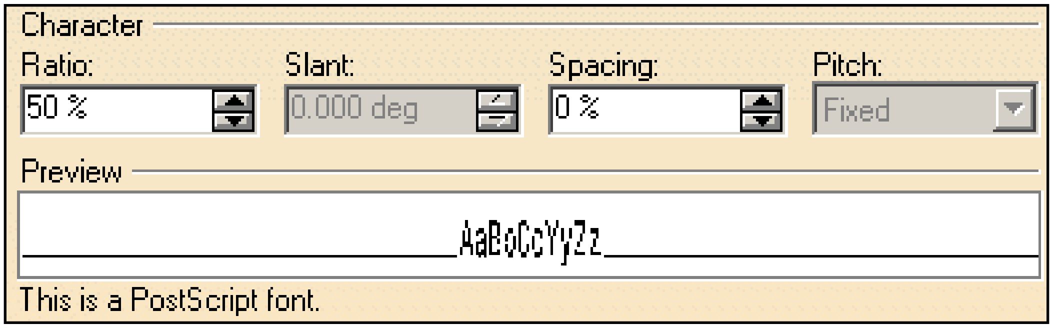
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Select the annotation text.



2. Right click and select the **Properties** contextual menu.

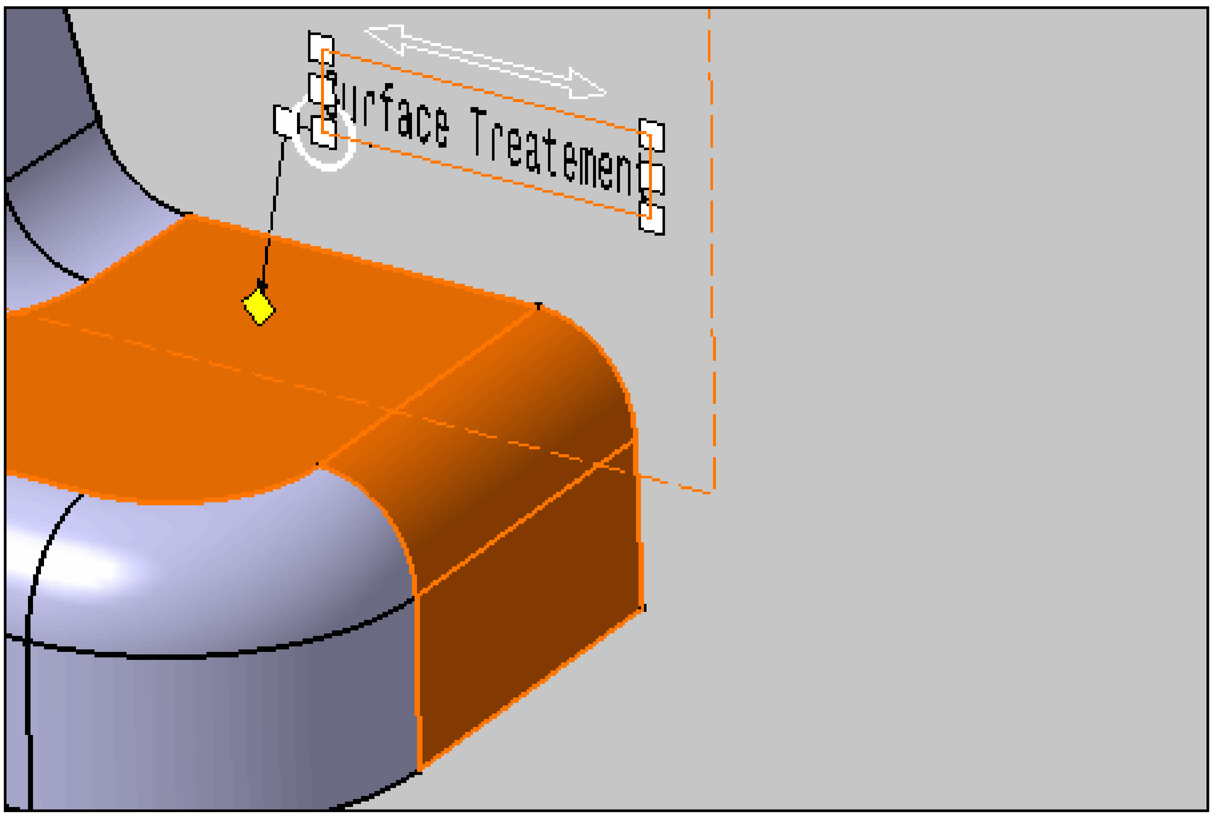
3. Select the **Font** tab and set the **Character Ratio** to 50%.



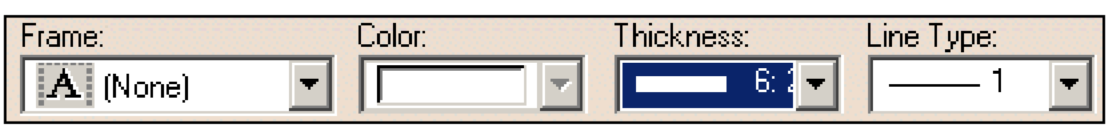
The **Font** tab is dedicated to several options defining the font.
These options are the same as the ones available from the **Text Properties** toolbar, except for the color you can assign.

4. Click **Apply**.

The text is modified accordingly.



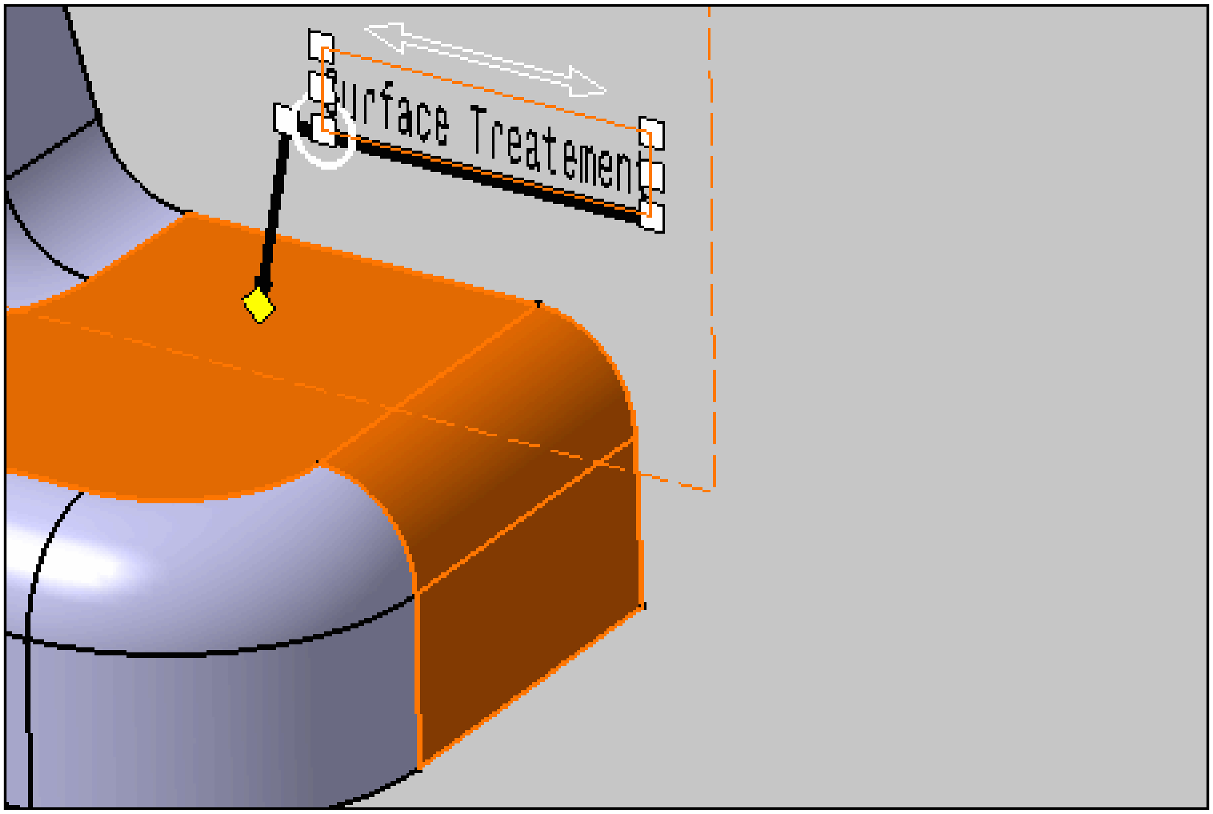
5. Select the **Text** tab and set the **Thickness** option to 2.00 mm to make the leader more visible.



The options available let you edit the position of your text as well as the leader properties but not the arrow, see Editing the Shape of an End Manipulator.

6. Click **OK**.

The text leader is modified accordingly.

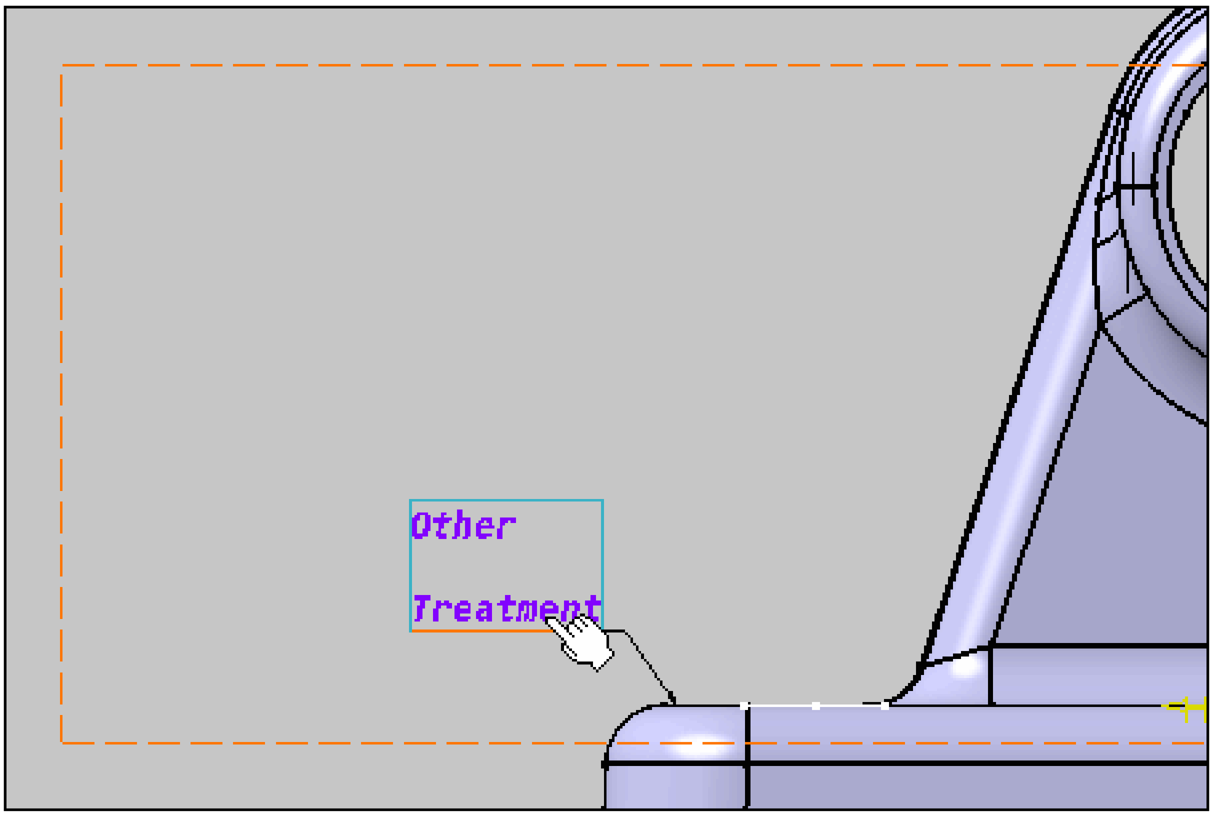


Setting Graphical Properties as Default

This task shows you how to set graphical properties as default properties.
Note that the operating mode described here is also valid for datum, datum targets and geometrical tolerances.

Open the Annotations_Part_04.CATPart document.

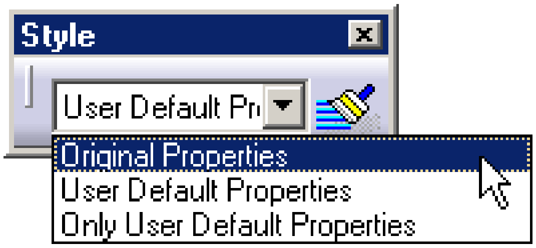
1. Select the text.



2. Right-click and select the **Set as Default** contextual option.

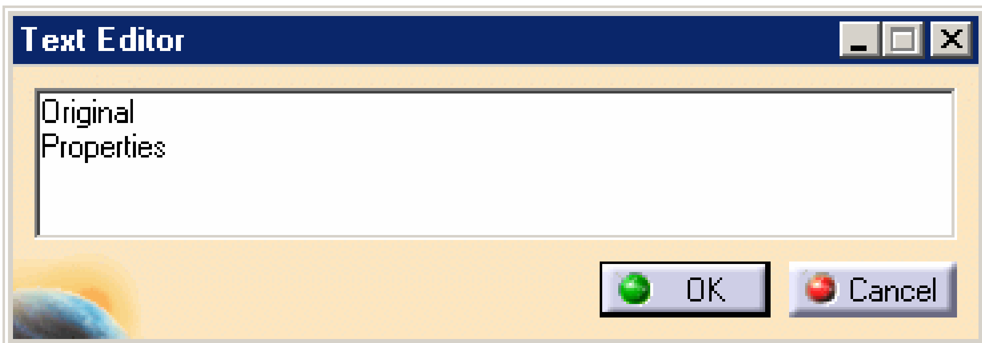
The **Set as Default** command record the graphical properties regardless of the selected option in the **Style** toolbar.
Default graphical properties before the first use of the command are application default properties.
See Text Graphical Properties.

3. Select the **Original Properties** option in the **Style** toolbar.

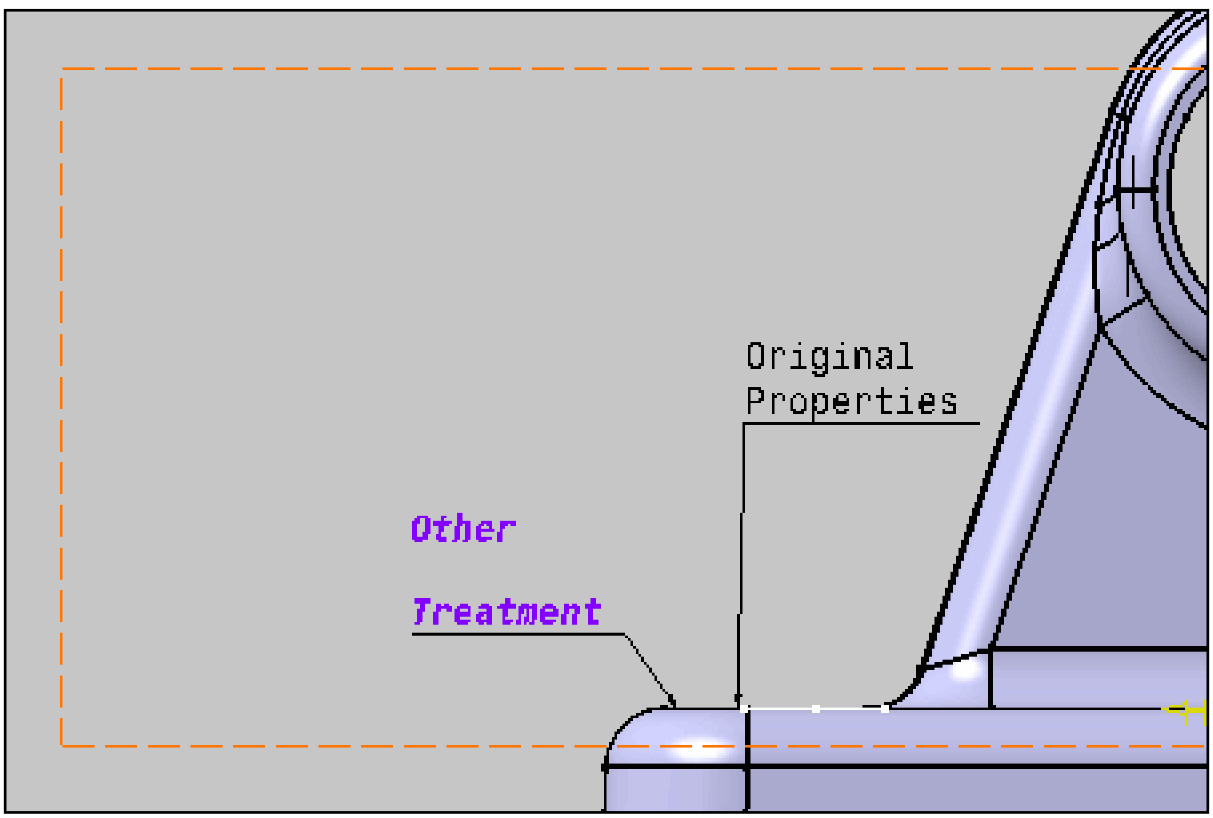


4. Create a new text and enter the following text: Original + Press Enter + Properties

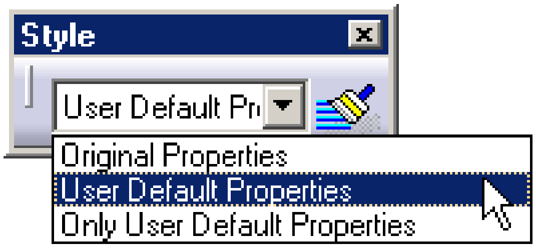
See Creating an Annotation Text.



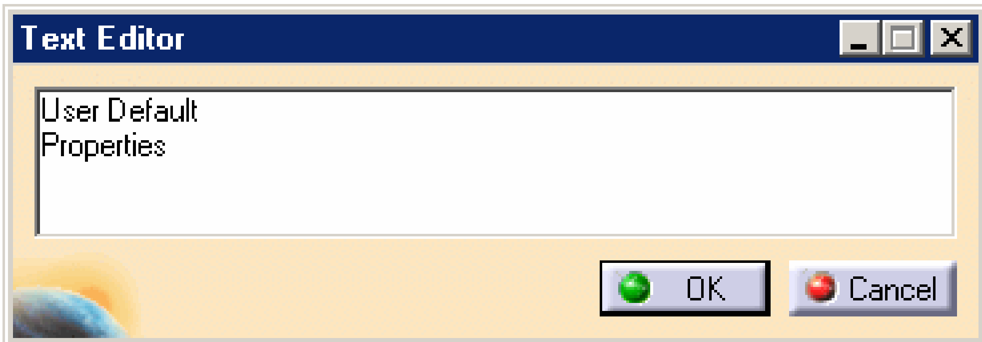
The text appears in the geometry and takes only the application default properties.



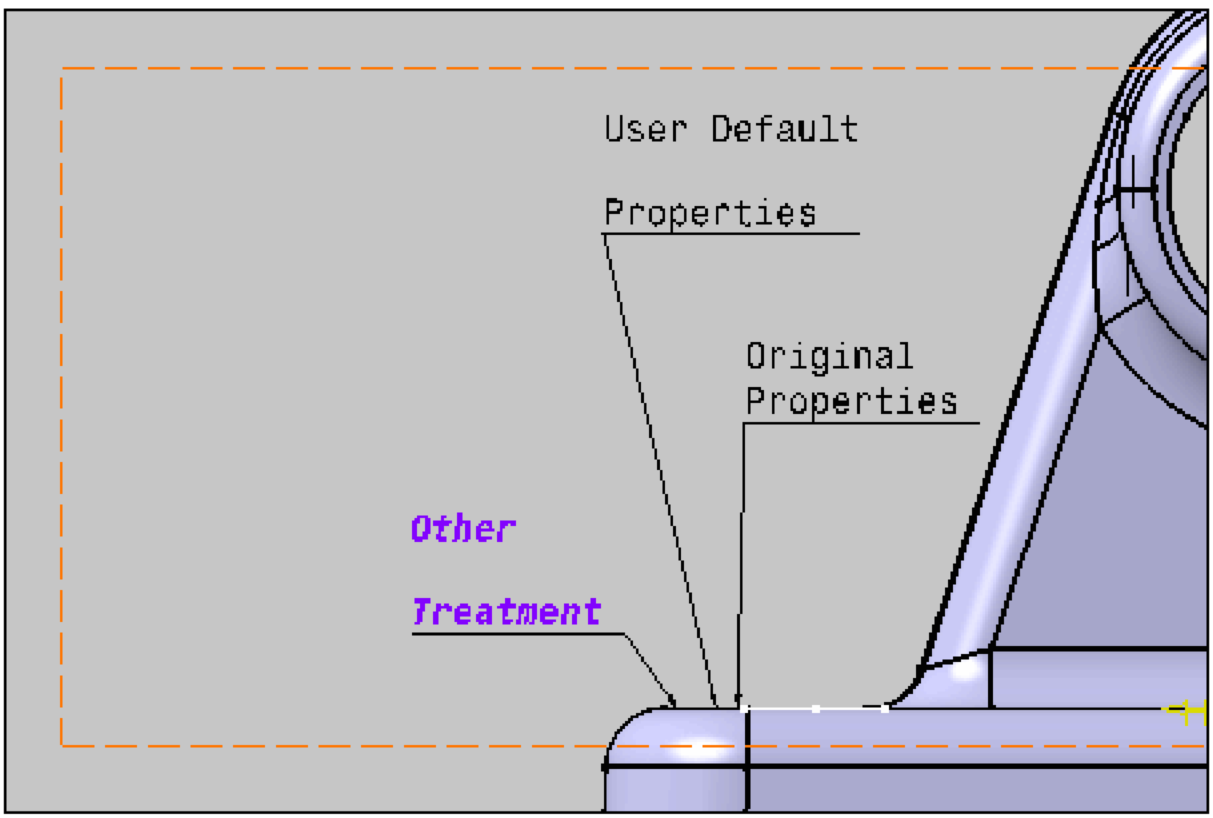
5. Select the **User Default Properties** option in the **Style** toolbar.



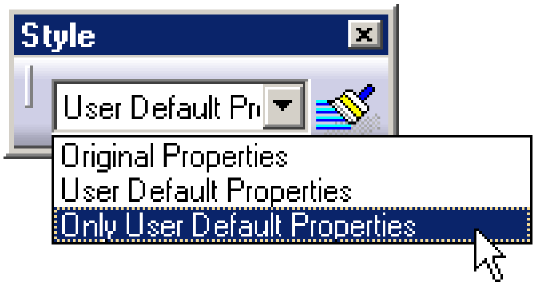
6. Create a new text and enter the following text: User Default + Press Enter + Properties



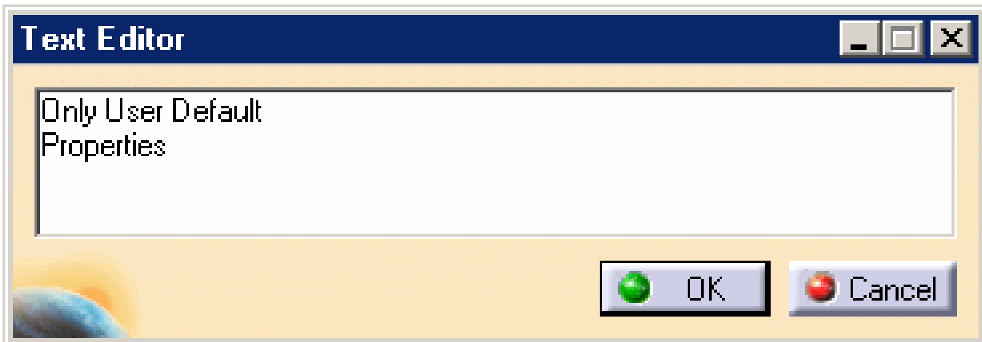
The text appears in the geometry and takes the application default properties and the **Text: Line Spacing** property from the first text.



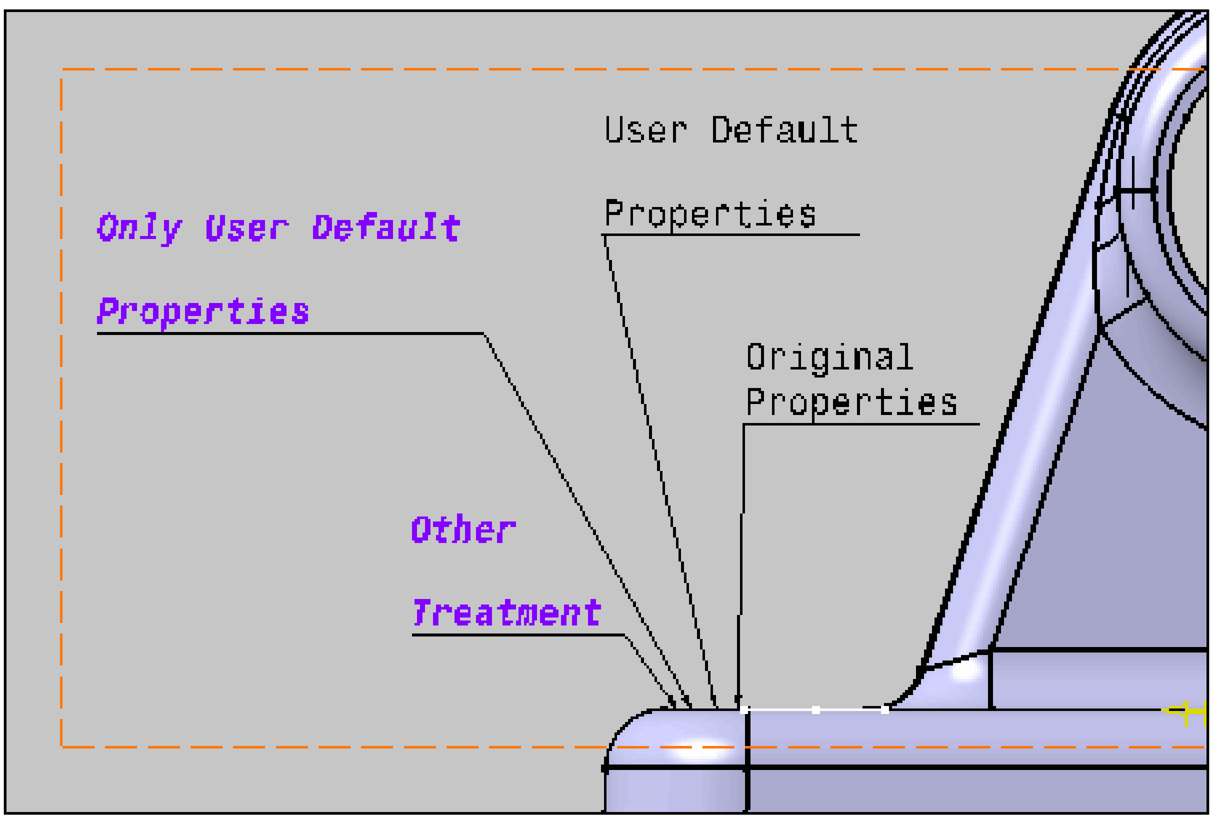
7. Select the **Only User Default Properties** option in the **Style** toolbar.



8. Create a new text and enter the following text: Only User Default + Press Enter + Properties



The text appears in the geometry and takes all the application properties from the first text: **Text: Line Spacing**, **Font: Color** and **Style**.



The leader color is never taken into account.

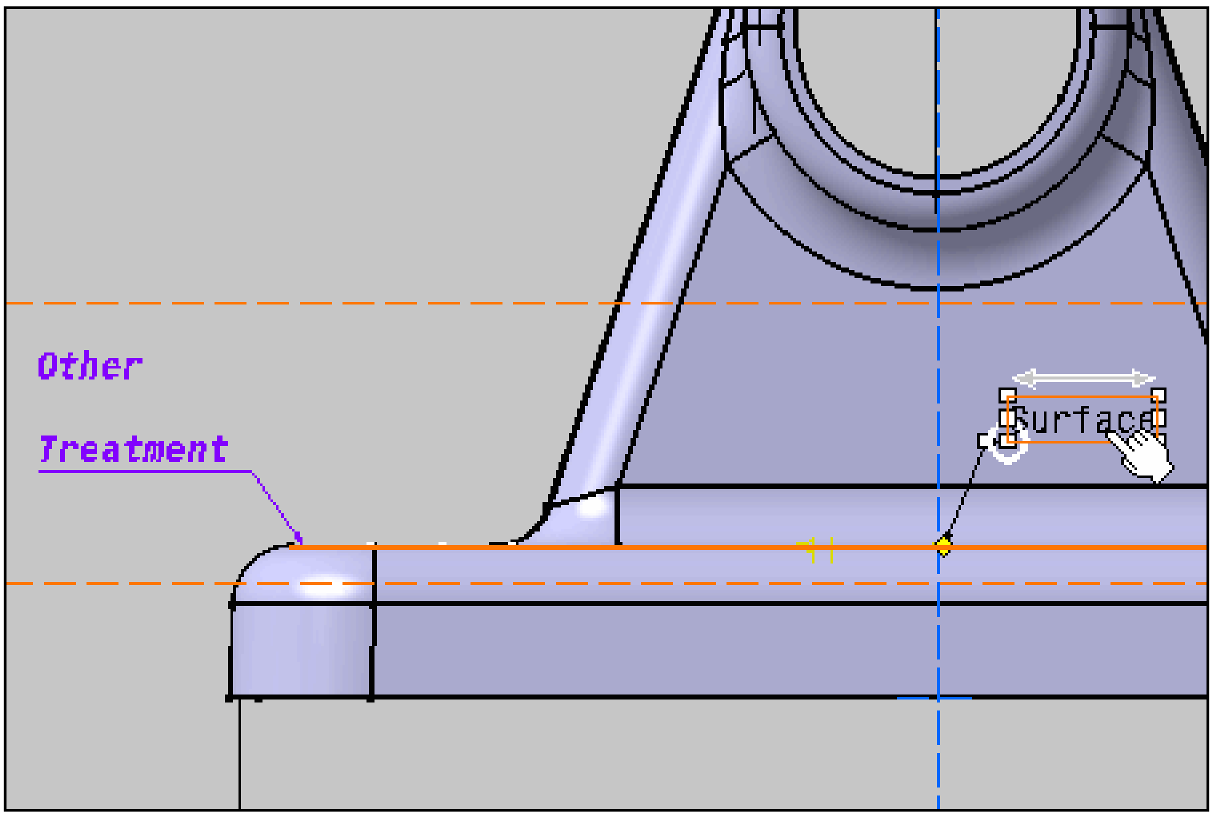
Copying Graphical Properties

This task shows you how to copy the graphical properties of a text to other existing texts.

Note that the operating mode described here applies to datum, datum targets and geometrical tolerances too.

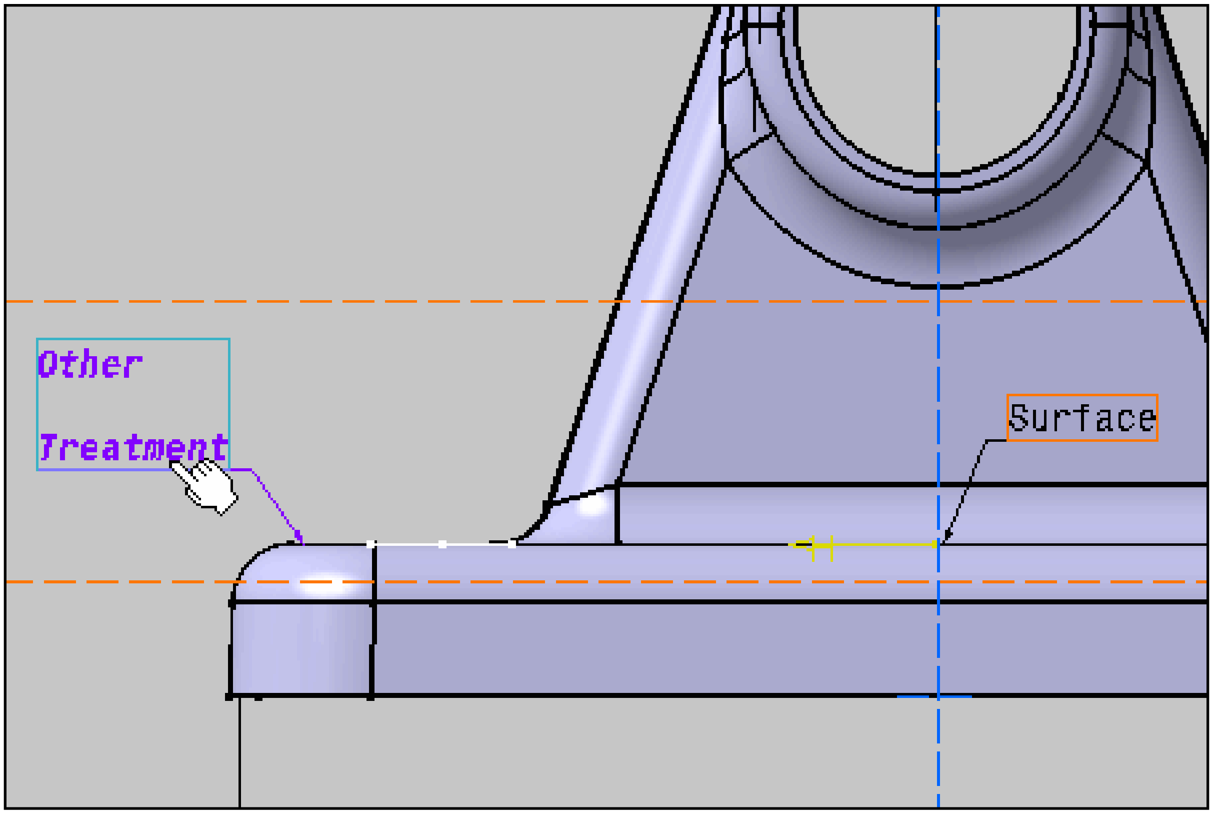
Open the Annotations_Part_04.CATPart document.

1. Select the text which graphical properties which are to be modified.

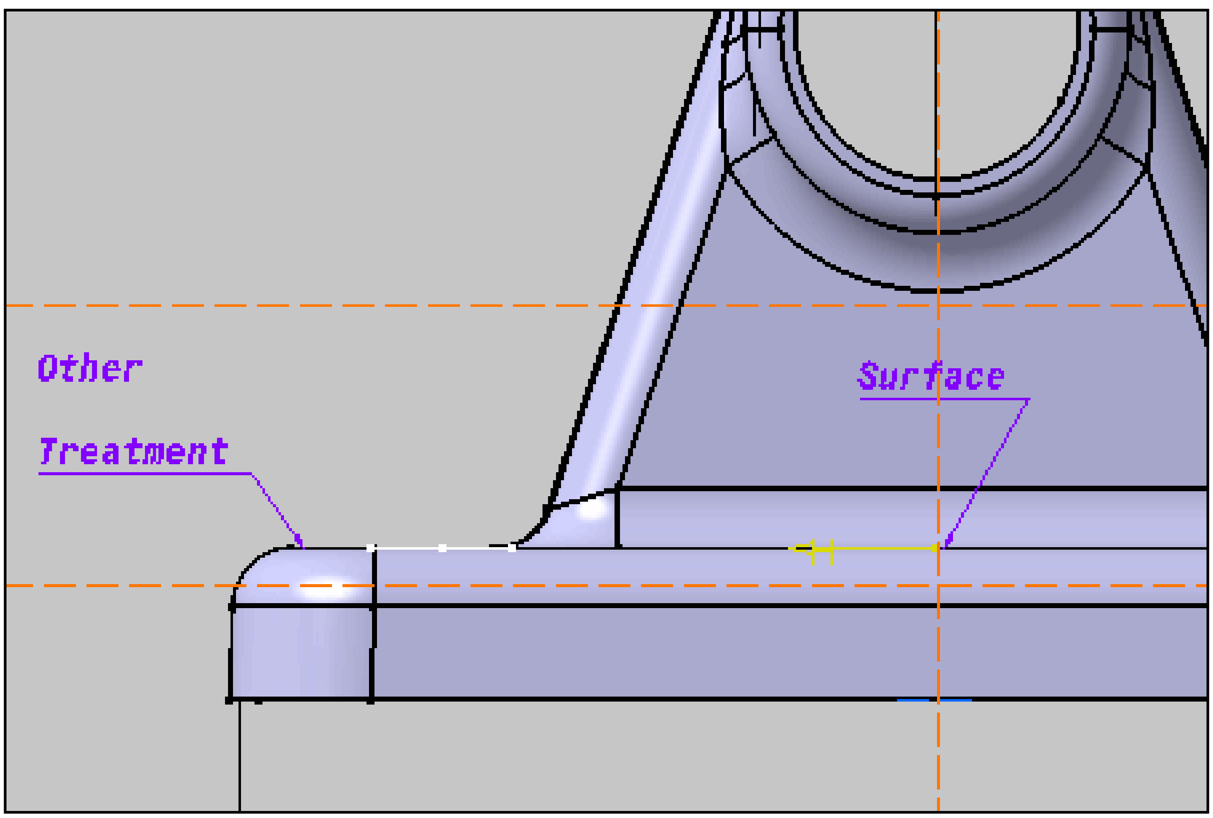


2. Click the **Copy Object Format** icon 

3. Select the text to be used as the graphic reference for the texts you selected.



The graphical properties assigned to the text used as the reference are now copied onto the selected text to be modified.
See Text Graphical Properties.



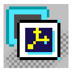
The leader color and anchor point are take into account.

Managing Annotations Display

 Filter Annotations: click the icon and specify the filter options.

 Create an Annotation Capture: click the icon to create the capture.

Display a Capture: right-click a capture and select Display Capture command from the contextual menu.

 Create a Camera: click the icon to create the camera.

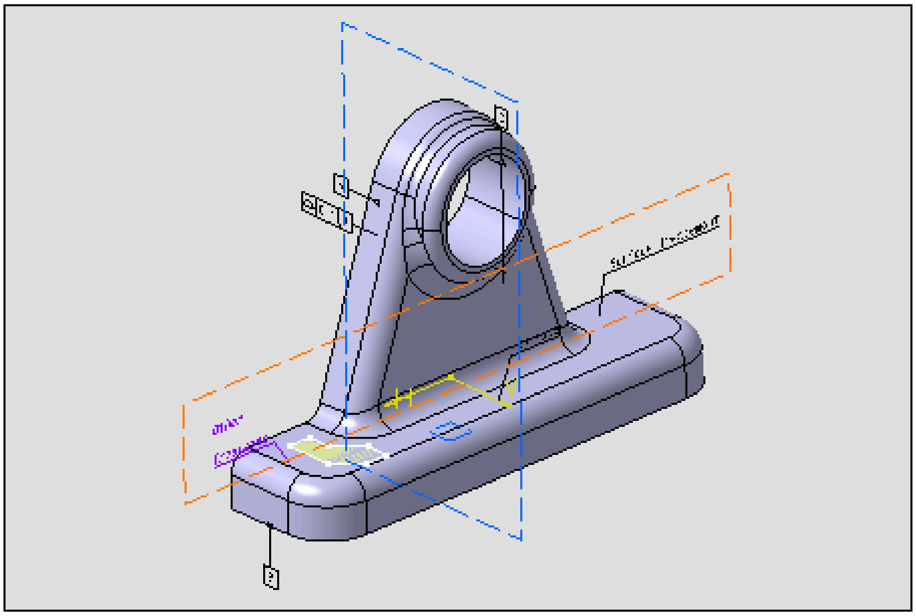
Manage Capture Options: set capture options.

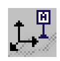
Use Capture Management: right-click one or several annotations and select Capture Management command from the contextual menu.

Disabling/Enabling Annotations

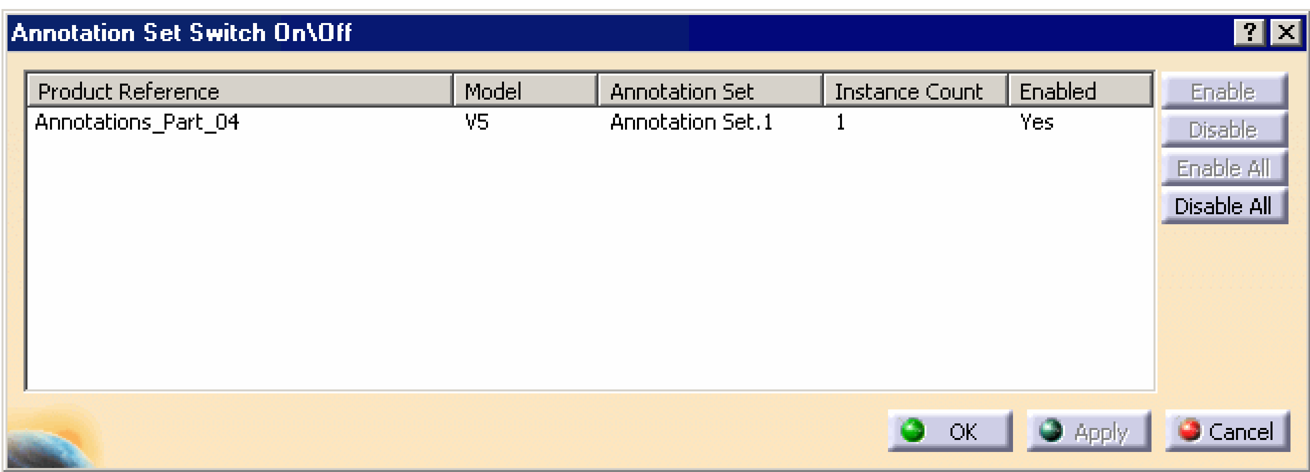
This task shows you how to disable and/or enable the annotations.

Open the Annotations_Product_04.CATProduct document.



1. Click the **List Annotation Set Switch On/Switch Off** icon: 

The **Annotation Set Switch On/Off** dialog box is displayed.



You can see the list of product references contained in the current document:

- In a CATPart document, the CATPart itself.
- In a CATProduct document or a CATProcess document, all the product references contained. The active component is not taken into account.

For each product reference are displayed:

- The Part Number.
- The model type.
- The list of annotation sets.
- The number of instances.
- The disable/enable status.

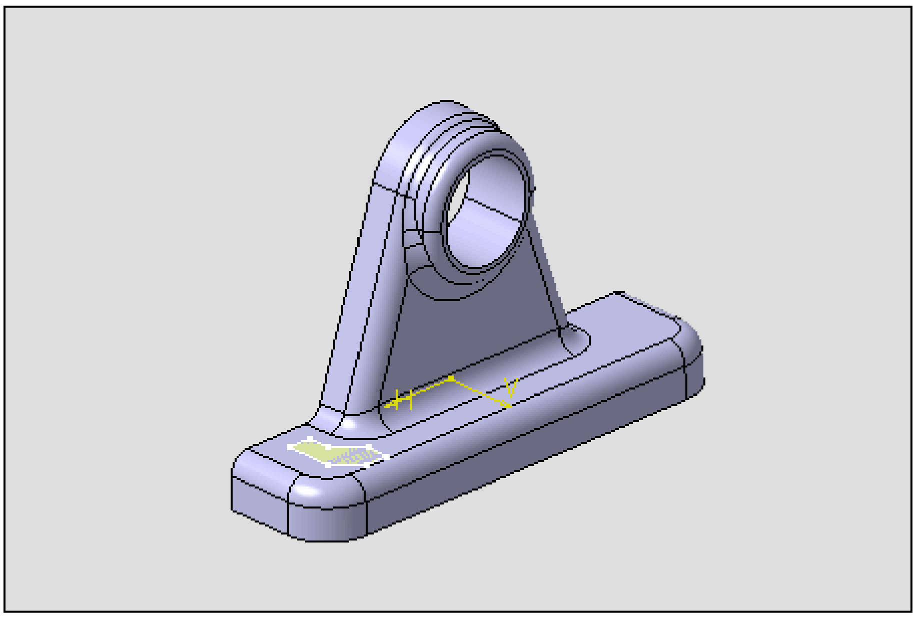
Functionalities:

- The **Enable** or **Disable** buttons affect the selected product references in the dialog box list.
- The **Enable All** or **Disable All** buttons affect all the product references in the document.

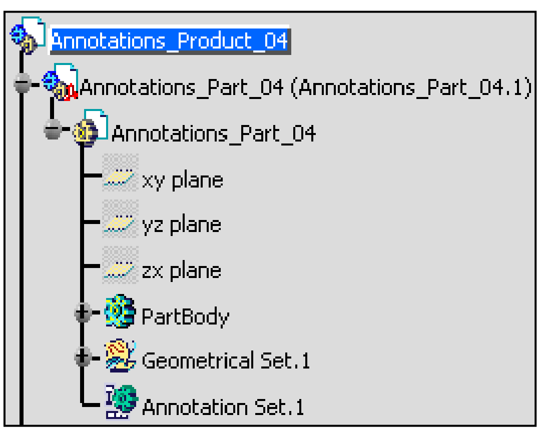
2. Click **Annotation_Part_04** then the **Disable** button.

3. Click **Apply** in the dialog box.

The annotations are disabled in the geometry.

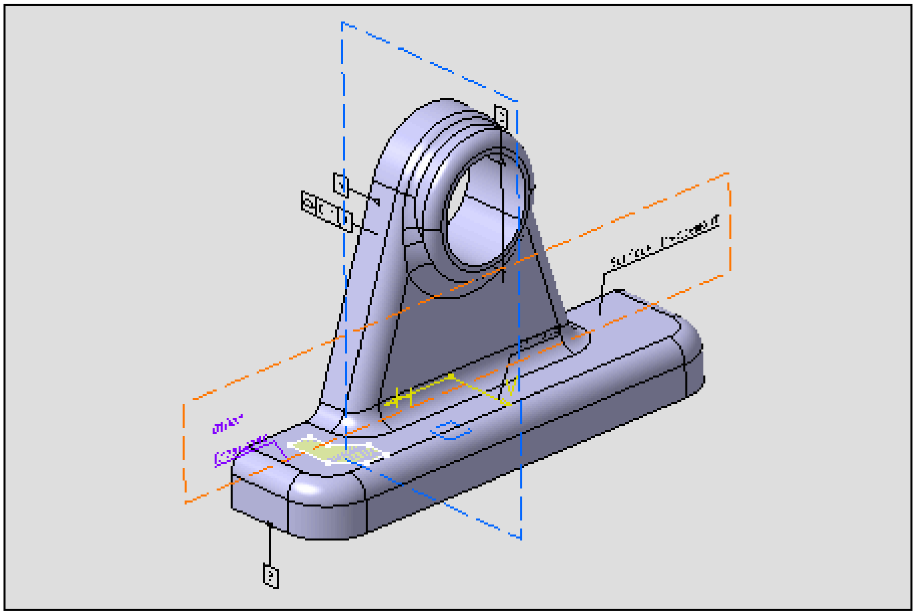


The specification tree no longer displays them.

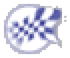


4. Click **Annotation_Part_04** then the **Enable** button.

The annotations are enabled in the geometry.



5. Click **OK**.



Filtering Annotations

This task shows you how to filter the display of annotations.

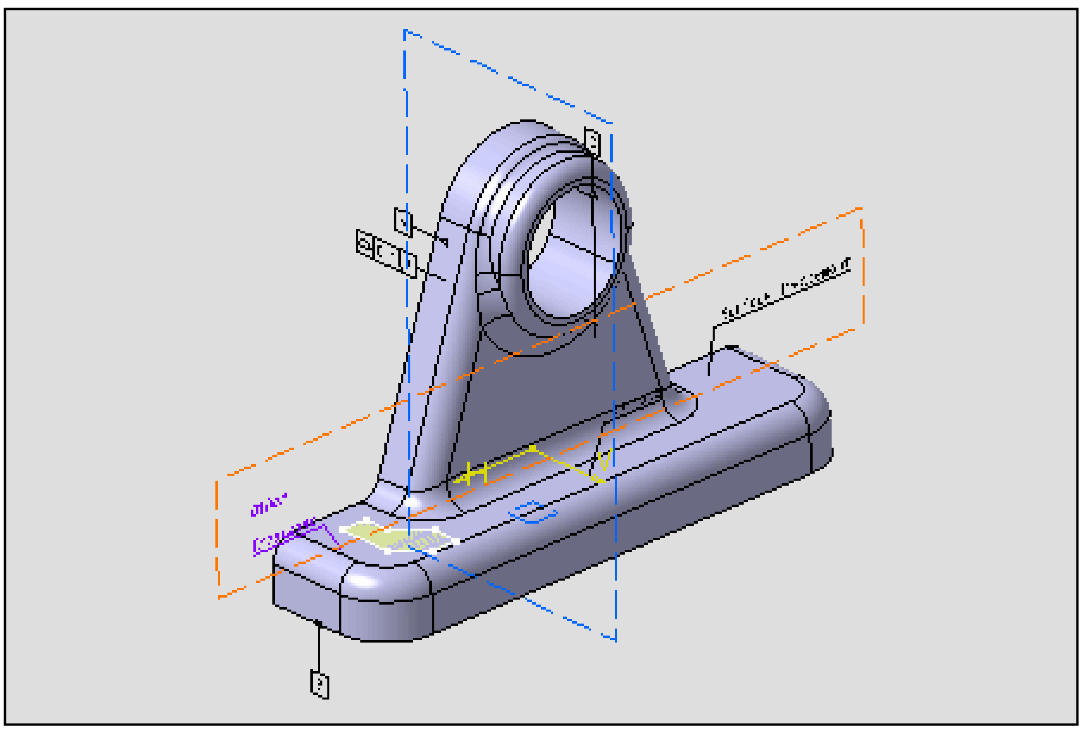
You can filter annotations display though the following features:

- Views/annotation planes
- Annotation sets
- Geometrical elements
- 3D annotations
- Any Part Design feature
- Any Generative Shape Design feature
- Restricted areas

You can filter annotations in the **Visualization mode** context. See Annotations and Cache System.

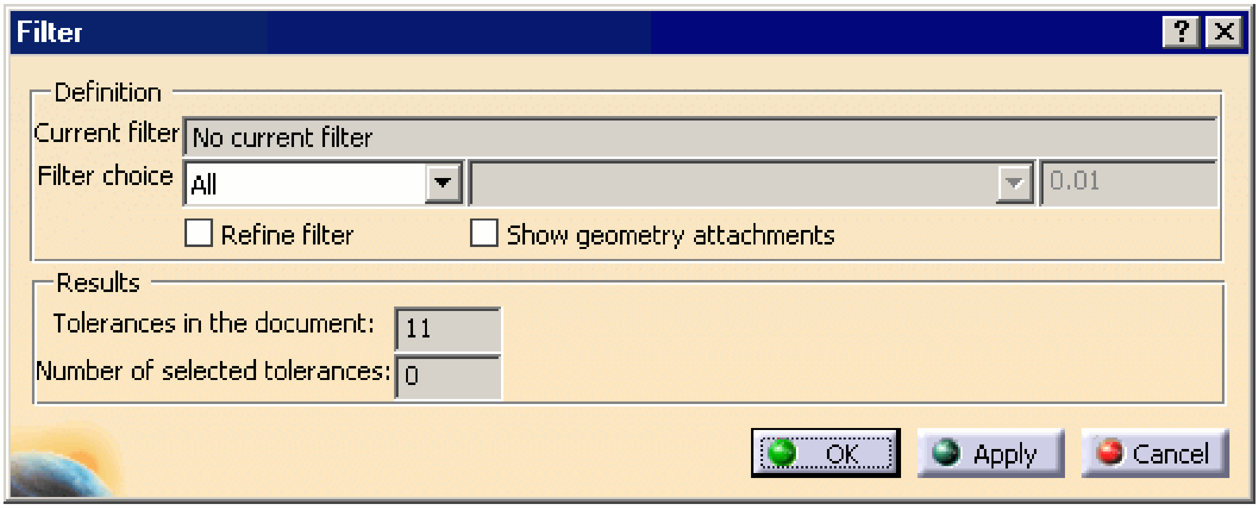
In the case of Part Design or Generative Shape Design features, only the annotations that are directly or indirectly applied to the geometrical elements which compose the feature will be displayed when applying the filter. In the case of restricted areas, only the annotations that are directly or indirectly applied to the geometrical elements which compose the restricting part of the restricted area will be displayed when applying the filter.

Open the Annotations_Product_04.CATProduct document.



1. Click the **Filter** icon:

The **Filter** dialog box is displayed.



The **Definition** area allows you to filter the display of annotations in the 3D viewer using the following criteria:

- All: displays all the geometrical tolerance annotations.
- None: displays no geometrical tolerance annotation.
- By type: non semantic.
- By sub-type: text, datum, datum targets, geometrical tolerances.
- By feature (Part Design or Generative Shape Design feature) or geometrical element.
- By capture.

The **Refine filter** option filters out tolerances still filtered.

The **Show geometry attachments** option displays the annotation leader if exists, and all the linked annotations between the leader and the filtered annotation if needed.

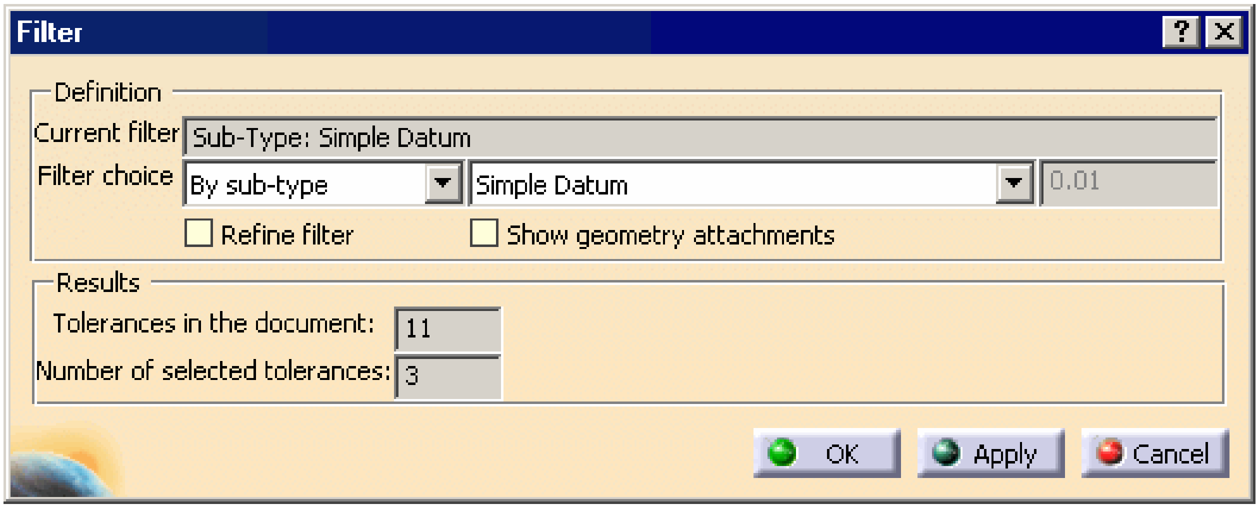
The **Results** area provides the following information:

- Number of specified tolerances attached to the 3D model
- Number of tolerances selected according to the choice indicated in the two previous fields

However, when default tolerances are specified, the number of tolerances displayed attached to the model does not correspond to the number of tolerances effectively specified. The default tolerance annotation is displayed once and the default tolerance specification is applied to several entities. These several specific toleranced entities are considered in the count of the **Tolerances in the document** field.

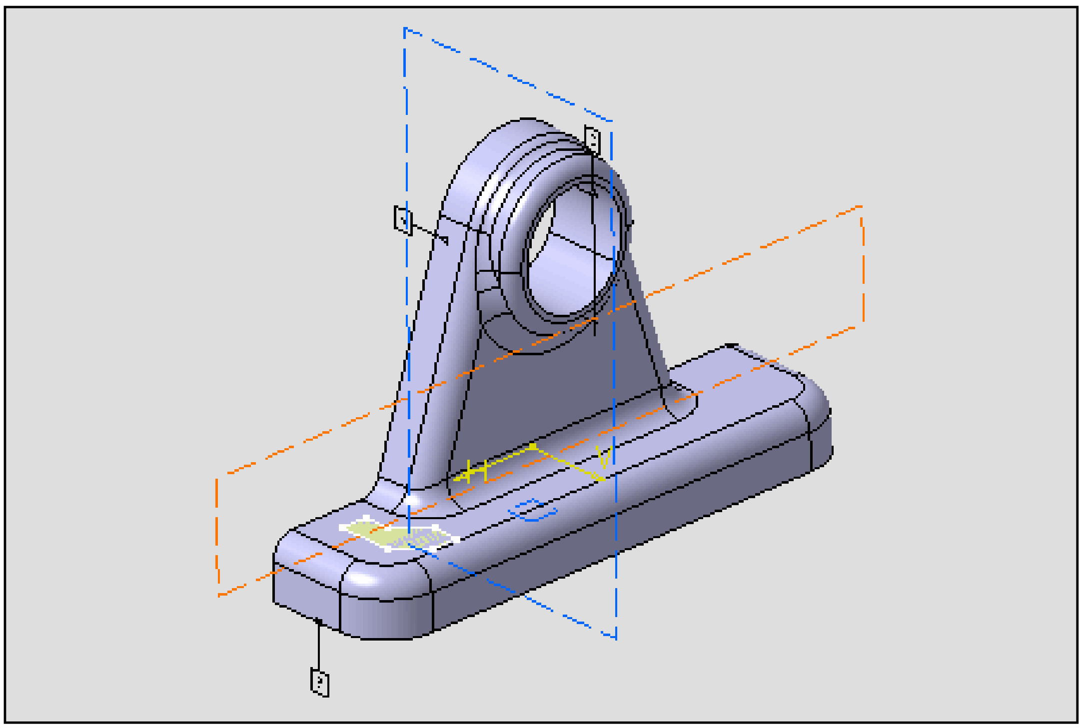
2. Set the **Filter** choice field to **By sub-type**.

3. Set the **Simple Datum** sub-type. The Number of selected tolerances field displays **3**.

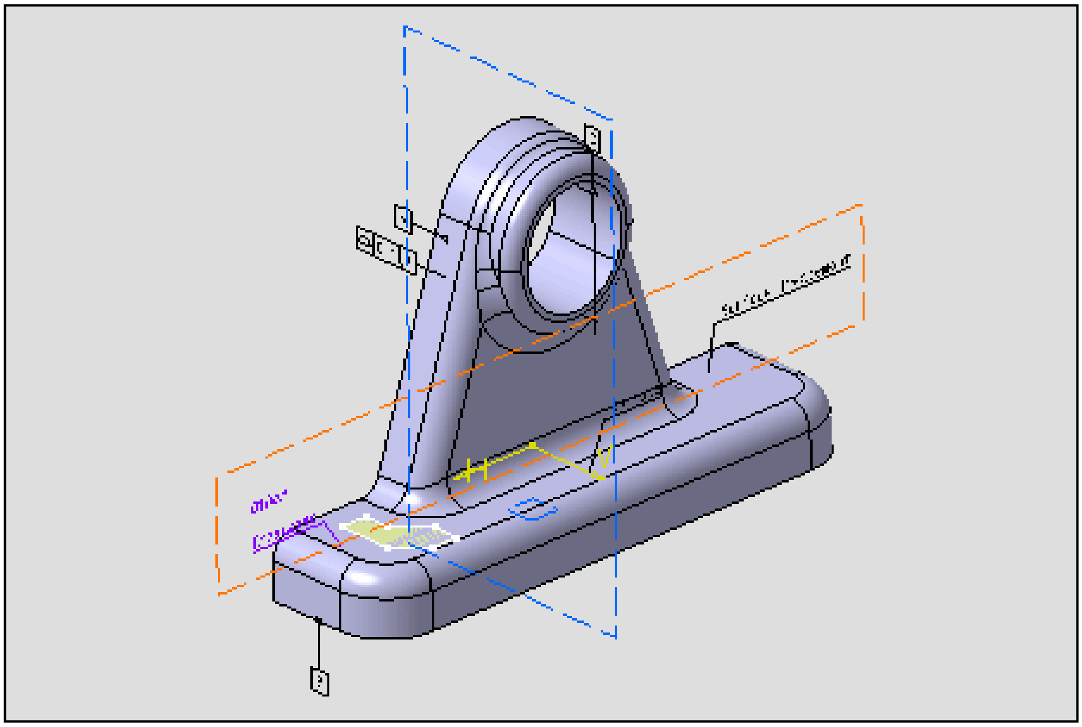


4. Click **Apply**.

Only simple datums are displayed.



5. Click **Cancel** to cancel the operation. All annotations are visible again.



Creating a Tolerancing Capture

This task shows you how to create a tolerancing capture.

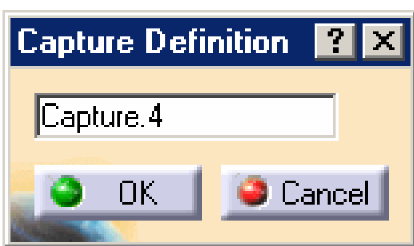
The purpose of capture features is to provide in 3D the views/sheets of 2D drawings flexibility for annotation display organization. It allows you to organize the display of 3D annotations the way you want.
You can, for instance, create capture for:

- Functional part area.
- Specifications answering to given functional requirements.
- 2D view equivalent.
- 2D sheet equivalent.
- etc.

Open the Tolerancing_Annotations_05 CATPart document.

1. Click the **Capture** icon: 

   The **Capture Definition** dialog box is displayed.

2. Click **Ok**.

   The **Capture.4** is created and displayed in the specification tree.

You are now in the Tolerancing & Annotation Captures workshop.

2. Click the **Exit from capture** icon:

You are back in the Functional Tolerancing & Annotations workbench.

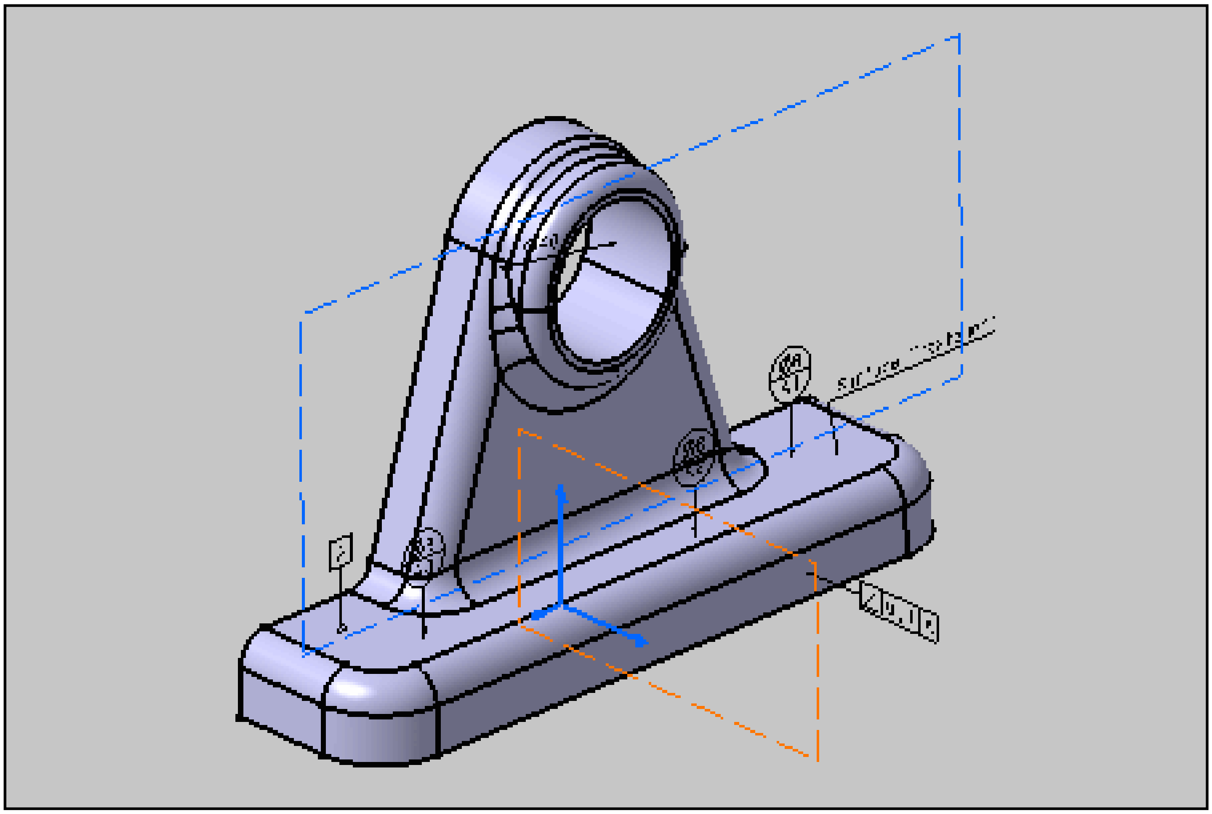
To edit a capture, double-click the capture in the specification tree.

Displaying a Tolerancing Capture

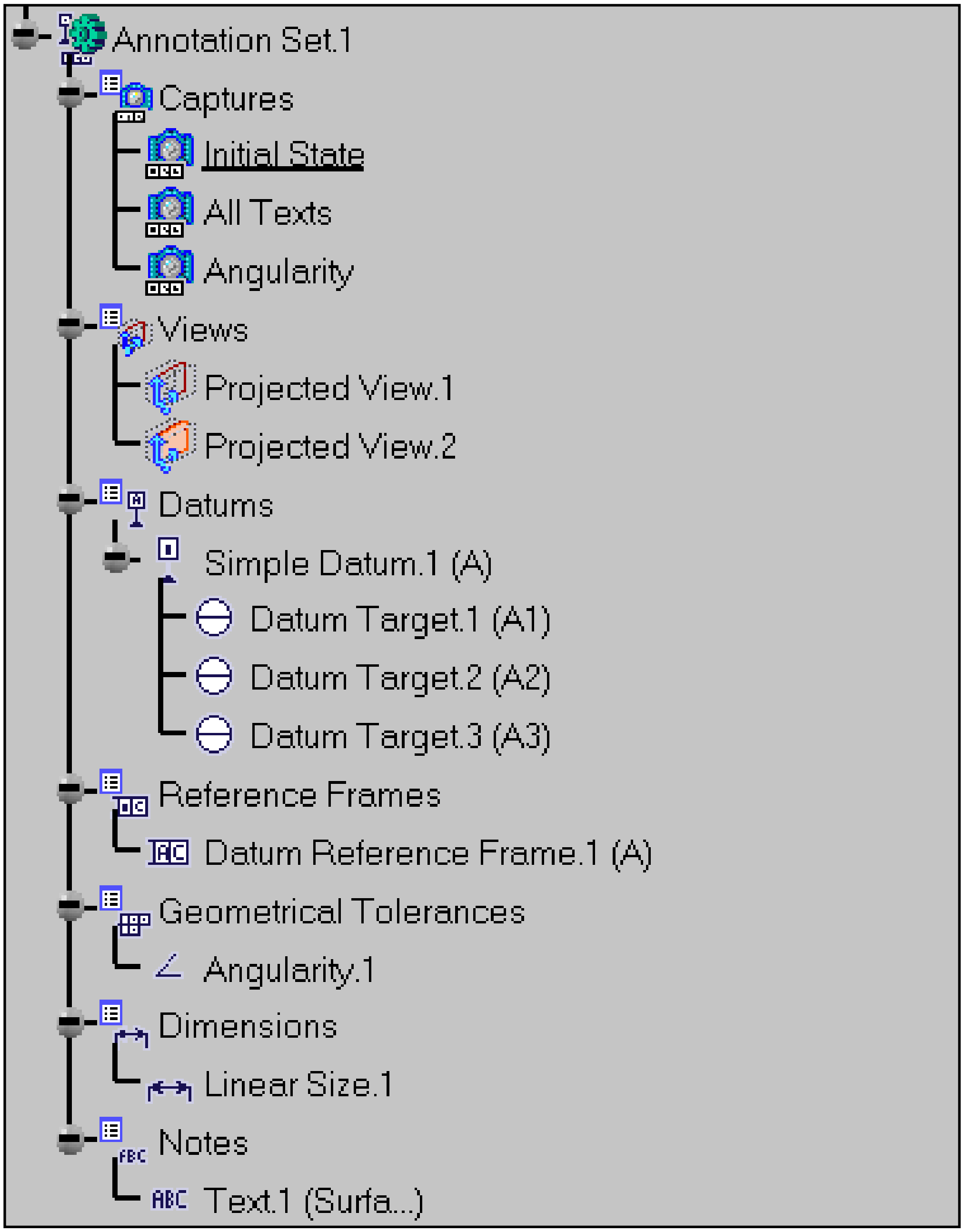
This task shows you how to display a tolerancing capture.
See Managing Capture Options.

Open the Tolerancing_Annotations_05 CATPart document.

The geometry looks like this.



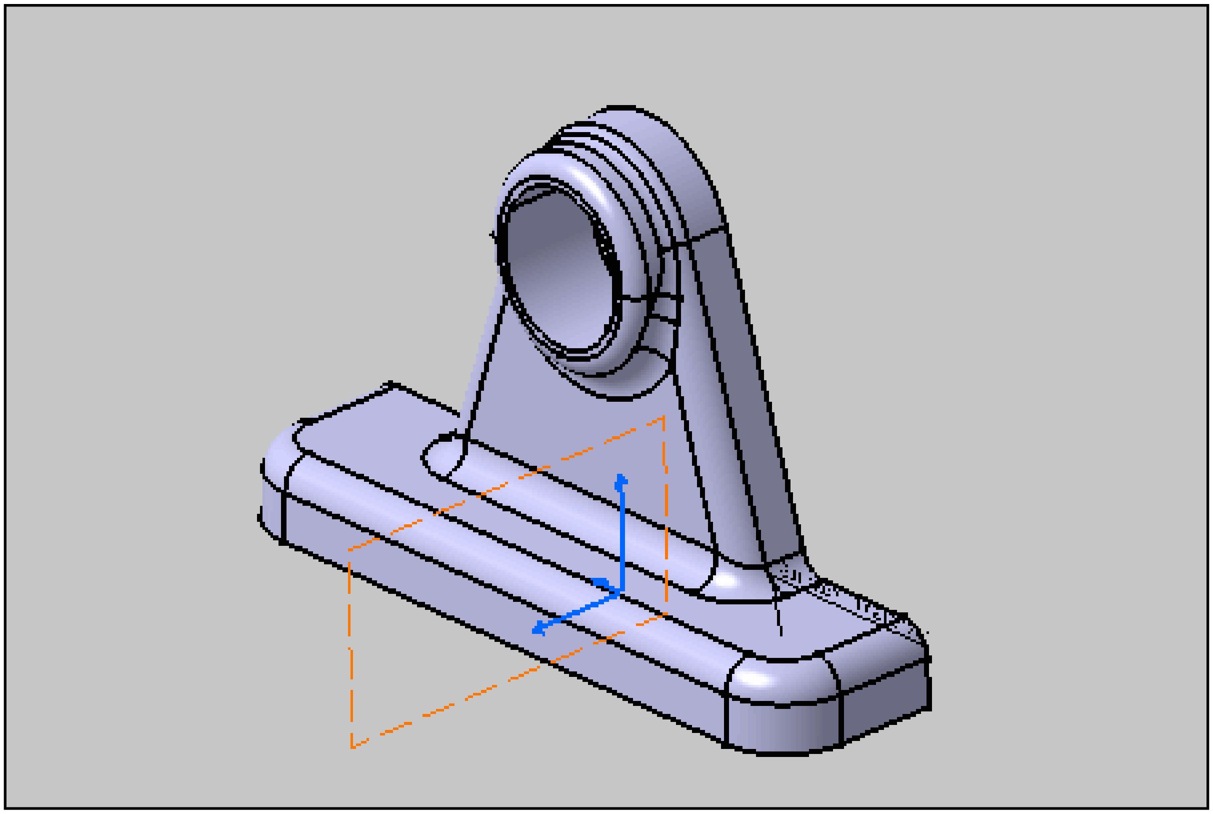
The specification tree looks like this.

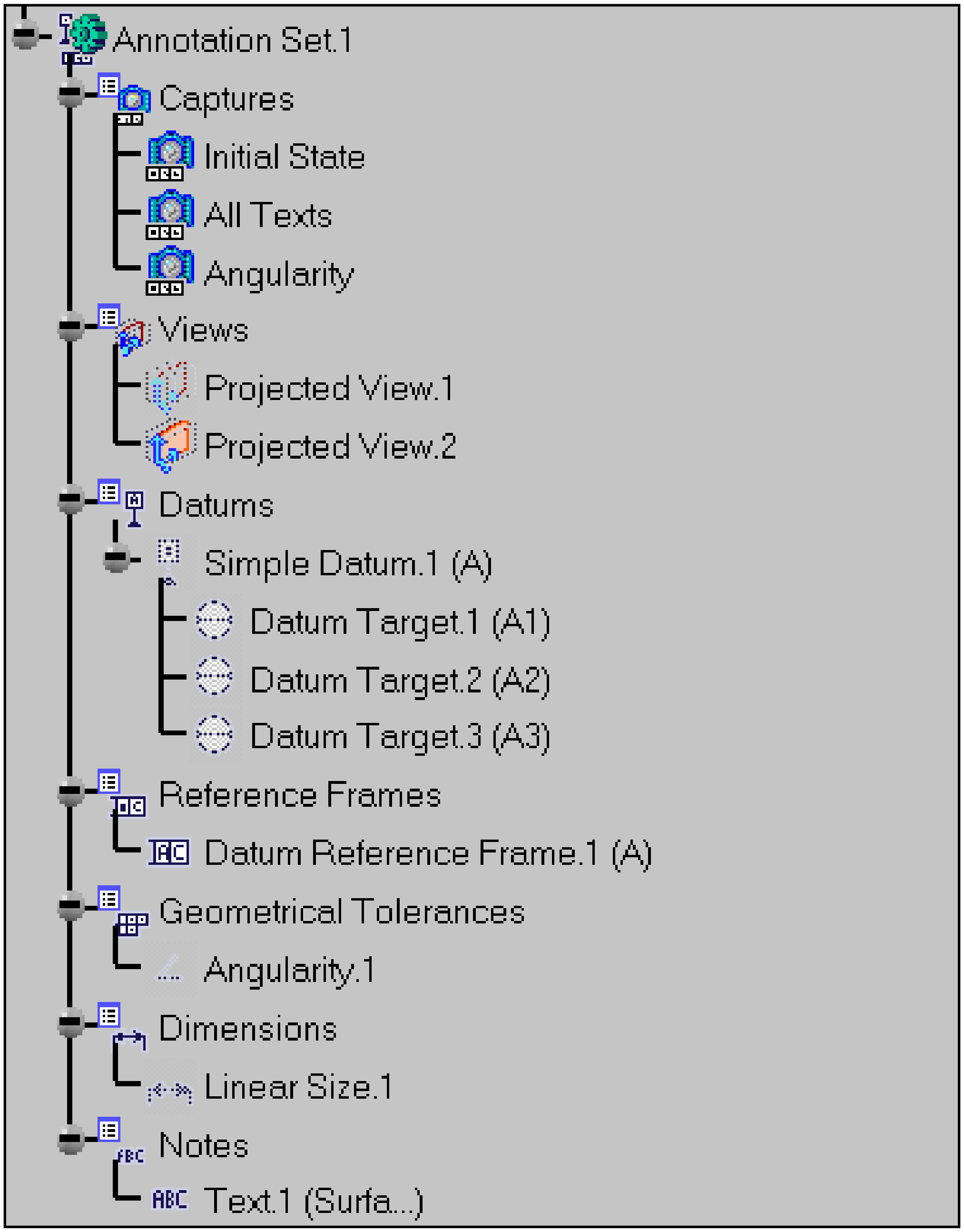


1. Right-click the **All Texts** capture and select **Display Capture** command from the contextual menu.

The geometry and specification tree look like this:

- The geometry is zoomed and rotated.
- All annotations are hidden except notes.
- **Projection View.1** is hidden.

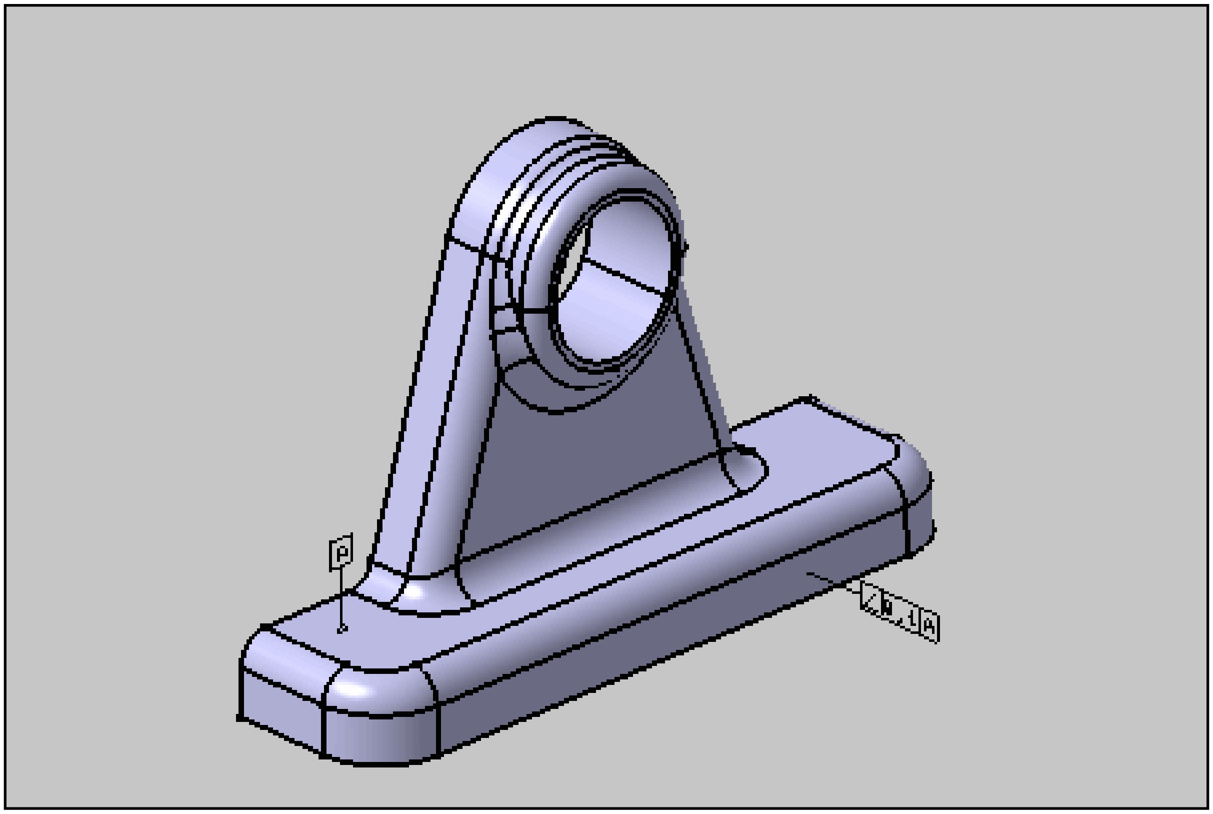


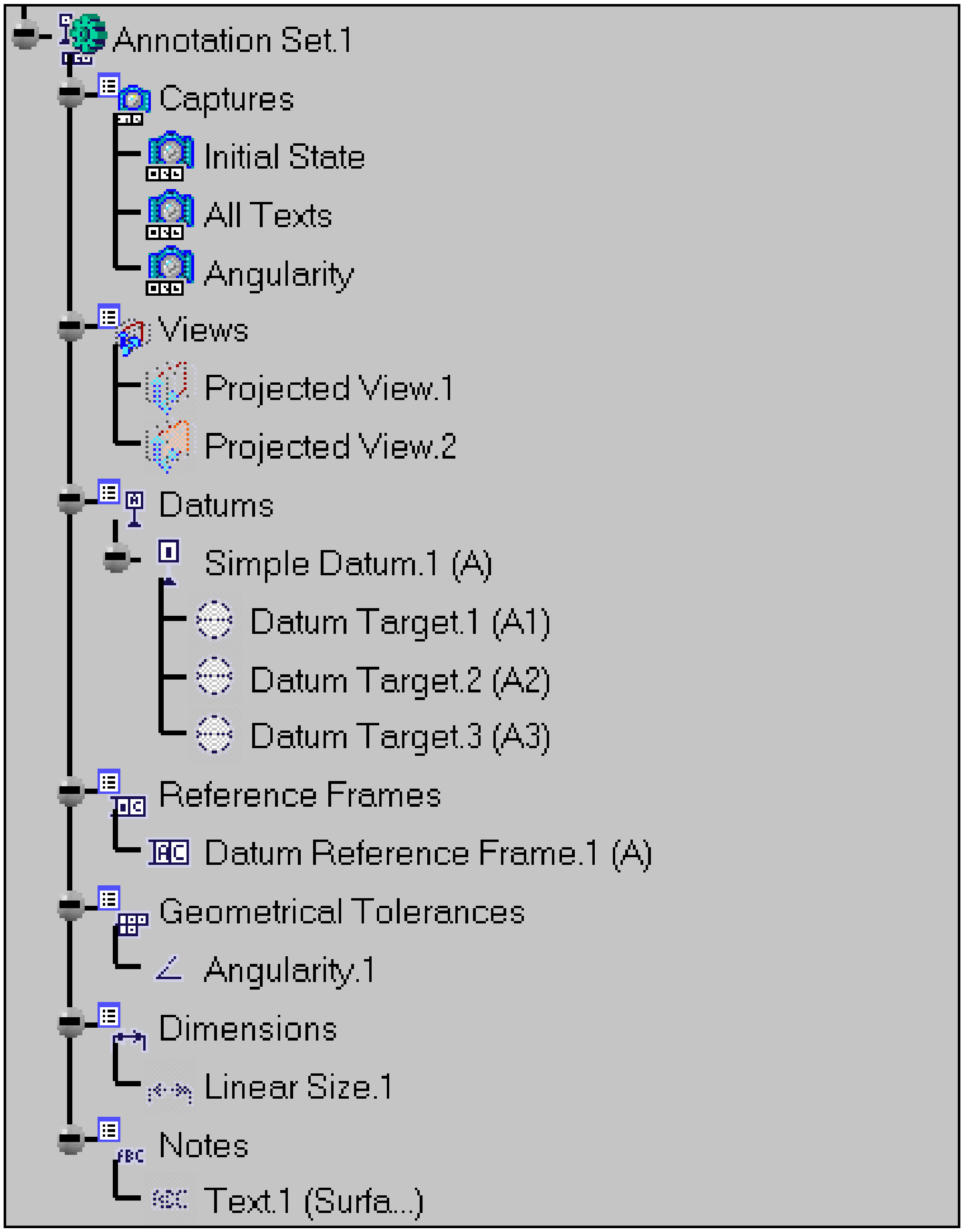


2. Right-click the **Angularity** capture and select **Display Capture** form the contextual menu.

The geometry and specification tree look like this:

- The geometry is moved and rotated.
- Annotation planes are hidden.
- Only the datum and the geometrical tolerance are shown.



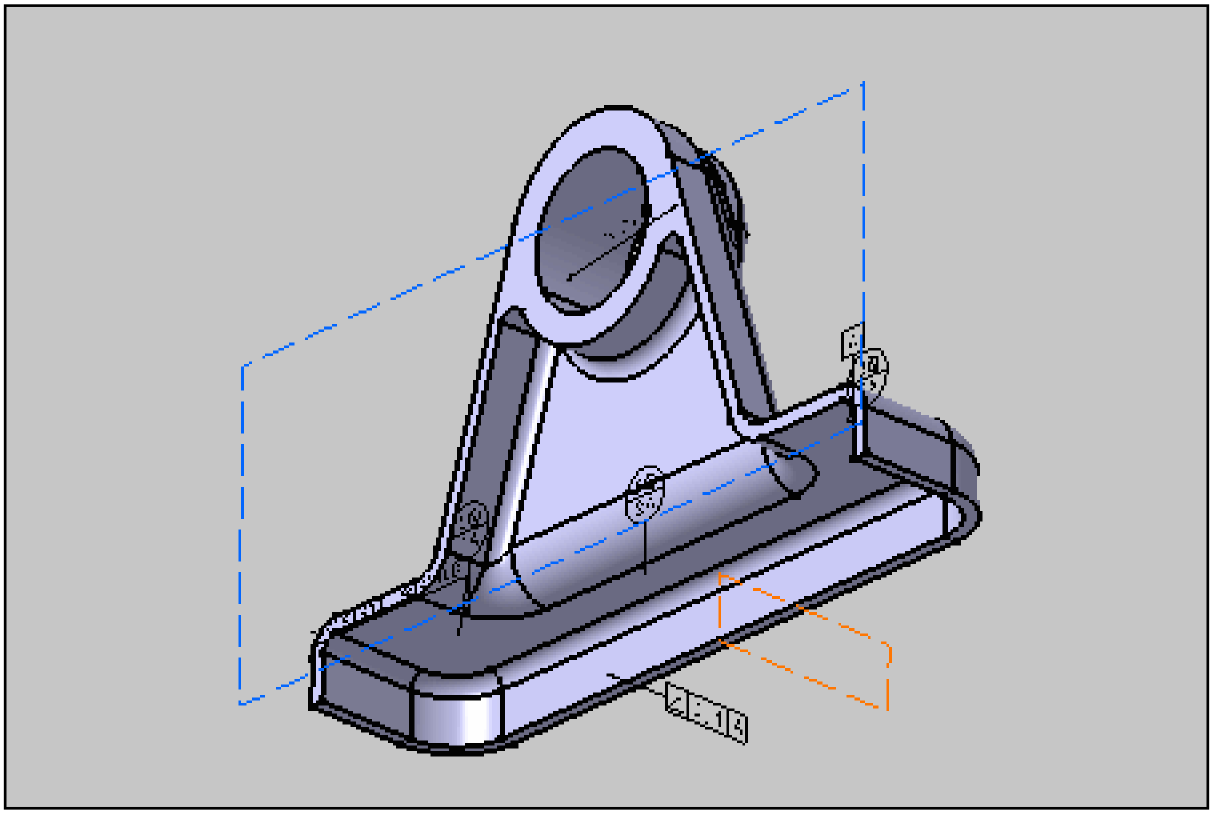

Annotation Set.1
Captures
Initial State
All Texts
Angularity
Views
Projected View.1
Projected View.2
Datums
Simple Datum.1 (A)
Datum Target.1 (A1)
Datum Target.2 (A2)
Datum Target.3 (A3)
Reference Frames
Datum Reference Frame.1 (A)
Geometrical Tolerances
Angularity.1
Dimensions
Linear Size.1
Notes
Text.1 (Surfa...)

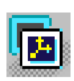
Creating a Camera

This task shows you how to create a camera in the tolerancing workshop.
This command is available for any workbenches from the **View -> Named Views...** menu.

Open the Tolerancing_Annotations_05 CATPart document.

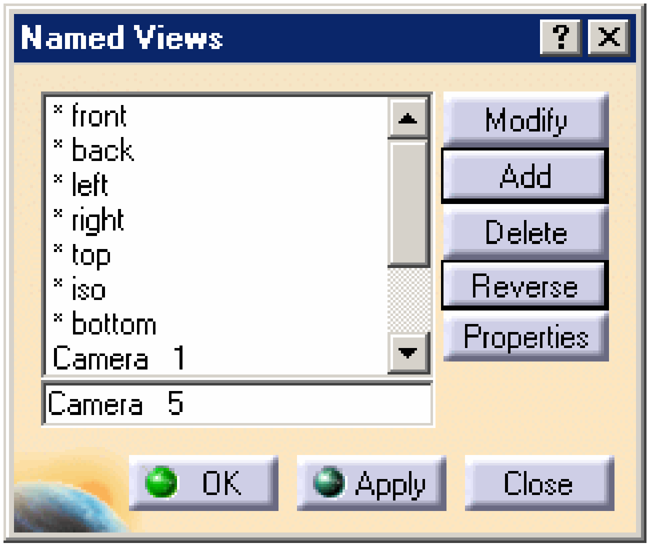
1. Move and rotate the part like this.



2. Select the **Named views** icon 

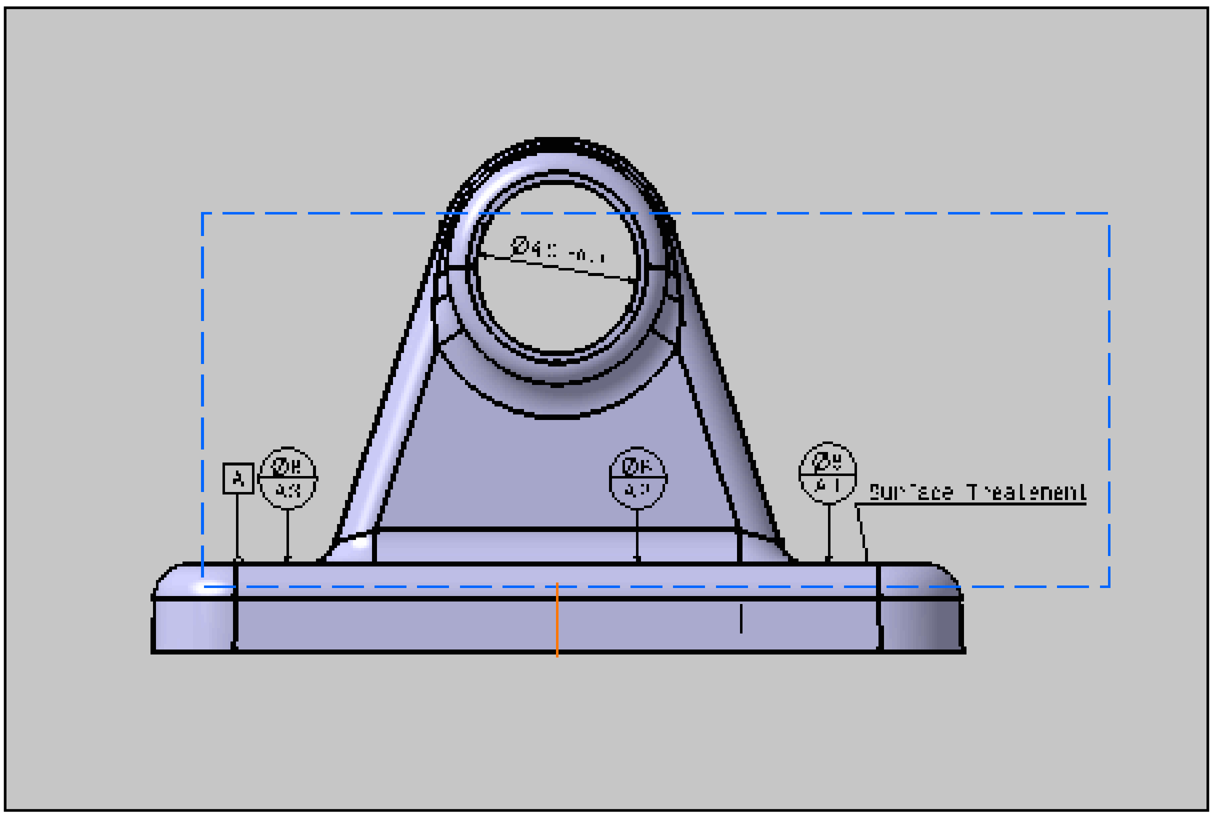
3. Click **Add** in the **Named Views** dialog box which appears.

The **Camera 5** is created.



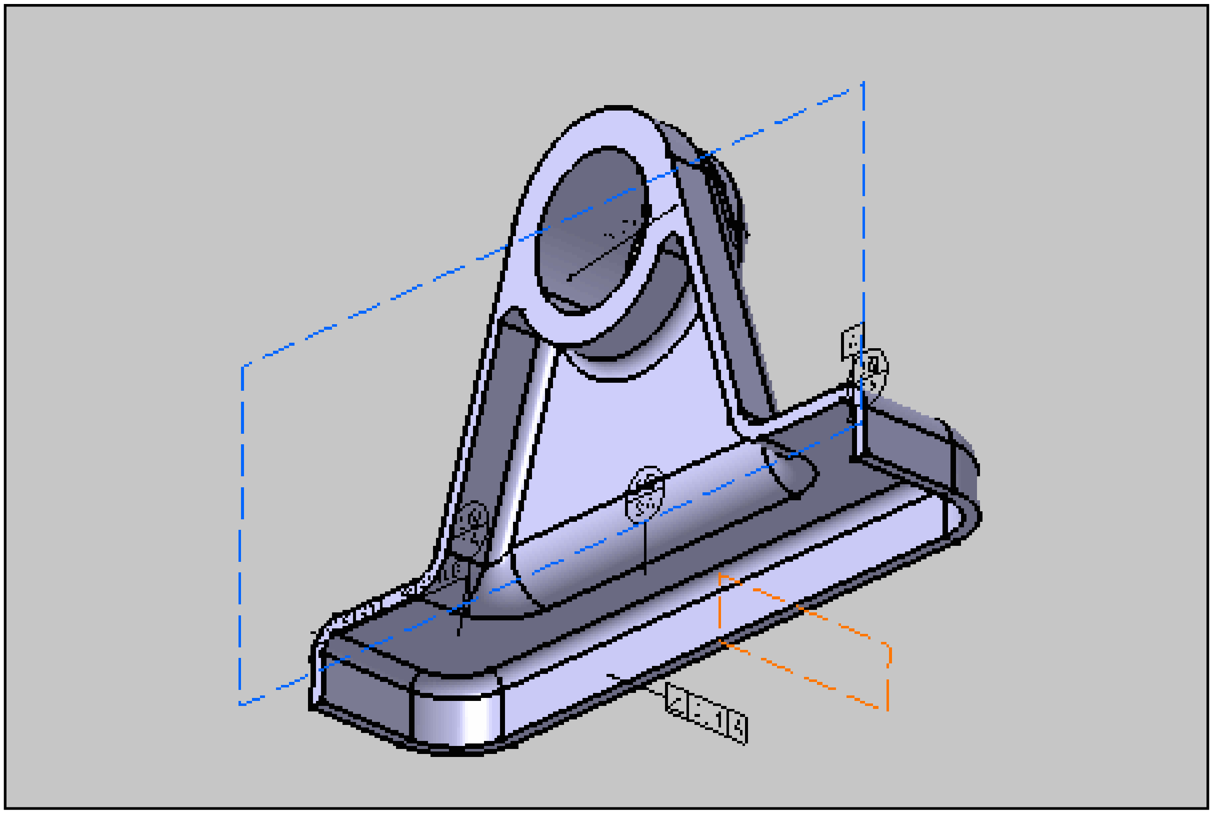
4. Select the *** right** camera the **Named Views** dialog box and click **Apply**.

The part move and rotate according to the camera.



5. Select the **Camera 5** camera and click **Apply**.

The part is moved and rotated according to the created camera.



6. Click **OK**.

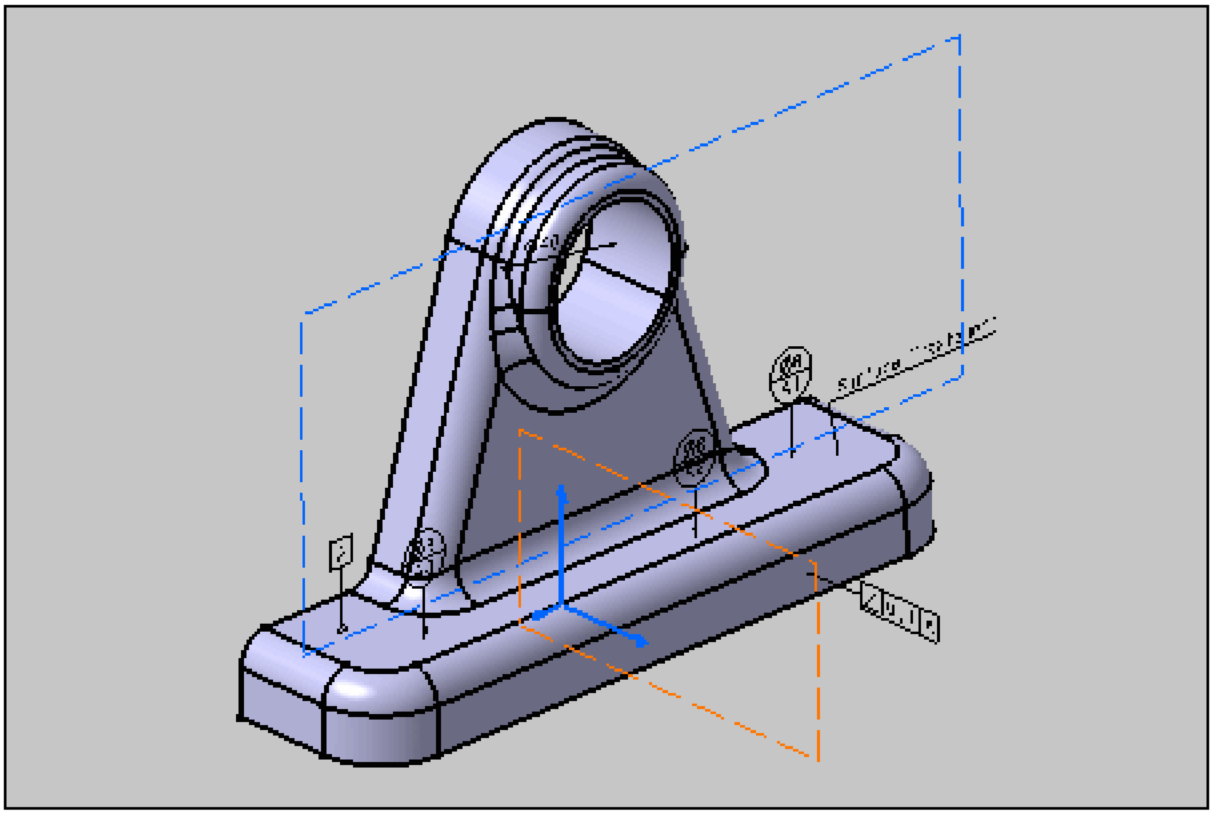
Managing Tolerancing Capture Options

This task shows you how to manage tolerancing capture options in existing capture.

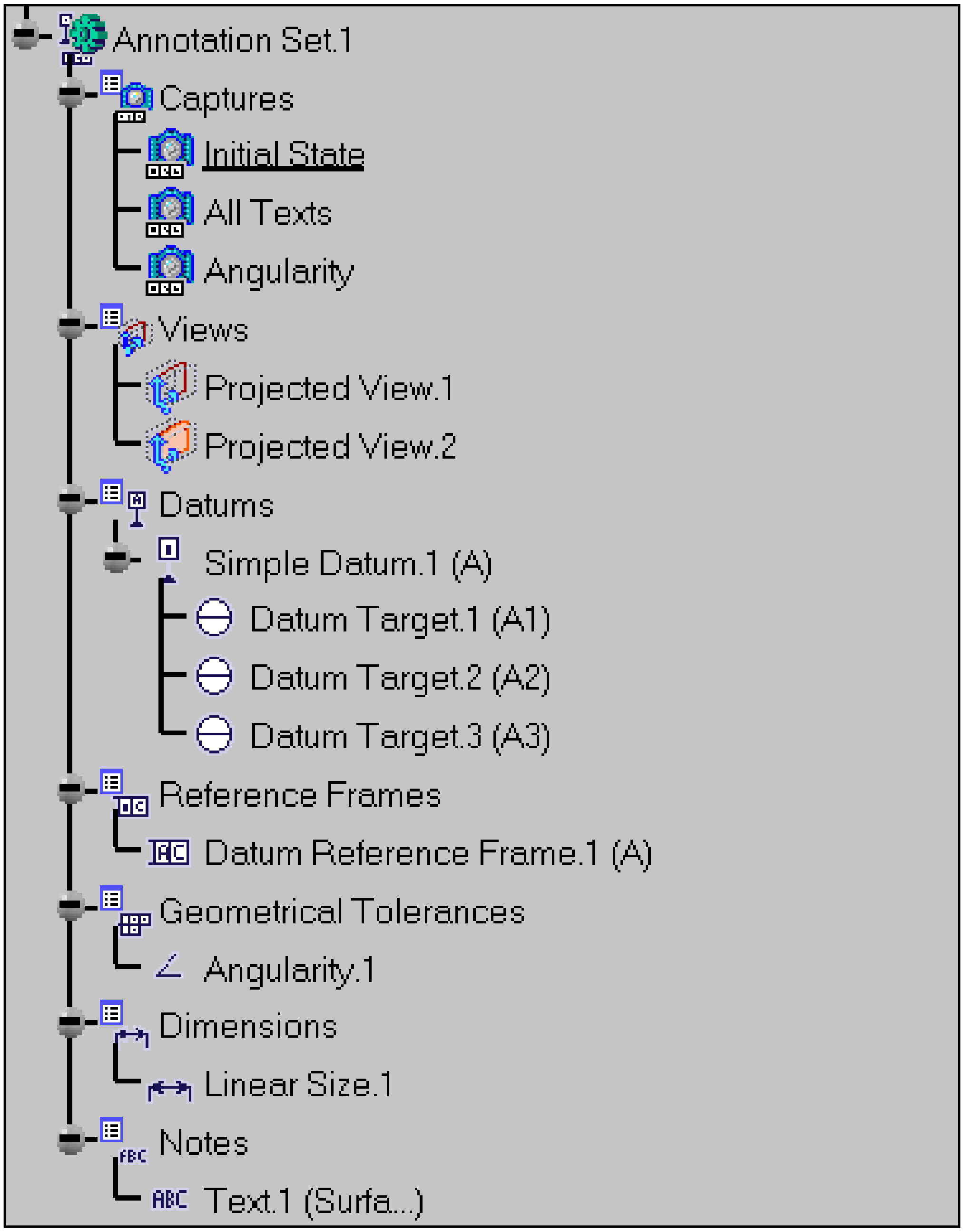
Open the Tolerancing_Annotations_05 CATPart document.

1. Double-click the **Initial State** capture.

   You are now in the Tolerancing & Annotation Captures workshop.

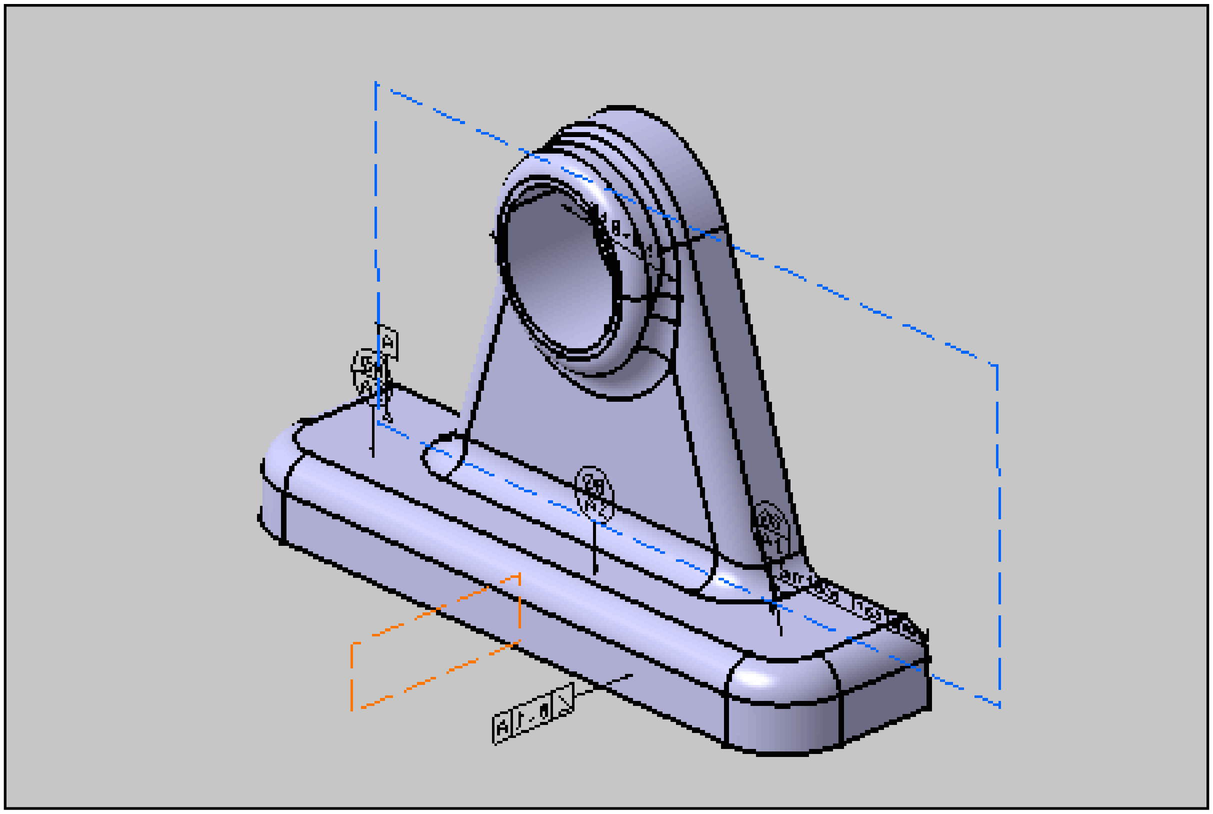


The geometry and specification tree are displayed according to the capture options.

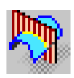


2. Select in the **Camera** combo box the **Camera 3**.

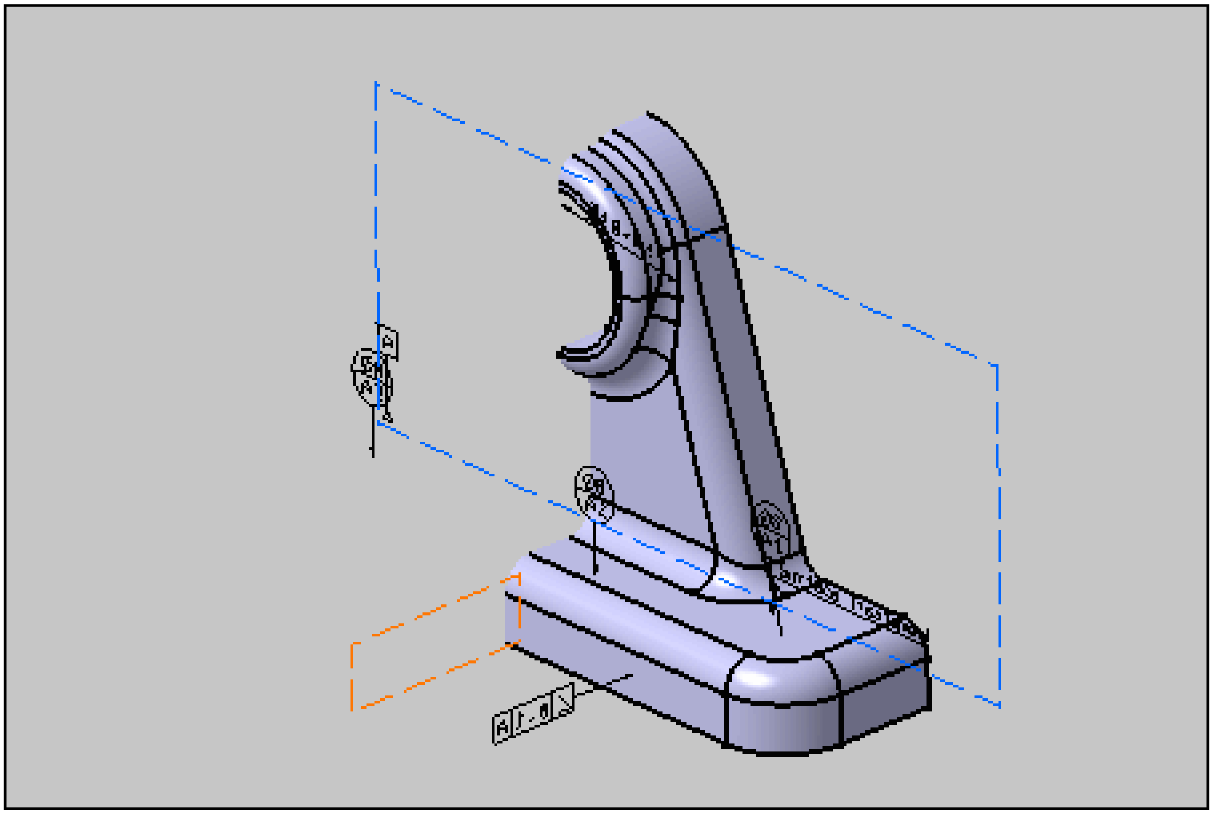
A camera is associated with the capture. See Creating a Camera.



You can re-use cameras created in the document with another workbench.

3. Click the **Clipping Plane** icon 

A clipping plane is associated with the capture.



4. Click the **Active View State** icon  , to associate the current annotation plane with the capture.

5. Click the **Current State** icon  and exit the workshop.

Now, all the new annotations or annotation planes are associated with this capture while it is activated.

This option is also available by right-clicking a capture in the **Functional Tolerancing & Annotation** workbench and selecting **Set Current** or **Unset Current** commands from the contextual menu.
You can associate annotations and/or an annotation plane to one or several captures without edit them.

Using the Capture Management

This task shows you how to use the Capture Management command.

This command allows you to:

- Associate one or several annotations or annotation planes with one or several captures.
- Disassociate one or several annotations from one or several captures.

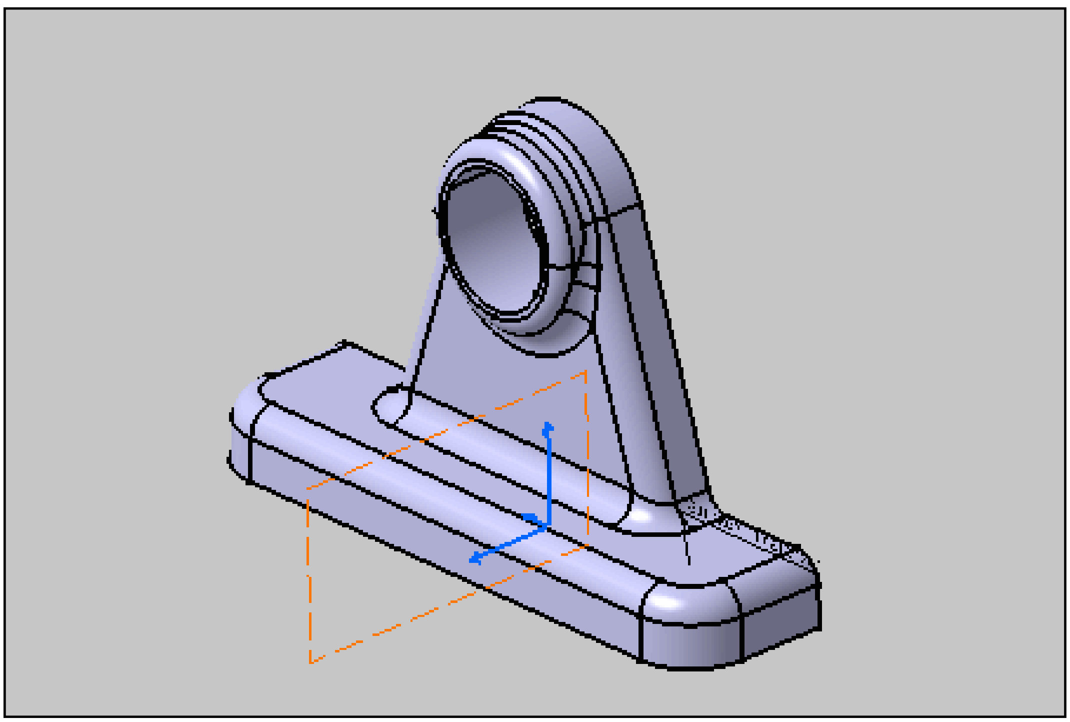
Annotations may be managed from existing captures in the document or its parent documents.

Open the Tolerancing_Annotations_05 CATPart document:

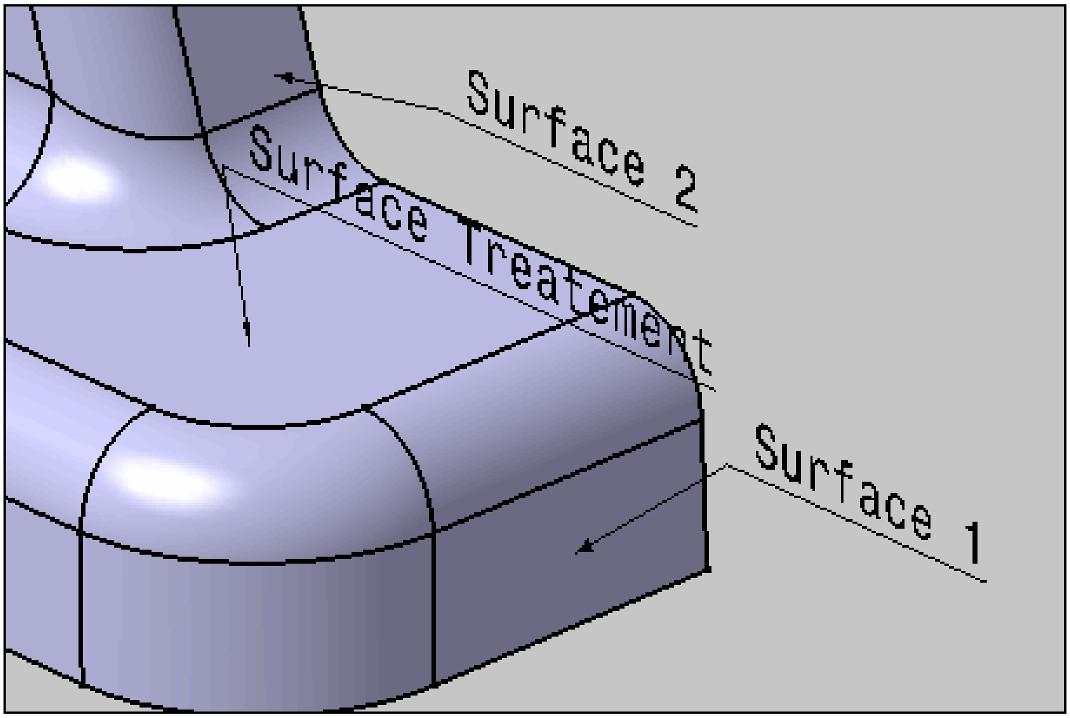
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Right-click the **All Texts** capture and select **Display Capture** command from the contextual menu.

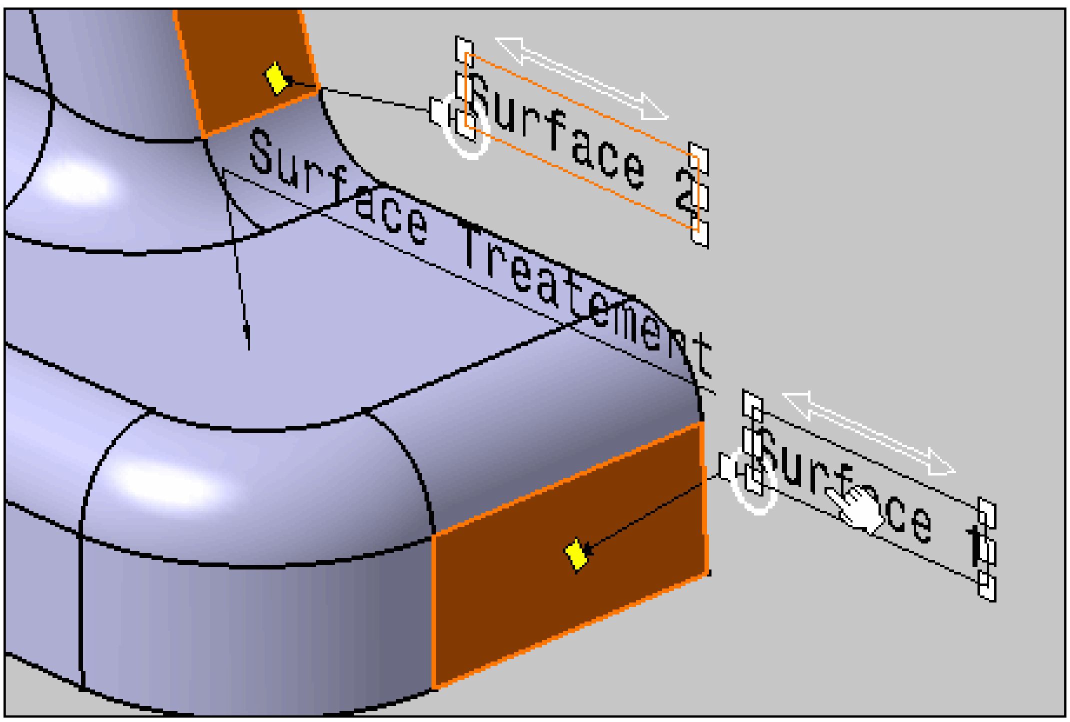
   The geometry is displayed according to the capture options.



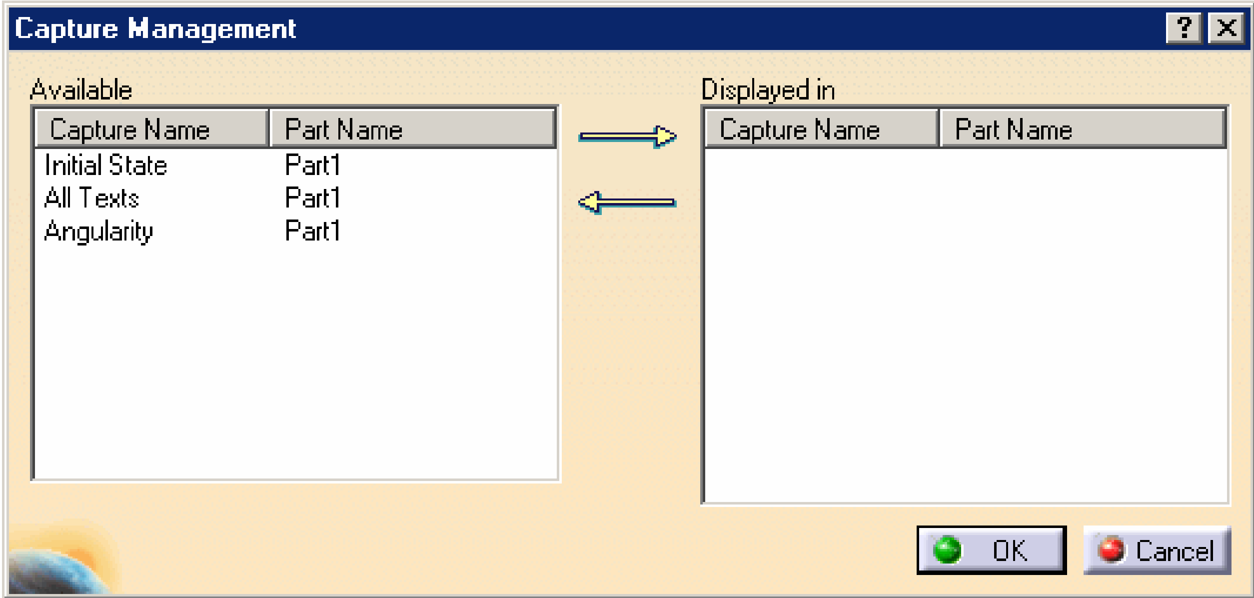
2. Create two new texts (See Creating an Annotation Text).



3. Select the two new texts, right-click one of them and select **Capture Management** command from the contextual menu.



The **Capture Management** dialog box appears.



The **Available** list displays the list of existing captures in the document and parent documents, where you can associate the selected annotations.
The **Displayed in** list displays the list of captures where ALL the selected annotations are still associated with.

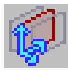
4. Select **All Texts** in the **Available** list and click the right-arrow.

5. Click **OK** in the **Capture Management** dialog box.

The new texts are now associated with the **All Texts** capture only. See Displaying a Tolerancing Capture.

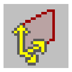
View/Annotation Planes

Use a View/Annotation Plane: view/annotation plane description.



Create a Projection View/Annotation Plane: click this icon and select a planar element.

Create a Section View/Annotation Plane: click this icon and select a planar element.



Create a Section Cut View/Annotation Plane: click this icon and select a planar element.

Create an Offset Section Cut/View: click this icon and define or select a cutting profile.

Create an Aligned Section Cut/View: click this icon and define or select a cutting profile.

Activate a View/Annotation Plane: double-click the desired view/annotation plane.

Edit View/Annotation Plane Properties: right-click the view/annotation plane, and select the **Properties** contextual command.

Manage View Associativity: right-click the view/annotation plane, and select the **Manage associativity** contextual command.

Using a View/Annotation Plane

In this section, you will learn about the different types of views. They are of two kinds:

- Views/annotation planes are specified around the geometry for automatically generating the corresponding views, sections and cuts of the 2D drawing.
- Extraction views are particular kinds of views, specifically aimed at preparing 2D extraction.

Views/Annotation Planes

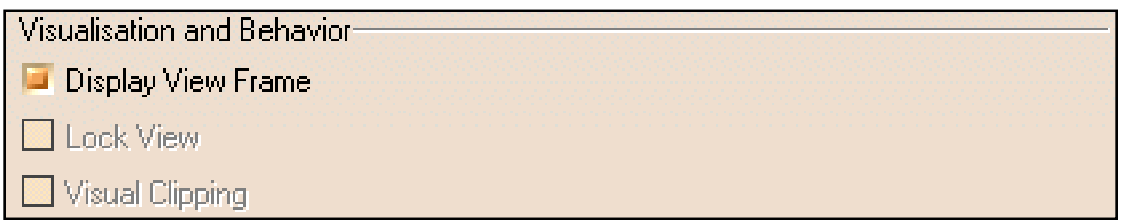
The views/annotation planes are by default displayed in the 3D with a dashed frame that is resized to always frame all the annotations that are linked to it.

When the view/annotation plane is active (the preferred view to receive newly created annotations), its origin and axis system is also displayed and the dashed frame is also resized to frame the axis system origin.

Note that when creating a new view/annotation plane by selecting a planar surface, the origin that is chosen is the part/product origin. If you want to choose the origin (and the axis orientation) of the view, you have to select an existing axis system while creating.

Note also that the position and orientation coordinates of a given annotation that are displayed in the **Position And Orientation** toolbar and in the **Position** region of the **Text** tab page of the annotation properties are expressed in the view axis system.

You can control the 3D display of the dashed frame of a view by modifying its **Display View Frame** property. See Editing View/Annotation Plane Properties.



You can also control the display of the active axis system by using the option **Current view axis display**. For more information, see 3D Annotations Infrastructure settings and/or Functional Tolerancing & Annotation settings in **Tools -> Options**.

Three types of annotations planes are available:

- Projection View/Annotation Plane
- Section View/Annotation Plane
- Section Cut View/Annotation Plane

Projection View/Annotation Plane

A projection view/annotation plane allows you to manage 3D annotations:

- Located in planes both parallel to this view/annotation plane and in the background and foreground spaces bounded by this view/annotation plane (or in any plane of the direction of planes defined by this view/annotation plane),
- Related to the geometry finding an intersection with this view/annotation plane,
- Lying on/belonging to this view/annotation plane.

This view/annotation plane allows you to specify a particular view/annotation plane for generating embedded 2D front/projection views, in the Generative Drafting workbench, during the 2D extraction of the 3D part and of the 3D annotations.

Annotations can be translated along the z axis of its local coordinate system.
Negative and positive z values can be used to define the translation, since the projection view/annotation plane will be used for the extraction of front views in the Generative Drafting workbench.

See Creating a Projection View/Annotation Plane.

Section View/Annotation Plane

A section view/annotation plane allows you to manage 3D annotations:

- Located in planes both parallel to this view/annotation plane and in the background space bounded by this view/annotation plane,
- Related to the geometry finding an intersection with this view/annotation plane,
- Lying on/belonging to this view/annotation plane.

This view allows you to specify a particular annotation for generating embedded 2D section views, in the Generative Drafting workbench, during the 2D extraction of the 3D part and of the 3D annotations.

Annotations can be translated along the z axis of its local coordinate system.
Only negative z values can be used to define the translation, since the section view/annotation plane will be used for the extraction of section views in the Generative Drafting workbench.

See Creating a Section View/Annotation Plane.

Section Cut View/Annotation Plane

A section cut view/annotation plane allows you to manage 3D annotations:

- Only related to the geometry finding an intersection with this view/annotation plane,
- Related to the geometry finding an intersection with this view/annotation plane.

This view allows you to specify a particular annotation for generating embedded 2D section views, in the Generative Drafting workbench, during the 2D extraction of the 3D part and of the 3D annotations.

Annotations cannot be translated along the z axis of its local coordinate system (z=0), since the section cut view/annotation plane will be used for the extraction of section cuts in the Generative Drafting workbench.

See Creating a Section Cut View/Annotation Plane.

Extraction Views

Extraction views are particular kinds of views. They specifically aimed at preparing the following types of views for 2D extraction:

- aligned section views/section cuts
- offset section views/section cuts

Extraction views are made up of several annotation planes (of the same type). You can create annotations in each view/annotation plane making up the extraction view. You will then be able to extract this extraction view to 2D in the Generative Drafting workbench, as well as all annotations defined in each component section view.

Extraction views, no matter their type, use a cutting profile as cutting plane.

Aligned section views/section cuts

An aligned section view/aligned section cut is created from a cutting profile defined from non parallel planes. In order to include in a section certain angled elements, the cutting plane may be bent so as to pass through those features. The plane and feature are then imagined to be revolved into the original plane.

Aligned section views are made up of several section views/annotation planes, as described in Section View/Annotation Plane above.

Aligned section cuts are made up of several section cut views/annotation planes, as described in Section Cut View/Annotation Plane above.

Offset section views/section cuts

Offset section views/offset section cuts let you show several features that do not lie in a straight line by offsetting or bending the cutting plane, which is often desirable when sectioning through irregular objects.

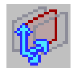
Offset section views are made up of several section views/annotation planes, as described in Section View/Annotation Plane above.

Offset section cuts are made up of several section cut views/annotation planes, as described in Section Cut View/Annotation Plane above.

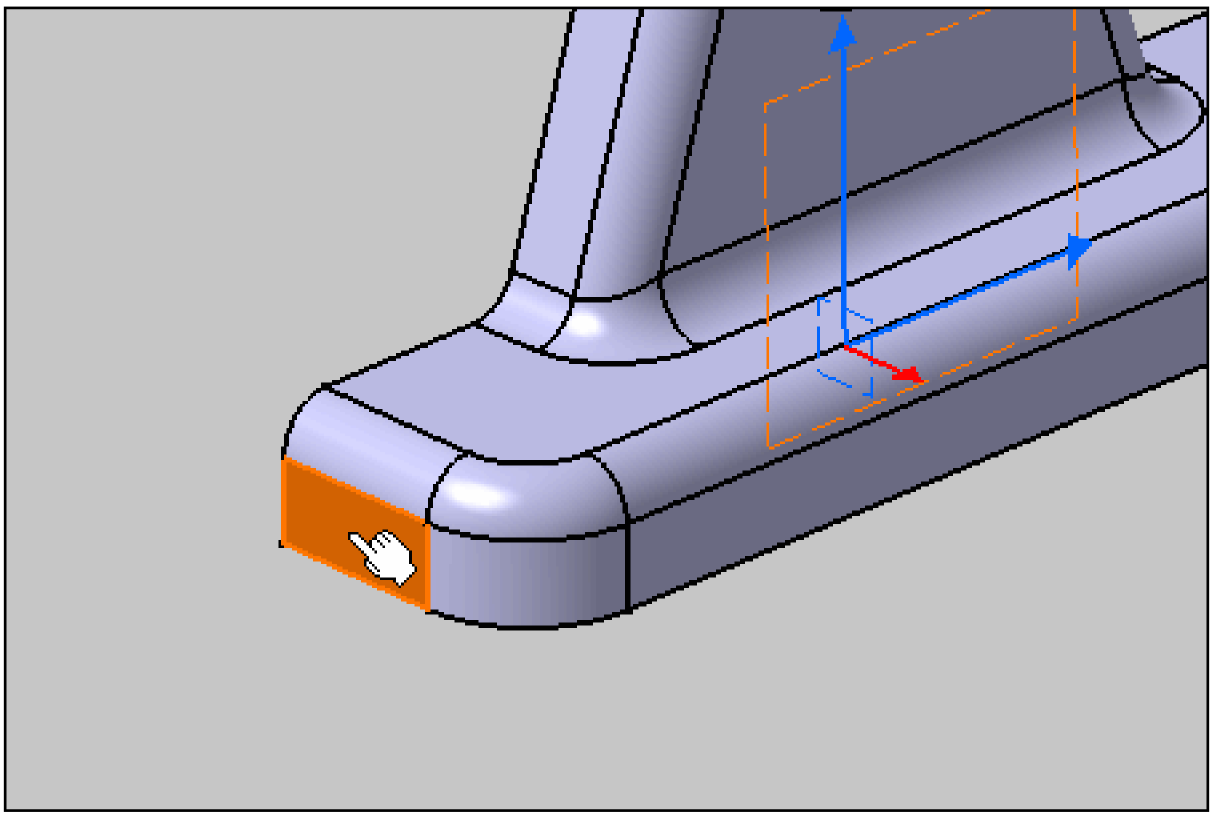
Creating a Projection View/Annotation Plane

This task shows you how to create a projection view /annotation plane.
See Using a View for more information.
See also Creating a Section View/Annotation Plane, Creating a Section Cut View/Annotation Plane.

Open the Common_Tolerancing_Annotations_01 CATPart document.

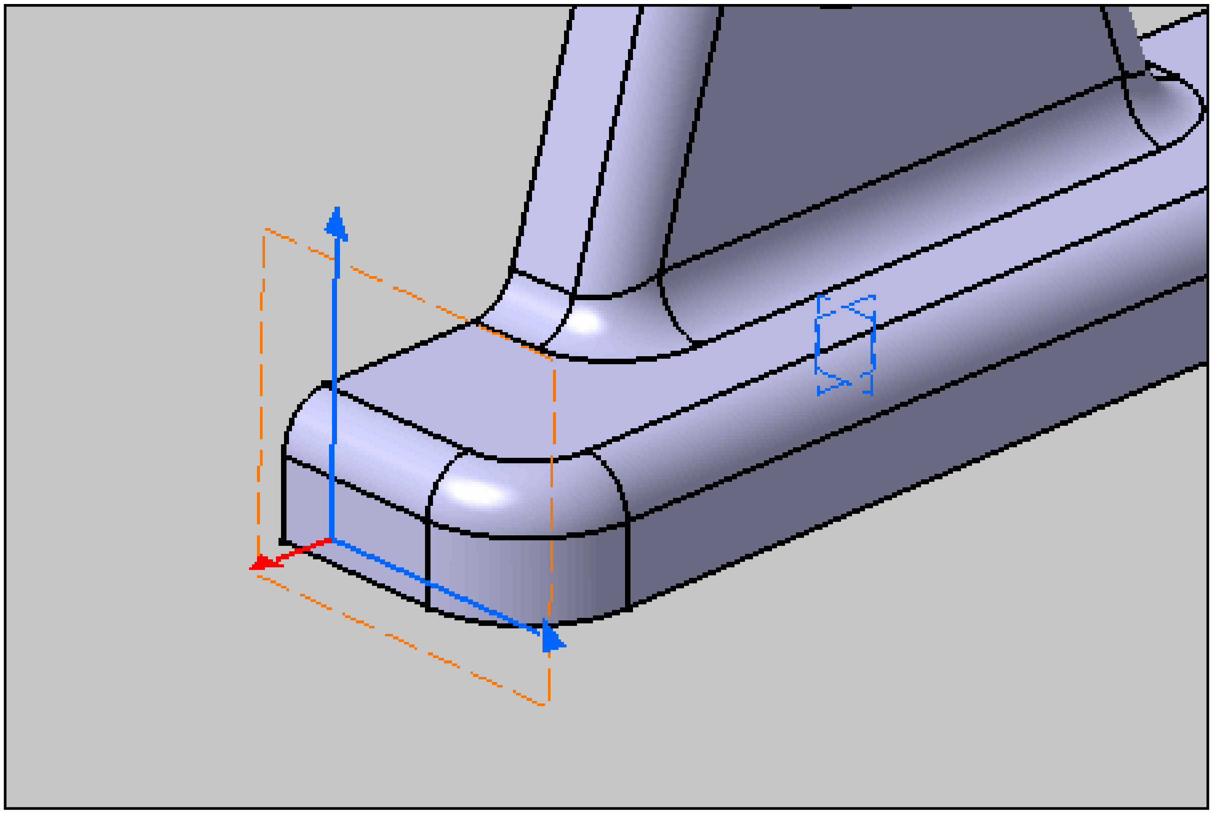
1. Click the **Projection View** icon: 

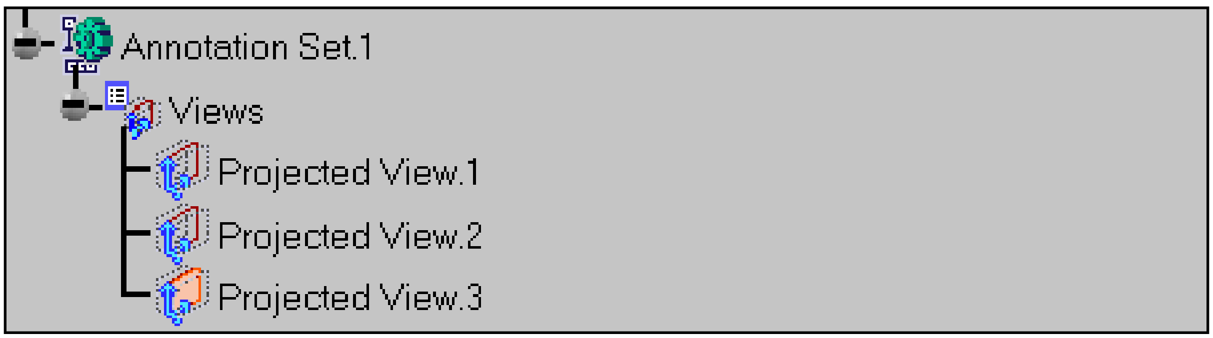
2. Select the face as shown.



You have to select a planar element only to perform this command.

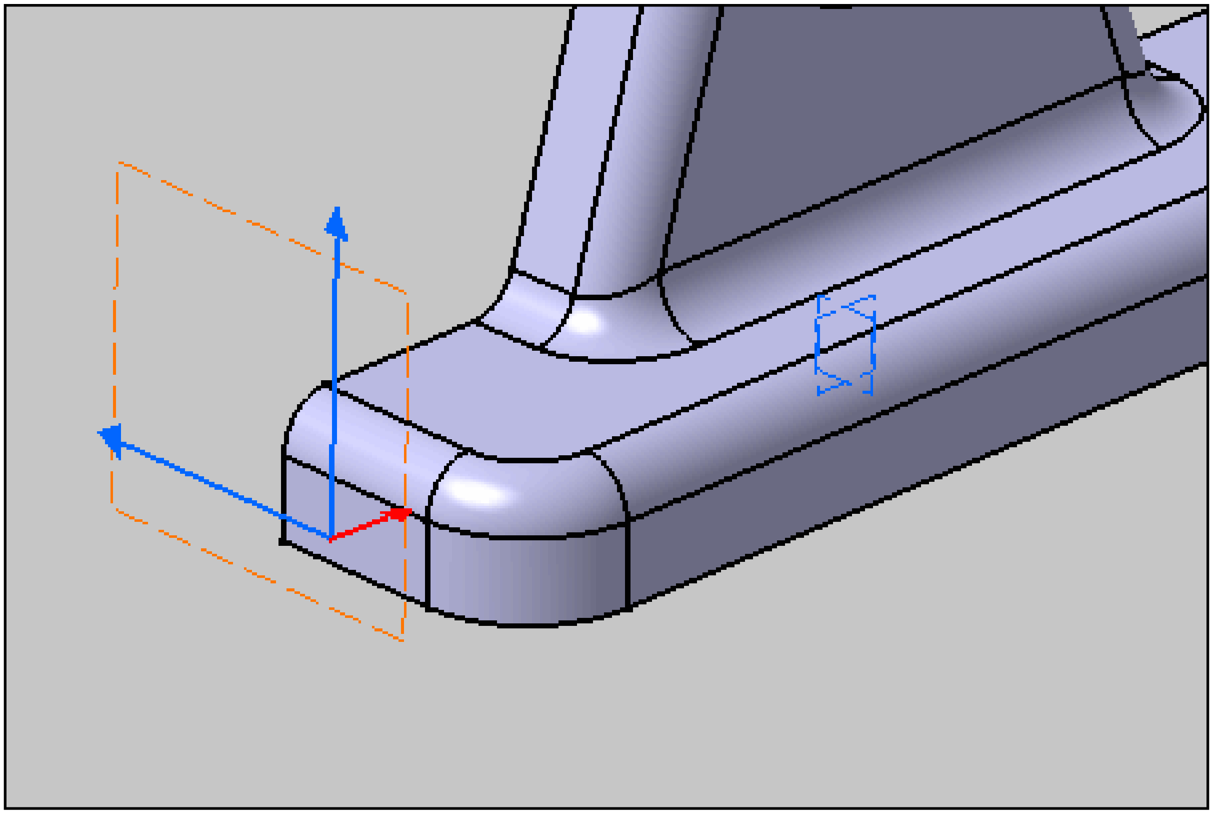
The projection view is created.
Projection views are represented by a blue reference axis, its normal axis is red until you create an annotation, and are identified as **Projection View.3** in the specification tree.





3. Right-click the annotation plane in the geometry or in the specification tree and select the **Invert Normal** contextual menu.

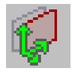
The projection view normal is reversed.



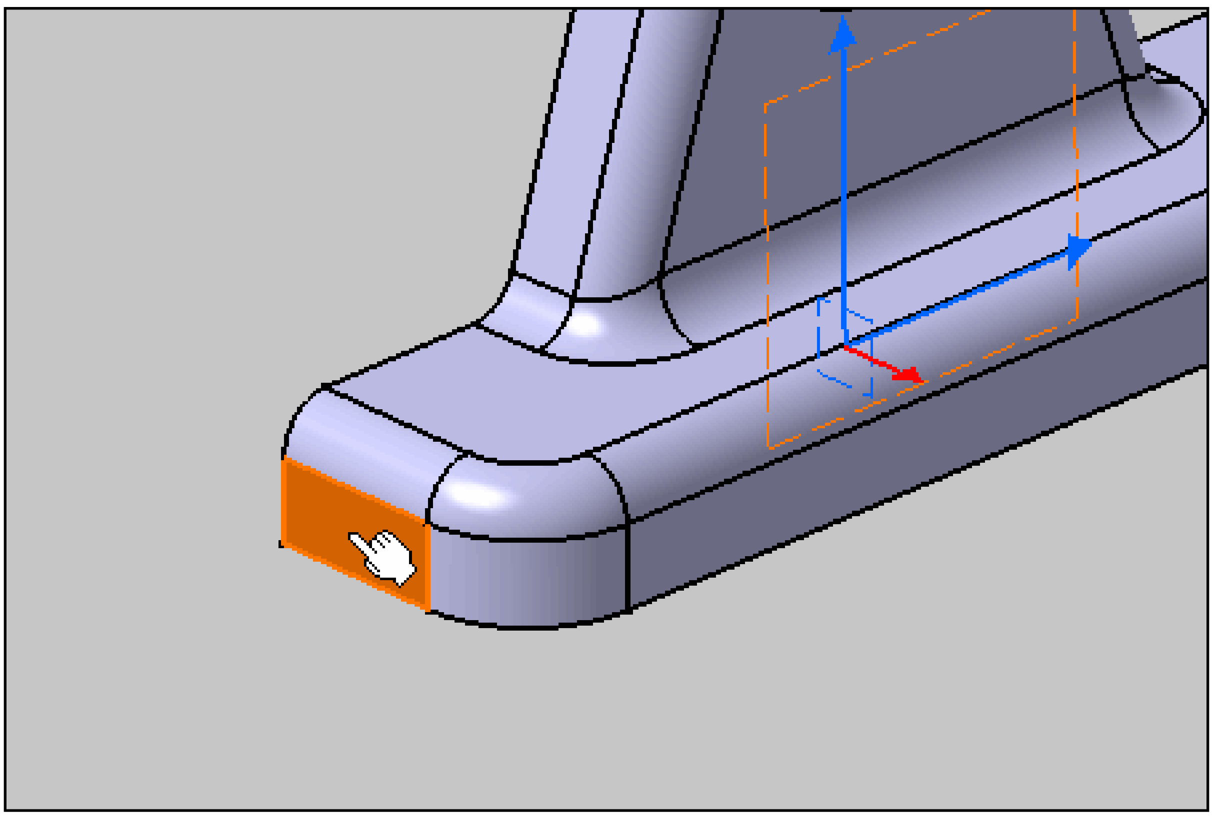
Creating a Section View/Annotation Plane

This task shows you how to create a section view /annotation plane.
See Using a View for more information.
See also Creating a Projection View/Annotation Plane, Creating a Section Cut View/Annotation Plane.

Open the Common_Tolerancing_Annotations_01 CATPart document.

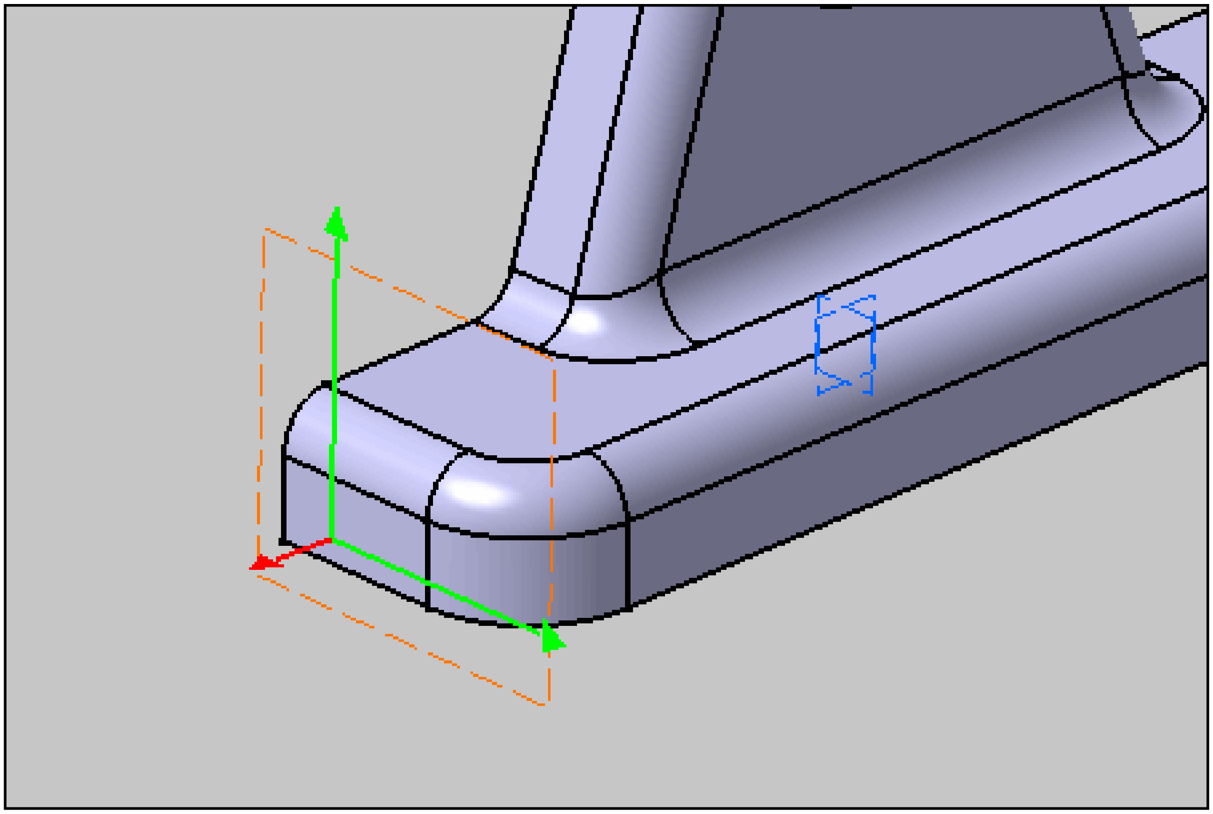
1. Click the **Section View** icon: 

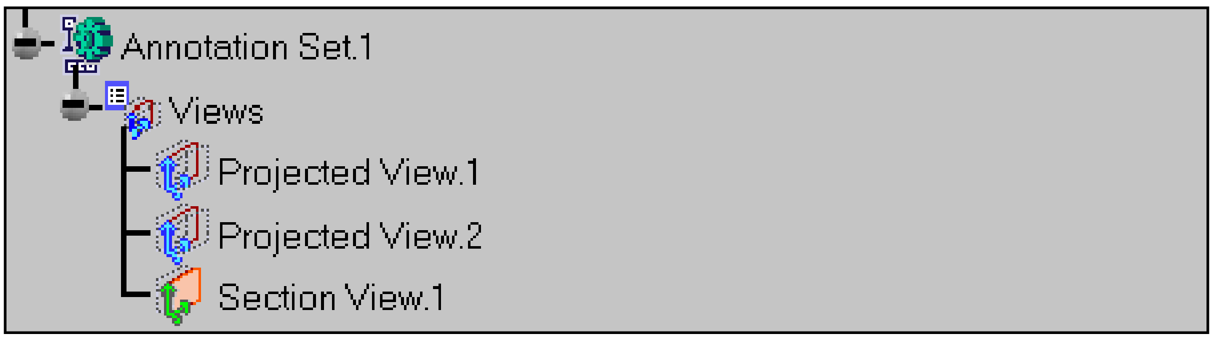
2. Select the face as shown.



You have to select a planar element only to perform this command.

The section view is created.
Section views are represented by a green reference axis, its normal axis is red until you create an annotation, and are identified as **Section View.1** in the specification tree.

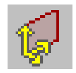



Annotation Set.1
Views
Projected View.1
Projected View.2
Section View.1

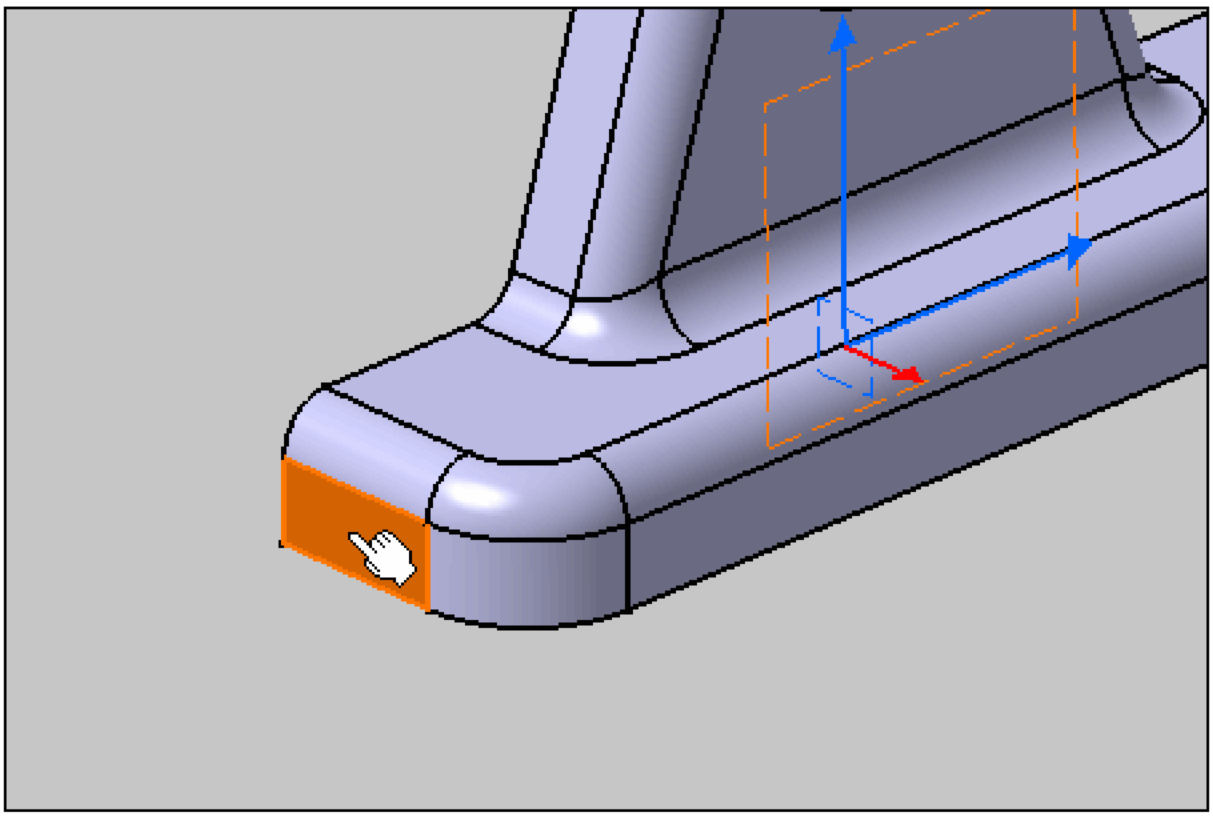
Creating a Section Cut View/Annotation Plane

This task shows you how to create a section cut view /annotation plane.
See Using a View for more information.
See also Creating a Projection View/Annotation Plane, Creating a Section View/Annotation Plane.

Open the Common_Tolerancing_Annotations_01 CATPart document.

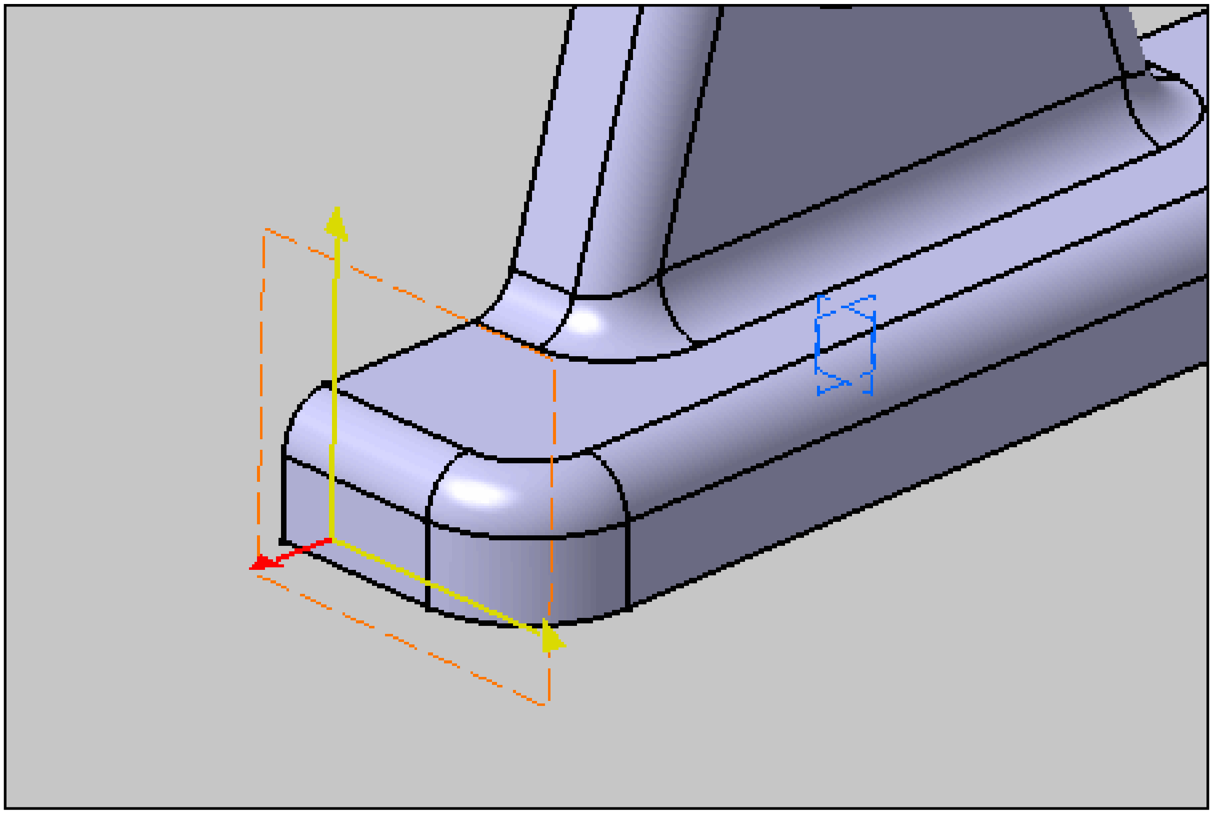
1. Click the **Section Cut View** icon: 

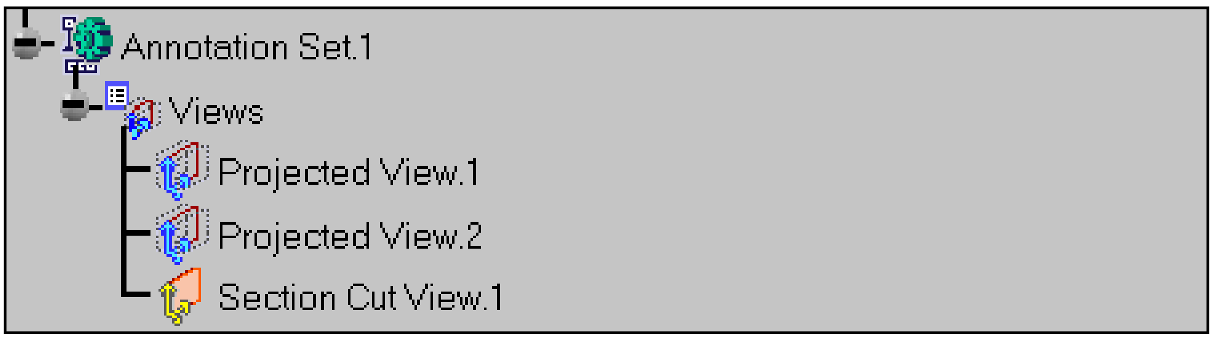
2. Select the face as shown.



You have to select a planar element only to perform this command.

The section cut view is created.
Section views are represented by a yellow reference axis, its normal axis is red until you create an annotation, and are identified as **Section Cut View.1** in the specification tree.




Annotation Set.1
Views
Projected View.1
Projected View.2
Section Cut View.1

Creating an Aligned Section View/Section Cut

This task shows you how to create an aligned section view or an aligned section cut using a cutting profile as cutting planes. Aligned section views/section cuts are specifically aimed at preparing views for 2D extraction. See Using a View for more information.

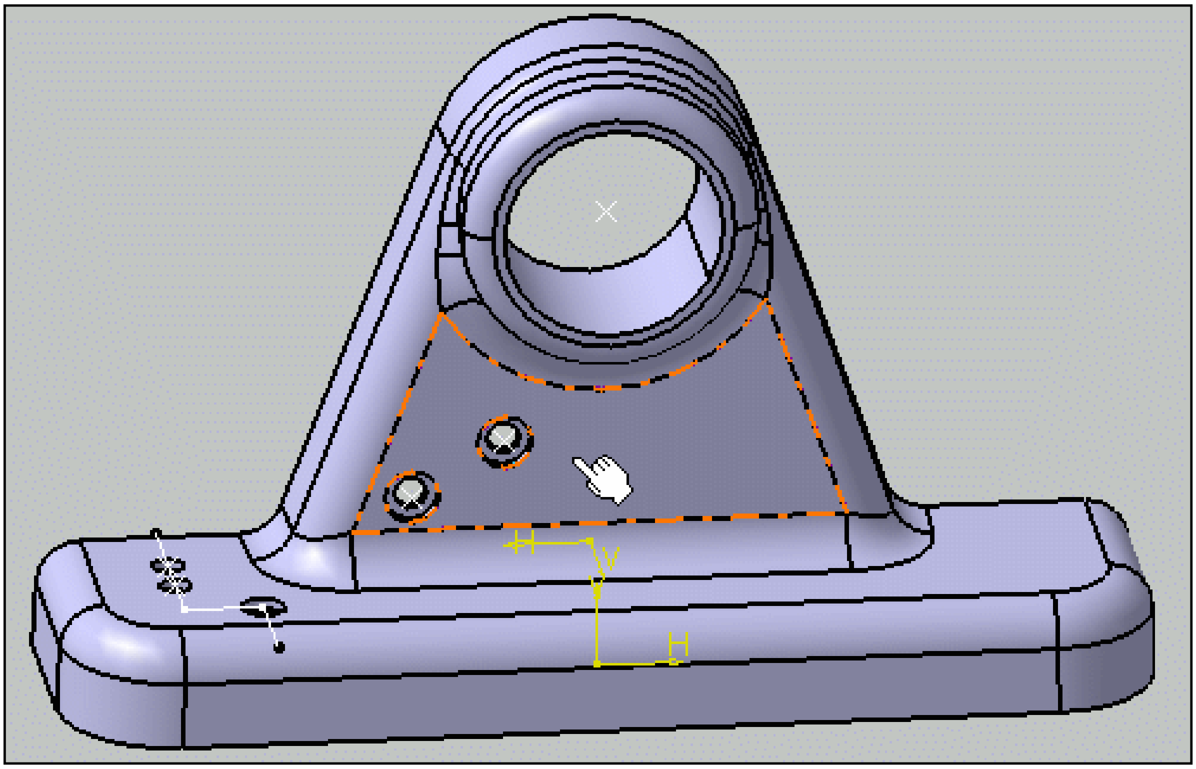
An aligned section view/aligned section cut is created from a cutting profile defined from non parallel planes.

Open the Common_Tolerancing_Annotations_03 CATPart document.

1. Click the **Aligned Section View / Section Cut** icon: The **Section View Creation** dialog box is displayed.
2. Specify the type of view that you want to create: **Section View** or **Section Cut**. For the purpose of this scenario, select **Section View**.
3. Click the sketch icon to sketch a new profile that will be used as cutting plane.

Note that, at this stage, you can also select an existing sketch, if one is available; in this case, you can only select a sketch which is valid for the type of view to be created. For more information on using this method, you can refer to Creating an Offset Section View/Section Cut: the procedure is similar when creating aligned section views/section cuts.

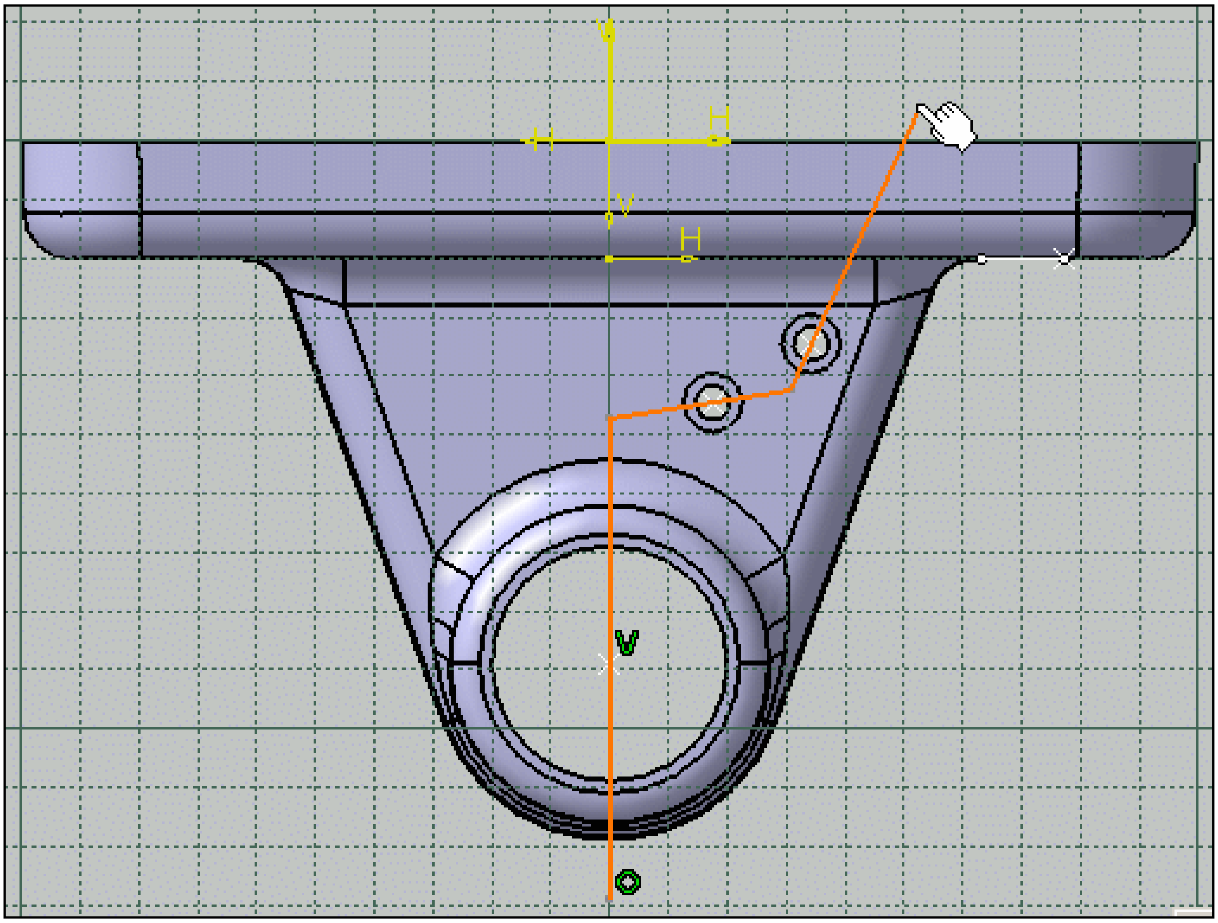
4. Select a planar surface.



The application switches to the Sketcher workbench to let you sketch the cutting profile.

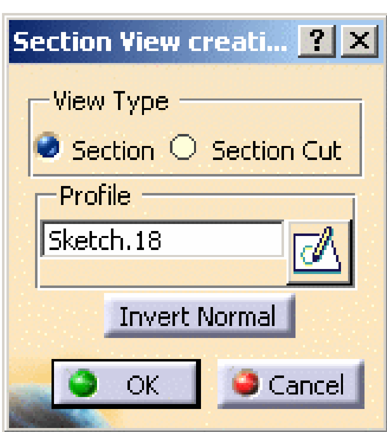
5. Click the **Profile** icon: 

6. Sketch your cutting profile as shown here.



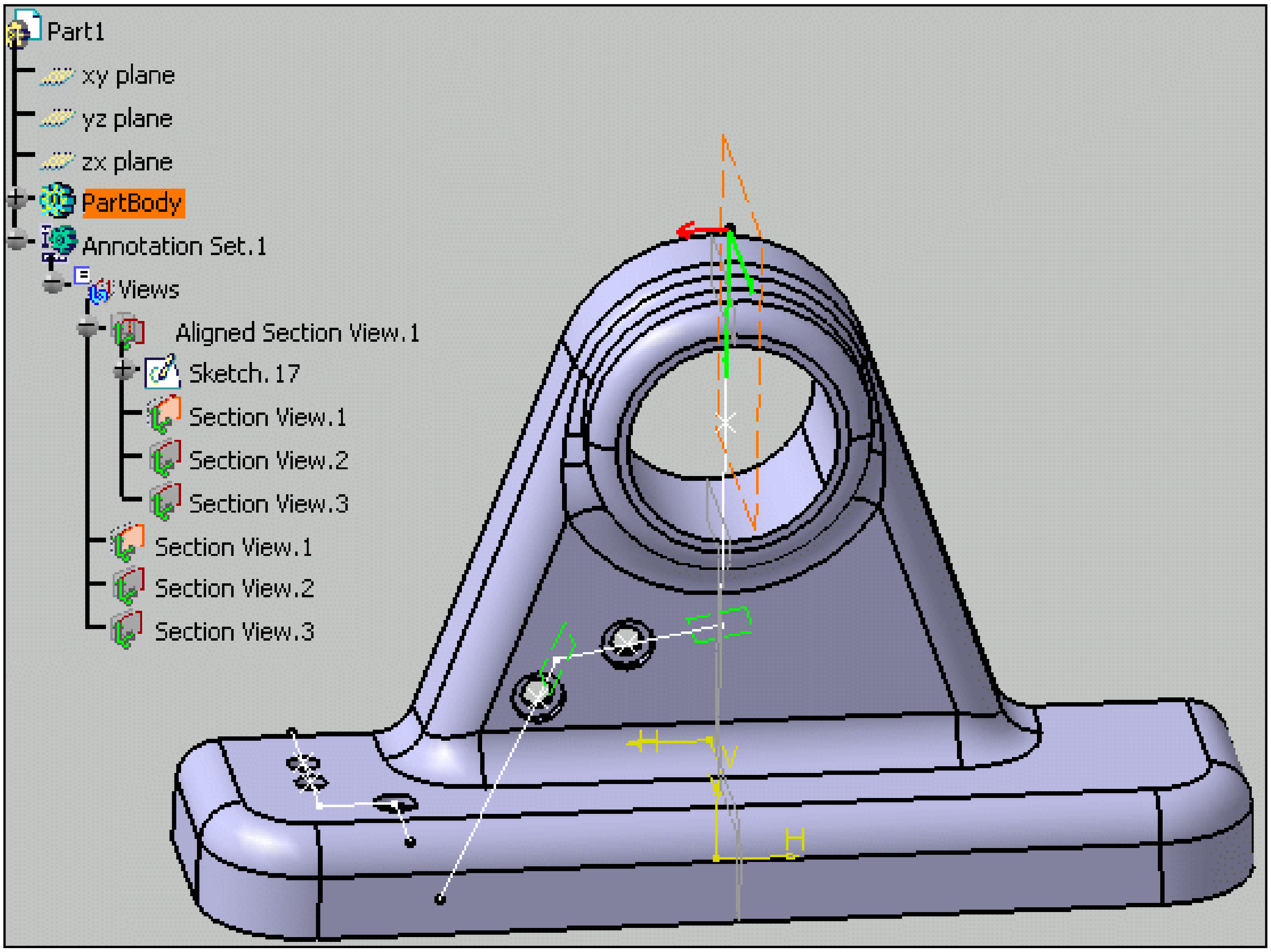
You can constrain the sketch lines in order to ensure their position according to some features of the part.

7. When you are done, click the **Exit workbench** icon  to exit the Sketcher workbench. The newly created sketch, Sketch.16, is now selected in the Profile field of the Section View Creation dialog box.



Clicking the **Invert Normal** button lets you invert the normal of the aligned section view: this actually inverts the normal of the section views/annotation planes that make up the aligned section view.

8. Click **OK**. The aligned section view is now created and listed in the specification tree; it cannot be activated. It is made up of three distinct section views/annotation planes, each of which can be activated and behaves like a regular section view/annotation plane. Each view is associative to the sketched line that defines it.



You can now start creating annotations in each section view of the aligned section view. If you then extract the aligned section view to 2D in the Generative Drafting workbench, all the annotations defined in each component view will be generated.

9. Optionally, right-click Aligned Section View.1 in the specification tree and select **Invert Normal** in the contextual menu if you want to invert the normal of the aligned section view: this actually inverts the normal of all the section views/annotation planes that make up the aligned section view.

- The **Invert Normal** contextual command is only available if there is no annotation attached to any component view of the aligned section view/section cut.
- You cannot perform the following operations for the component sections views/section cuts of an aligned section view/section cut: Delete, Invert Normal and Manage Associativity.

Creating an Offset Section View/Section Cut

This task shows you how to create an offset section view or an offset section cut using a cutting profile as cutting plane. Offset section views/section cuts are specifically aimed at preparing views for 2D extraction. See Using a View for more information.

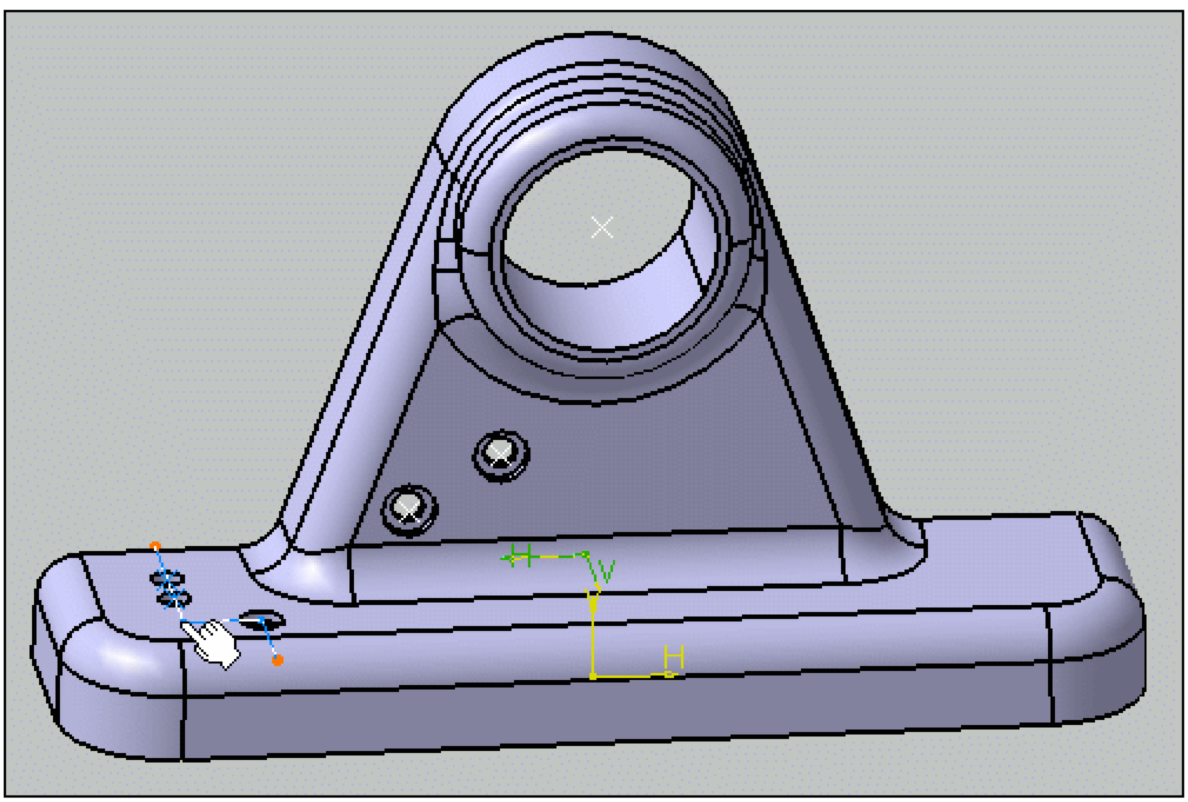
Offset section views/offset section cuts let you show several features that do not lie in a straight line by offsetting or bending the cutting plane, which is often desirable when sectioning through irregular objects.

Open the Common_Tolerancing_Annotations_03 CATPart document.

1. Click the **Offset Section View / Section Cut** icon: The Section View Creation dialog box is displayed.
2. Specify the type of view that you want to create: **Section View** or **Section Cut**. For the purpose of this scenario, select **Section Cut**.
3. Select the profile that will be used as cutting plane. You can only select a sketch which is valid for the type of view to be created.

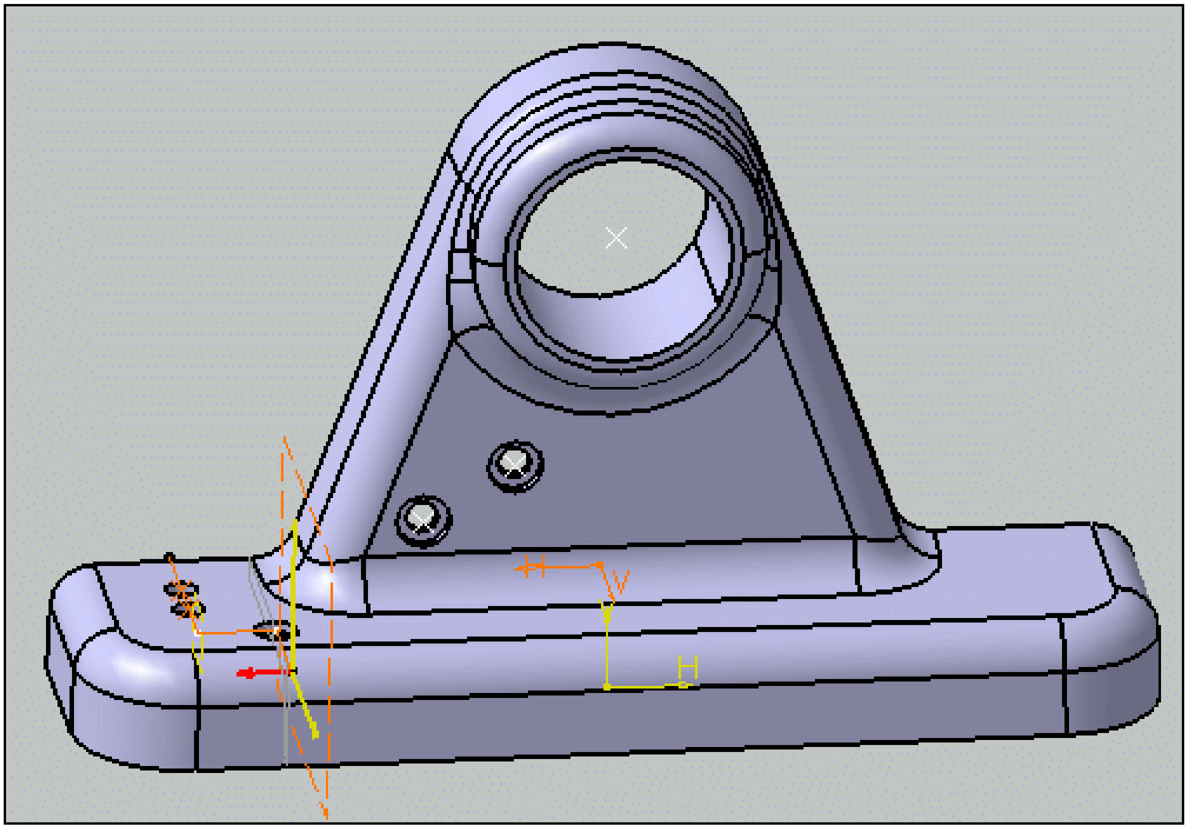
Note that, at this stage, you can also click the sketch icon to sketch a new profile to use as cutting plane.

For more information on using this method, you can refer to Creating an Aligned Section View/Section Cut: the procedure is similar when creating offset section views/section cuts.

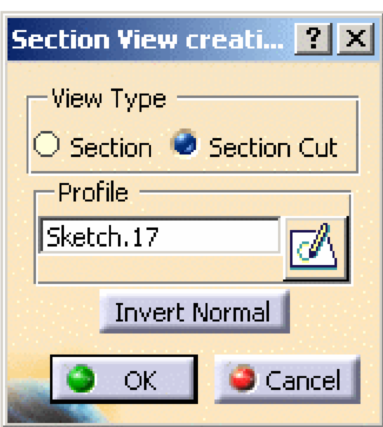


The offset section cut is previewed: it is made up of two distinct section cut views/annotation planes.

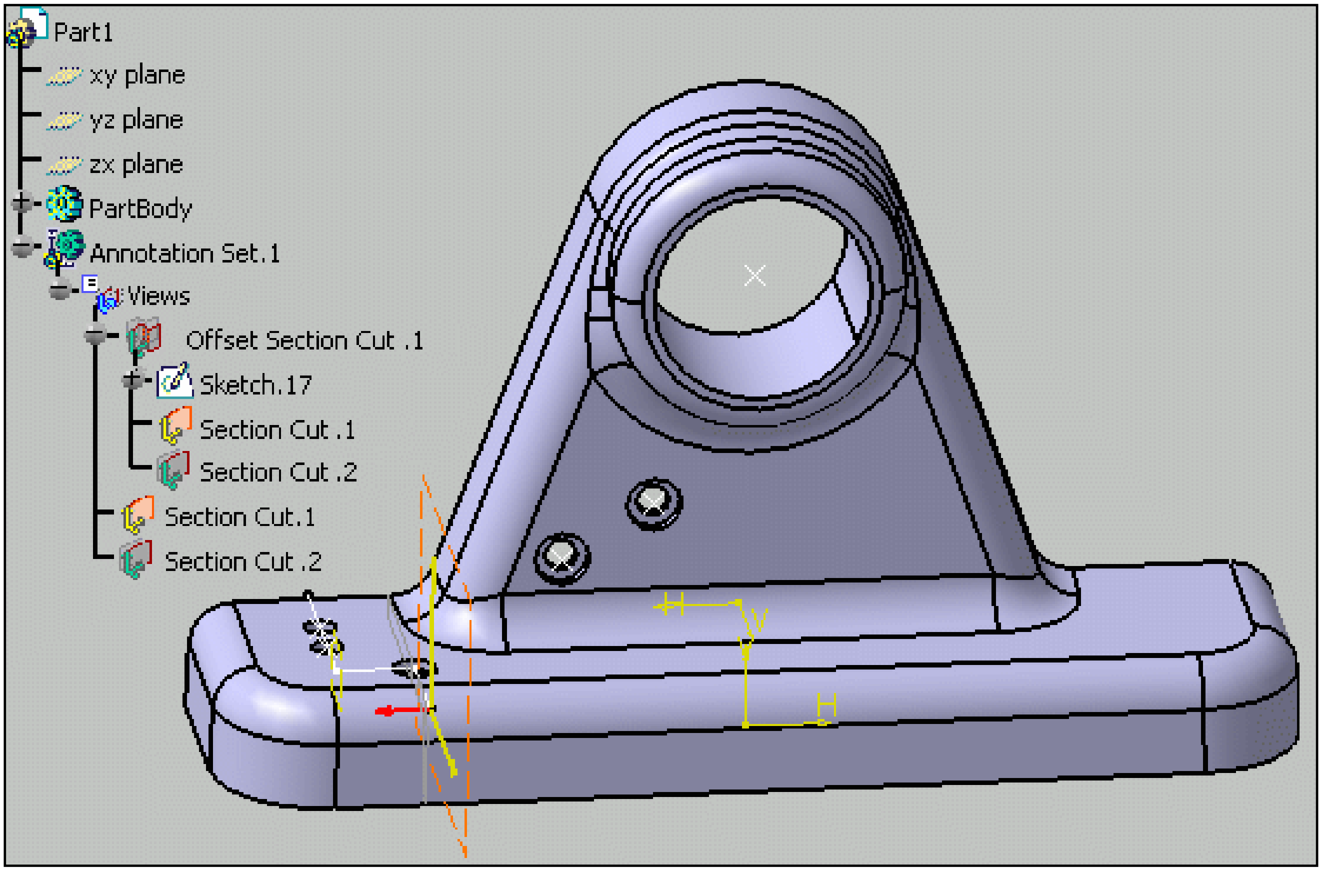
4. Click the **Invert Normal** button to invert the normal of the offset section cut: this actually inverts the normal of the two section cut views/annotation planes that make up the offset section cut.



5. Click **OK** in the Section View Creation dialog box.



The offset section cut is now created and listed in the specification tree; it cannot be activated. Each section cut/annotation plane that make it up can be activated and behaves like a regular section cut/annotation plane. Each section cut is associative to the sketched line that defines it.



You can now start creating annotations in each section cut of the offset section cut. If you then extract the view to 2D in the Generative Drafting workbench, all the annotations defined in each component view will be generated.

6. Optionally, right-click Offset Section Cut.1 in the specification tree and select **Invert Normal** in the contextual menu if you want to invert the normal of the offset section cut: this actually inverts the normal of all the section cuts/annotation planes that make up the offset section cut.

You cannot perform the following operations for the component section views/section cuts of an offset section view/section cut: Delete, Invert Normal and Manage Associativity.

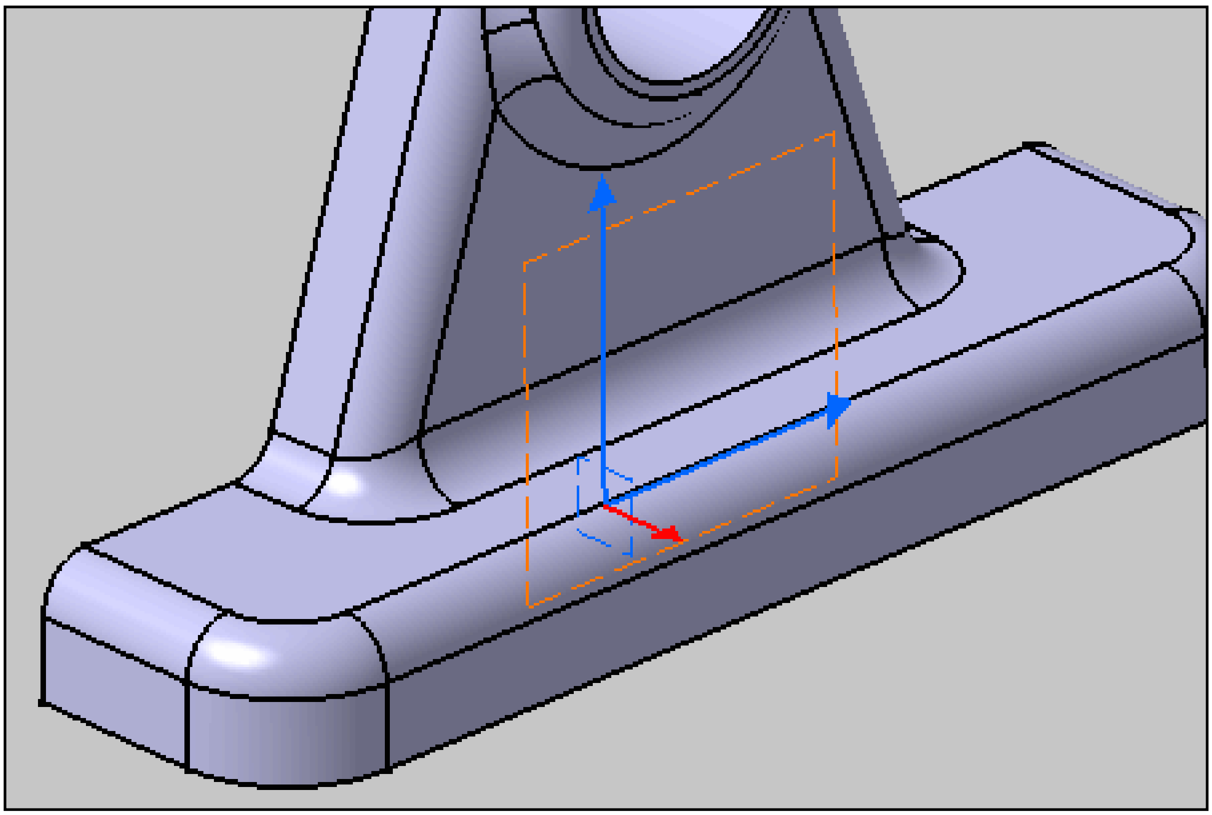
Activating a View/Annotation Plane

When a tolerancing set includes several annotation planes, to activate a plane, you either double-click it or use the **Activate View** contextual command.

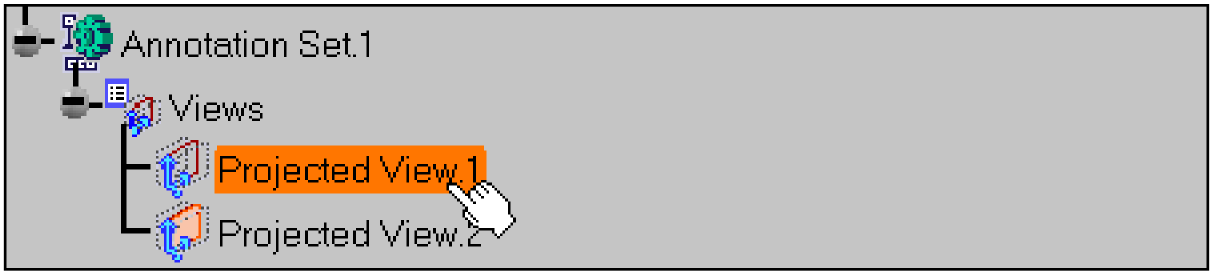
Open the Common_Tolerancing_Annotations_01 CATPart document.

The active annotation plane is orange-colored in the specification tree.

The active annotation plane frame is orange-colored in the geometry and its reference axis appears.

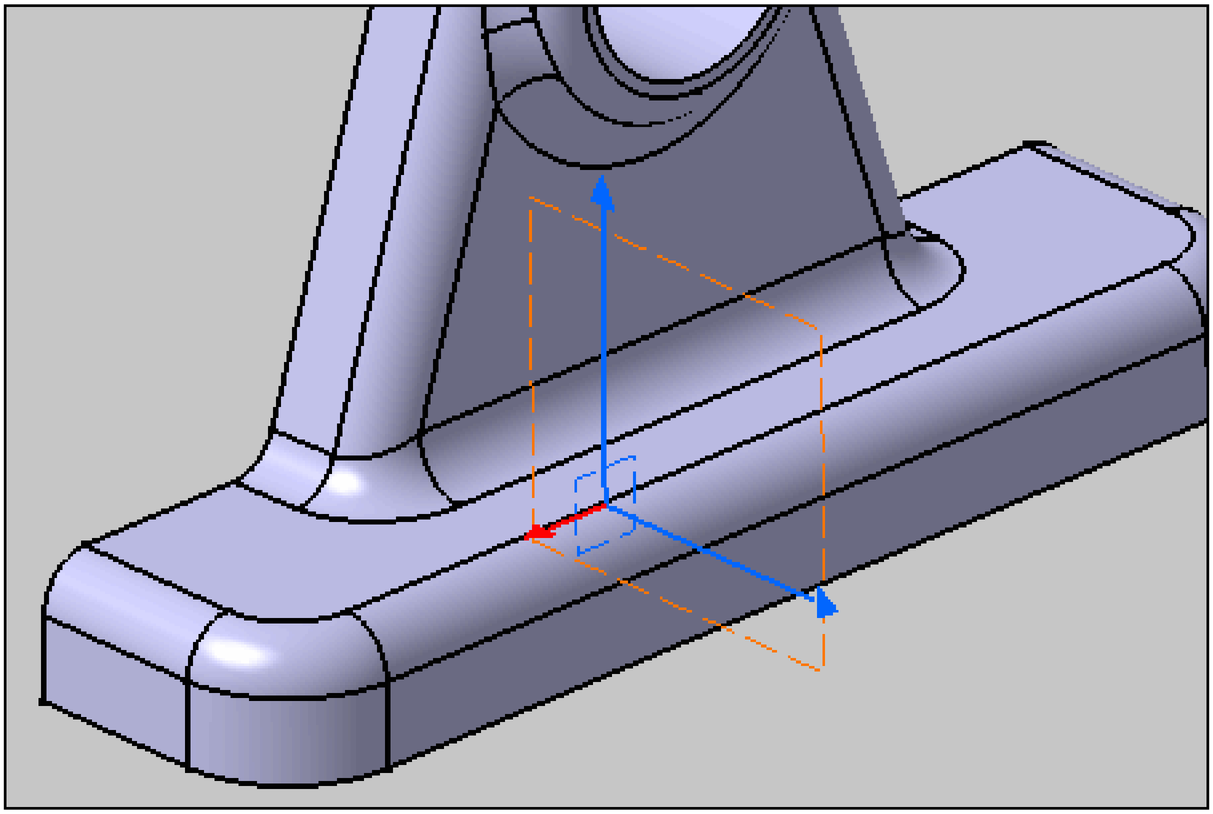


1. Double-click the **Projection View.1** annotation plane.



The **Projection View.1** is orange-colored in the specification tree.

The **Projection View.2** frame is orange-colored in the geometry and its reference axis appears.

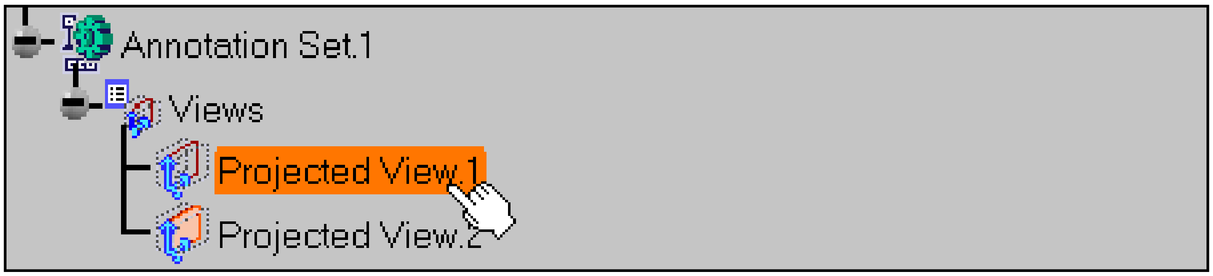


Editing View/Annotation Plane Properties

This task shows you how to rename an annotation plane and hide its frame.

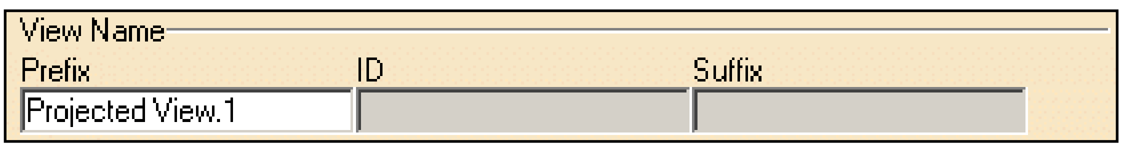
Open the Common_Tolerancing_Annotations_01 CATPart document.

1. Right-click the **Projection View.1** annotation plane and select the **Properties** contextual command.

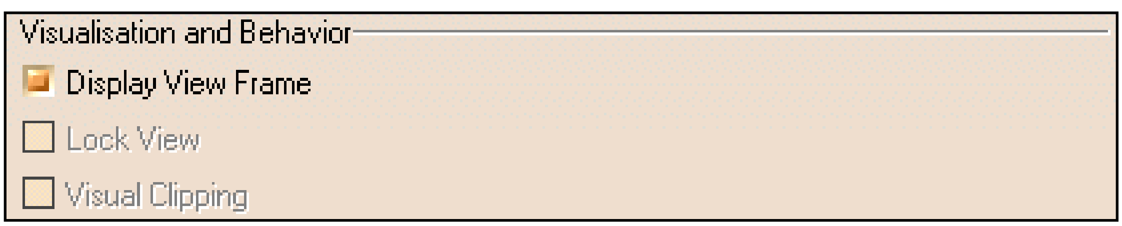


2. Select the **View** tab in the **Properties** dialog box which is displayed.

To rename the annotation plane, enter the new name in the **Prefix** field.



Check/uncheck the **Display view frame** option to show/hide the view/annotation plane frame.



3. Click **OK** to confirm and close the dialog box.

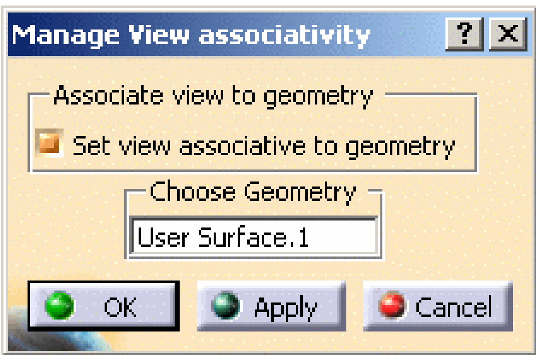
Managing View/Annotation Plane Associativity

This task shows you how to manage view/annotation plane associativity, by changing the definition plane of a view.

When views are associative to the geometry, any modification applied to the geometry or to the axis system is reflected in the view definition.

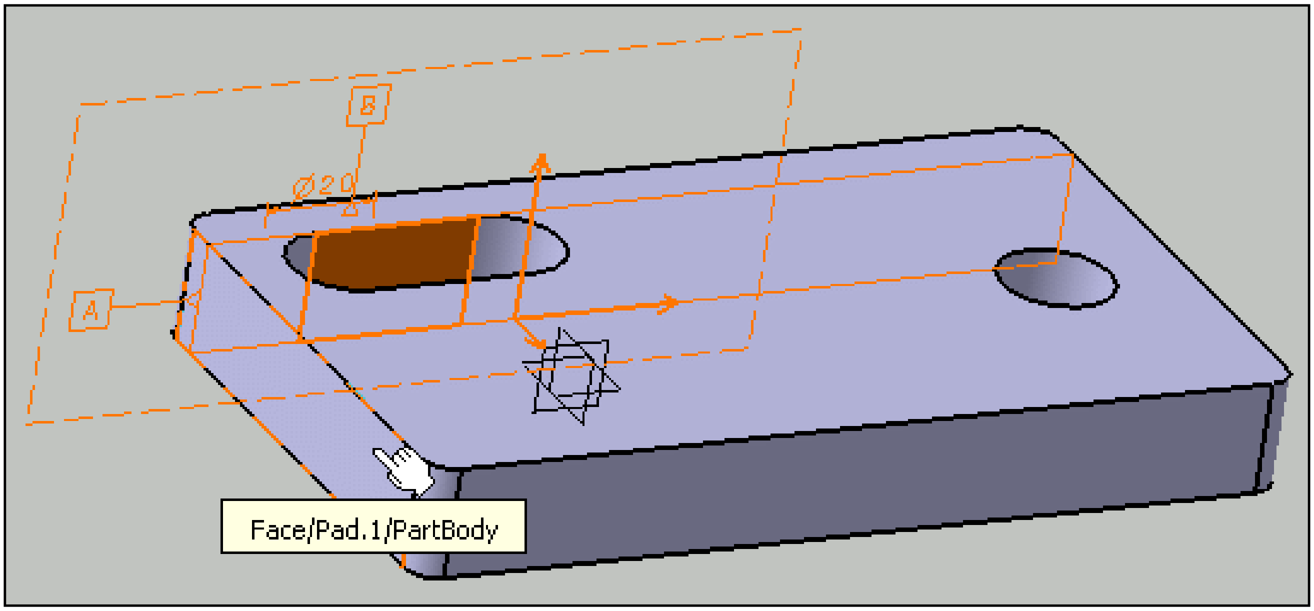
Open the Common_Tolerancing_Annotations_02 CATPart document.

1. Right-click the **Section View.1** annotation plane, and select the **Manage associativity** contextual command. The **Manage View Associativity** dialog box is displayed, indicating that the view is currently associative to User Surface.1.

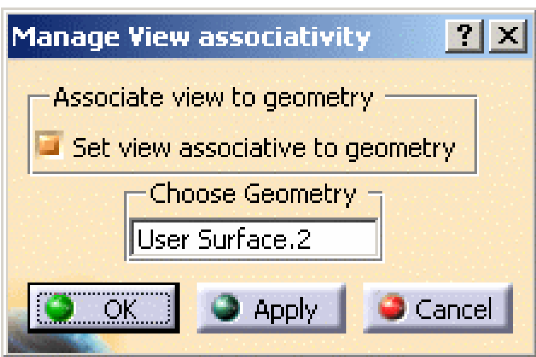


2. You have two possibilities:
   - If you want to disassociate the view from the geometry, uncheck the **Set view associative to geometry** field. In this case, you will then be able to modify the geometry or the axis system without changing the view definition.
   - If you want to associate the view to another geometry, select a planar face or an axis system.

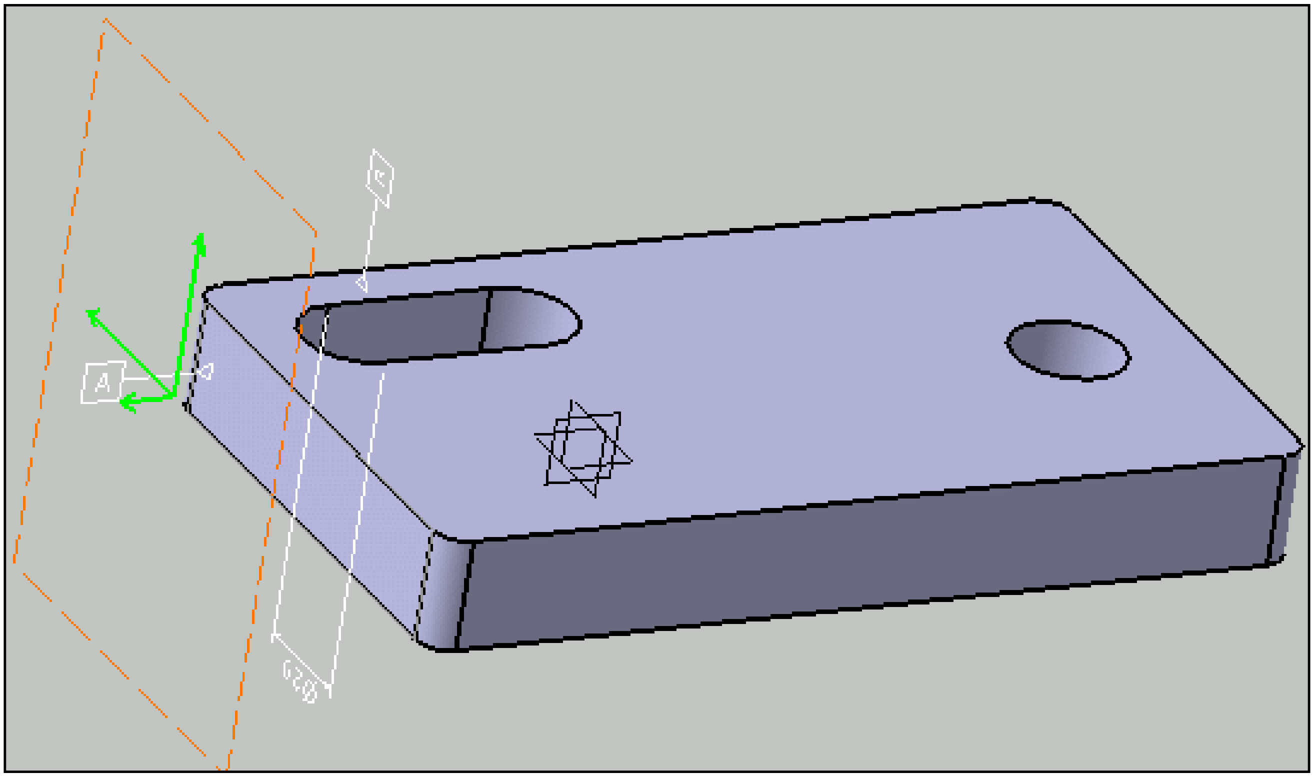
   For the purpose of this scenario, select the face as shown below to associate the view to another geometry.



The **Manage View Associativity** dialog box is updated, indicating that the view will now be associative to User Surface.2.



3. Click **OK**. The Section View.1 annotation plane is now associative to the specified surface. If you move the view definition plane or modify the axis system, the view will be re-defined accordingly.



Limitation

When extracted to 2D (using the **View from 3D** command in the Generative Drafting workbench), views from 3D are currently not associative to the geometry of the 3D view. So, if you modify the geometry of a 3D view, the definition of the corresponding 2D view will not be modified at the next update, even if the 3D view is associative to the geometry. This limitation should be fixed in an upcoming release.

Migrating Version 4 Data

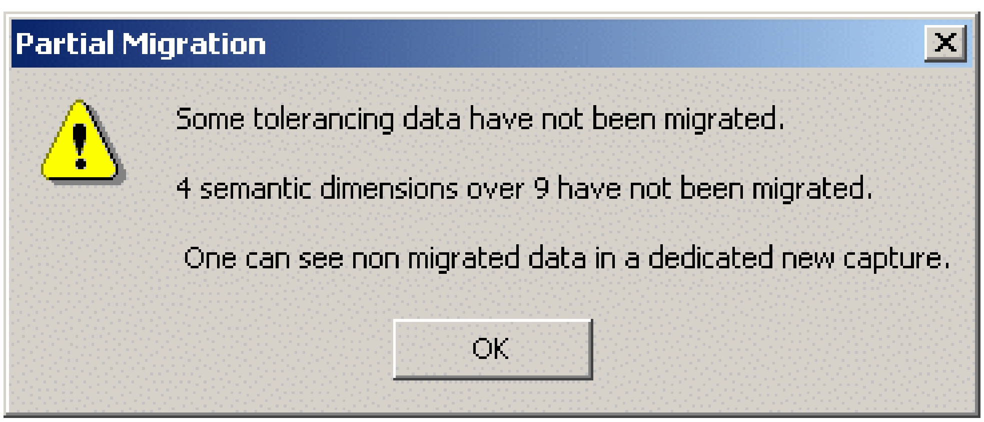
This task shows how to migrate Version 4 3D dimensioning, tolerancing and annotation data to Version 5 without migrating the geometry. See Version 4 Functional Dimensioning &Tolerancing Data Migration.

Launch the *Product Functional Tolerancing & Annotation* workbench (**Start -> Mechanical Design -> Product Functional Tolerancing & Annotation**). Select **Product.1** in the specification tree, and click **Insert -> Existing Component**. In the dialog box, select the V4data.model document and click **Open**.

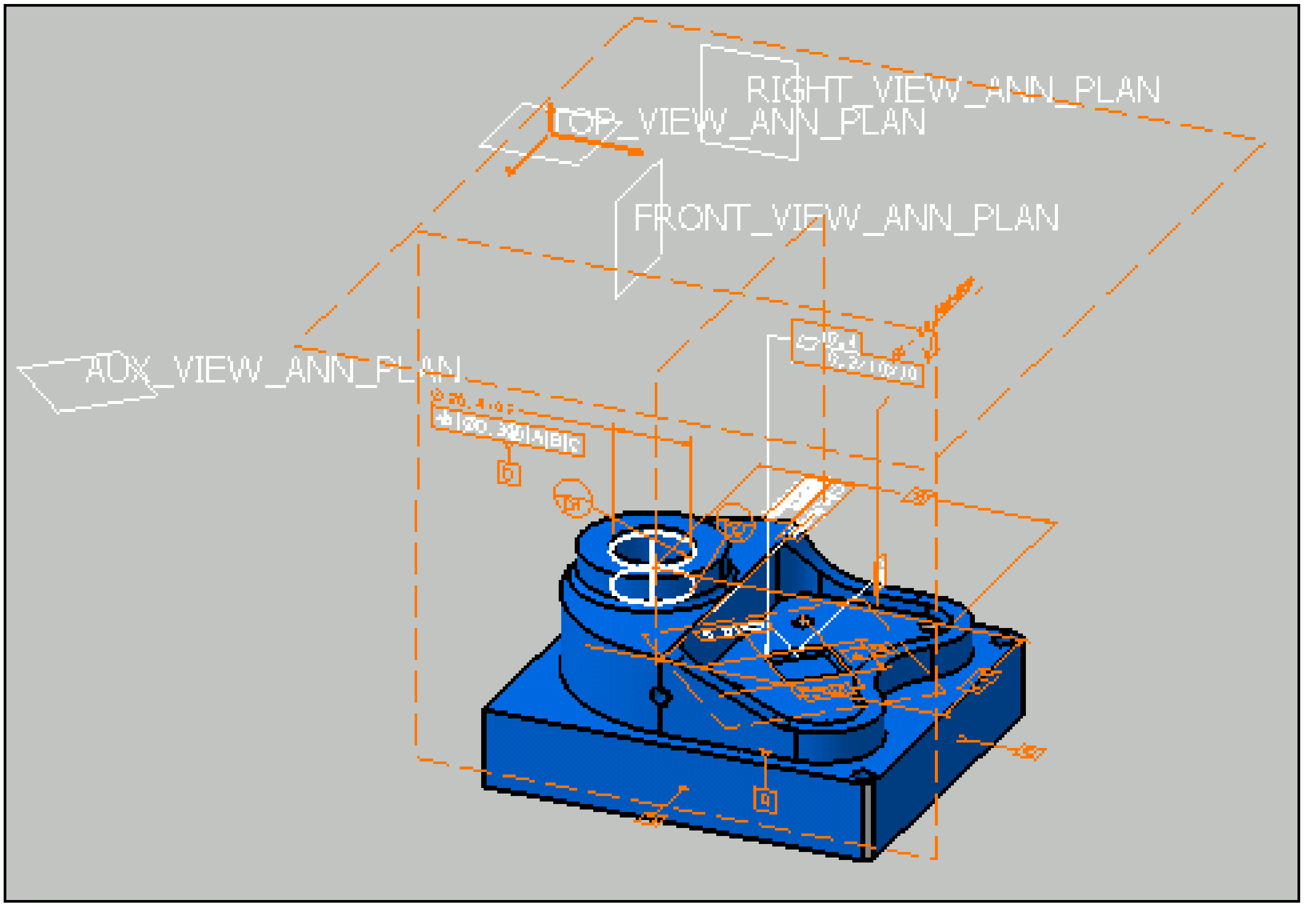
Make sure that the **3D Annotation Query Switch On/Switch Off** icon is activated.

1. In the specification tree, expand the **V4data** and then the ***MASTER** nodes under **Product.1**.
2. Copy the ***TOLP2** annotation set which is located under the ***MASTER** node, by selecting it and pressing **Ctrl+C** for example.
3. Paste the annotation set right under **Product.1**, using **Ctrl+V** for example.
4. Wait for the migration to complete. This may take some time, depending on the amount of data to handle.

When the migration is over, a dialog box appears, informing you that some data could not be migrated, and that a capture was created for non-migrated data. Click OK to close it.



Also, a new annotation set, **Copy of *TOLP2**, is created under the Product.1 node in the specification tree.



- The created annotation set uses the V4 parent standard.
- Migrated annotations are associative to V4 geometry.

5. Under the **Copy of *TOLP2** annotation set node, expand the **Captures** node and click **Non migrated annotations of *TOLP2** to visualize the V4 data that could not be migrated.

Creating Note Object Attributes

Note Object Attributes (NOA) lets you create a customized annotation from a text or a 2D component.

When extracting a Note Object Attribute to 2D, frames are not generated.

Note Object Attribute From a Text: click this icon, select a geometry then enter a text.

Note Object Attribute From a Ditto: click this icon, select a geometry then select a 2D component from a catalog.

Store a Note Object Attribute into a Catalog: create a new catalog, select the Note Object Attribute.

Note Object Attribute From a Text

This task shows you how to create a Note Object Attribute from a text.
See Instantiating a Note Object Attribute task and Note Object Attribute concept.

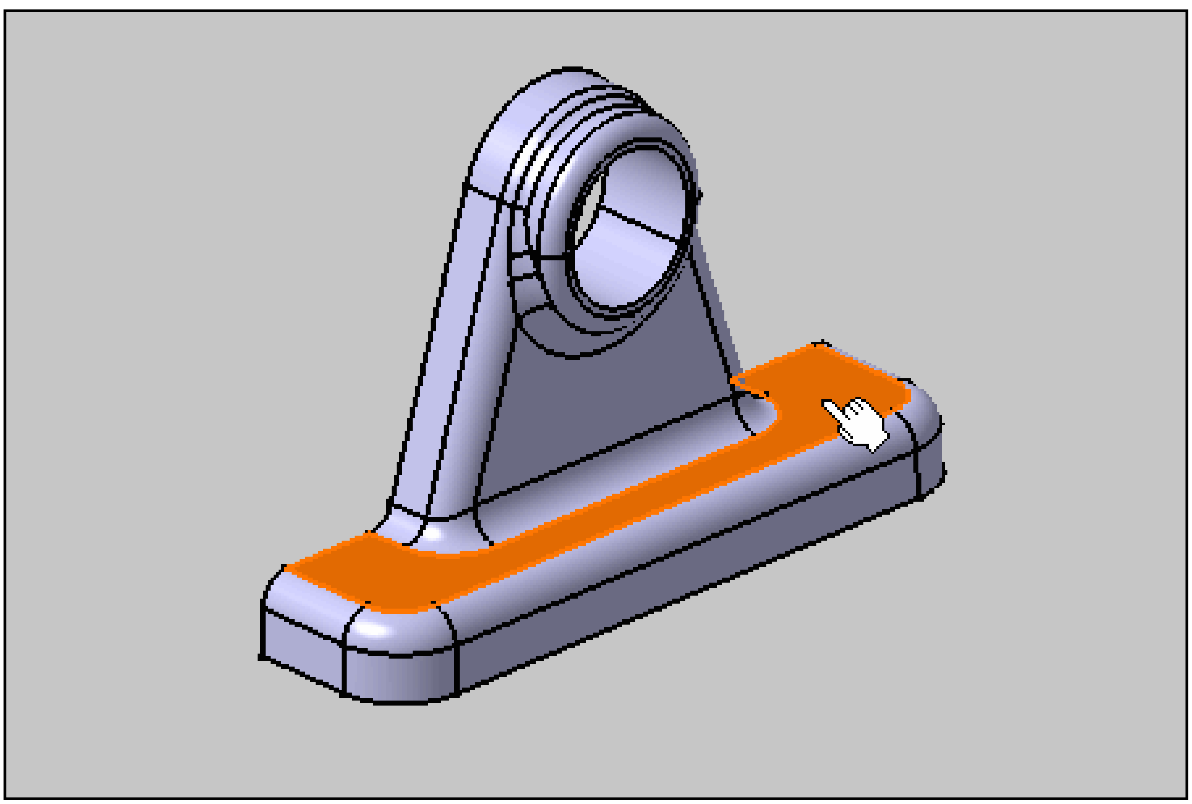
Open the Tolerancing_Annotations_01 CATPart document:

- Check the option allowing you to create a Note Object Attribute, see Tolerancing.
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

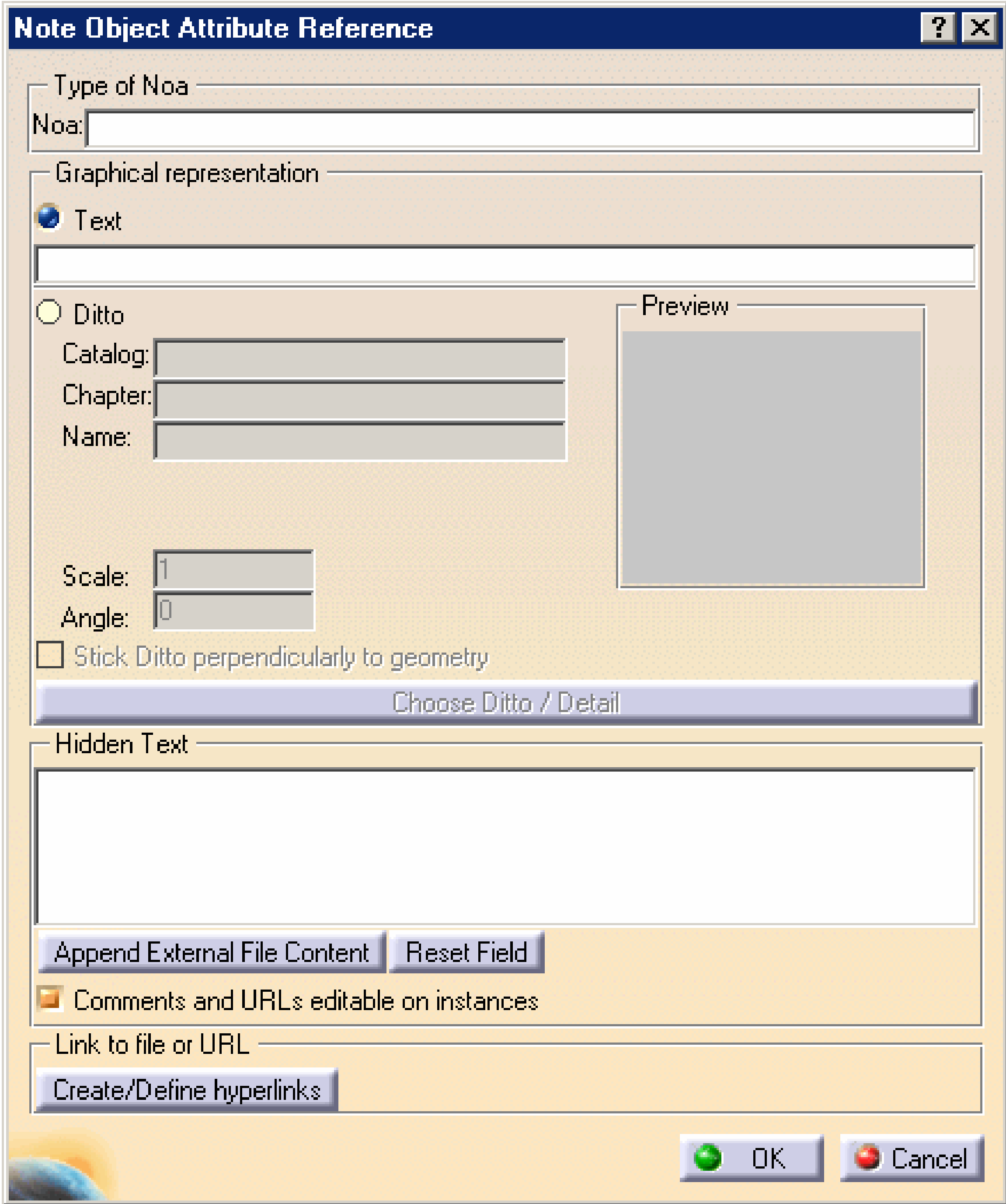
1. Click the **Note Object Attribute** icon: 

2. Select the surface as shown on the part.

This scenario illustrates the creation of a note object attribute by selecting geometry, but you can also select any Part Design or Generative Shape Design feature in the specification tree. In this case, the created annotation will not be attached to the selected feature, but to its geometrical elements at the highest level.



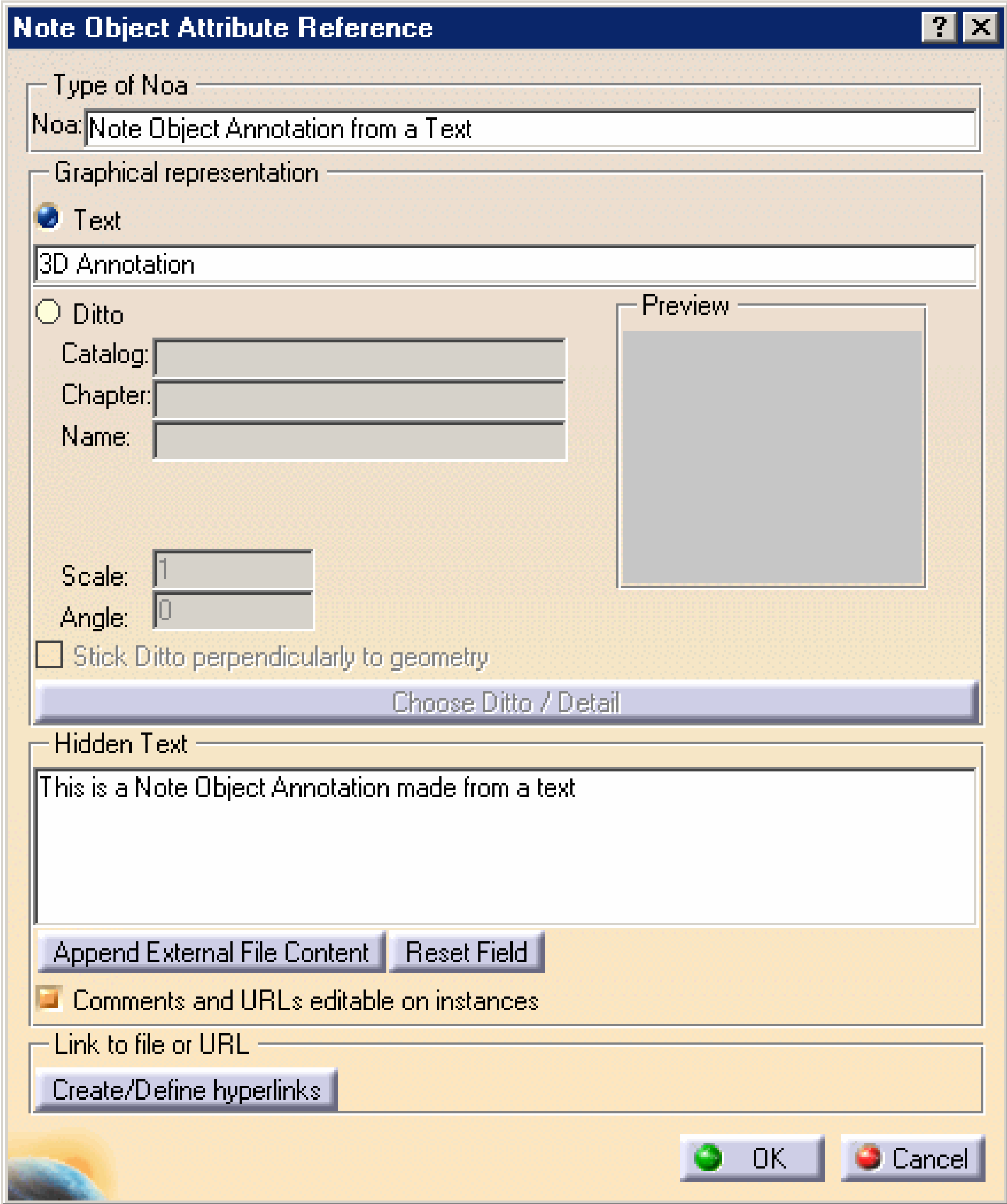
The **Note Object Attribute Reference** dialog box appears.



3. Select **Text** in **Graphical representation** and enter the following texts:

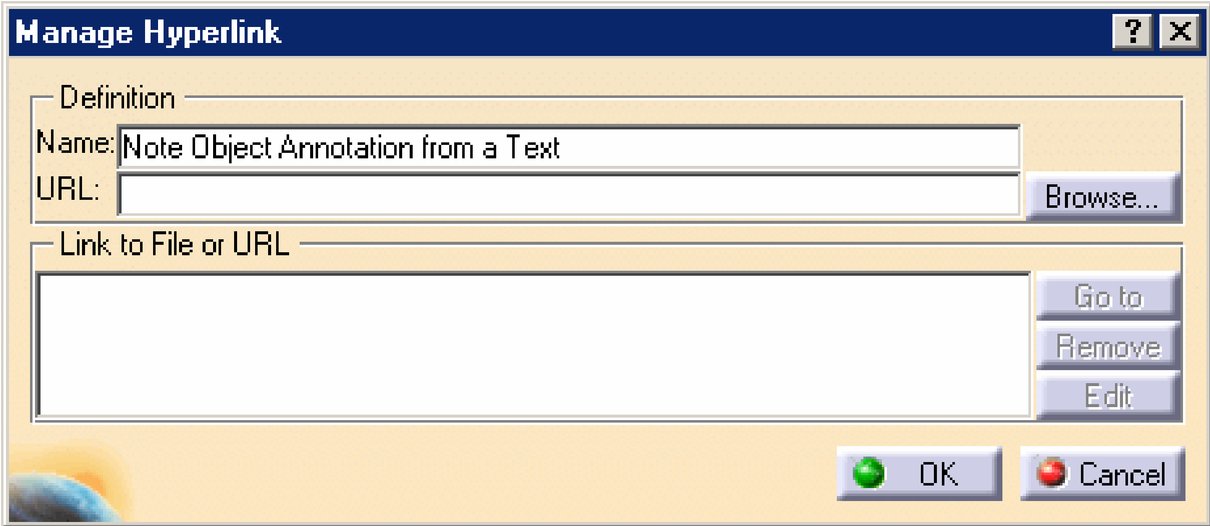
- The type of the Note Object Attribute: **Note Object Annotation from a Text**
- The text: **3D Annotation**
- A **Hidden text**: **This is a Note Object Annotation made from a text**

Hidden text in a Note Object Attribute is not extracted in drawing view.
The **Comments and URLs editable on instances** option allows user to modify the **Hidden Text** data during Note Object Attribute instantiation or modification.



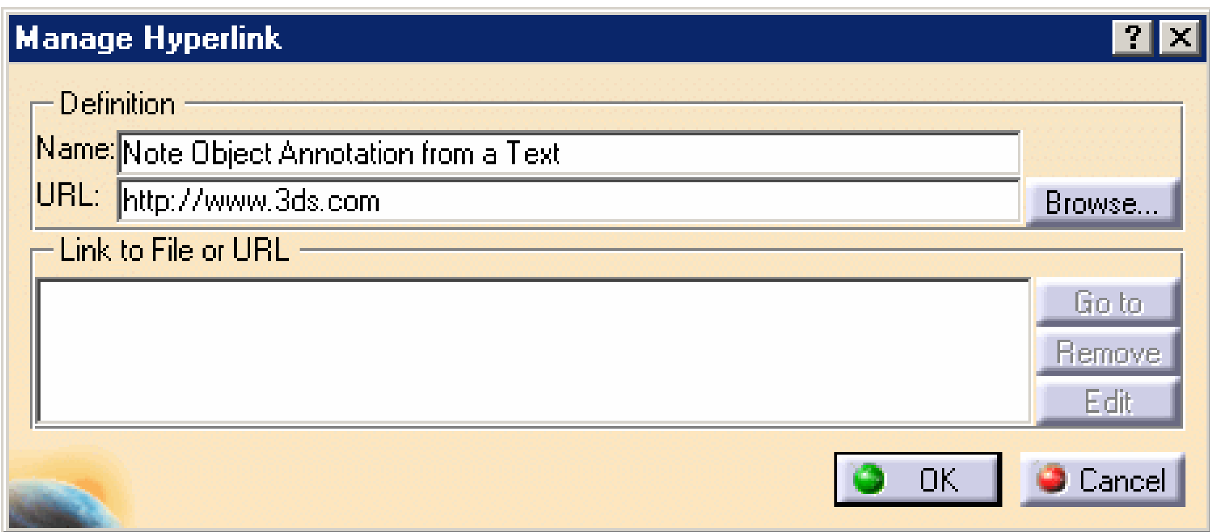
4. Click **Create/Define hyperlinks**.

The **Manage Hyperlink** dialog box appears.



Links in a Note Object Attribute are not extracted in drawing view.
You can add one or several links to a Note Object Attribute to describe it for example.

5. Enter the following link: http://www.3ds.com

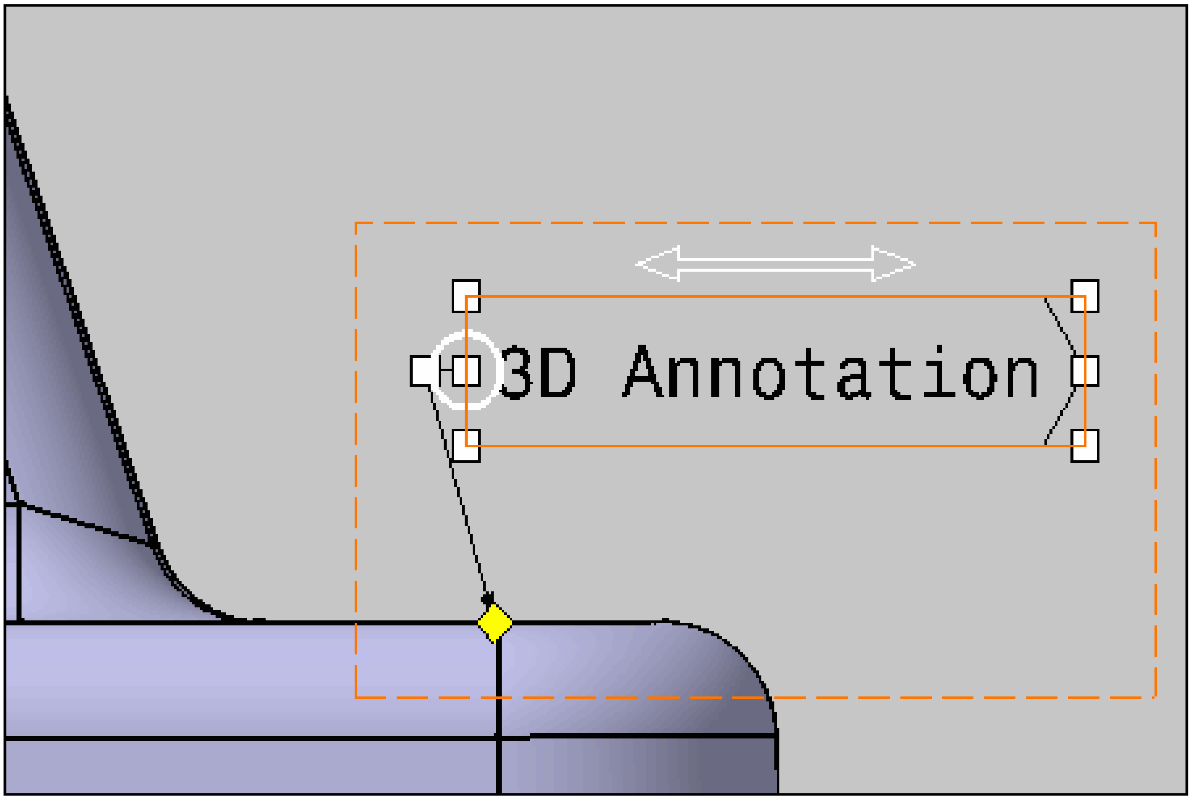


6. Click **OK** in the **Manage Hyperlink** dialog box.

The link has been added to the Note Object Attribute and you can retrieve this dialog box and the comments by double-clicking the annotation in the geometry or in the specification tree, in the **Note Object Attribute Edition** dialog box.

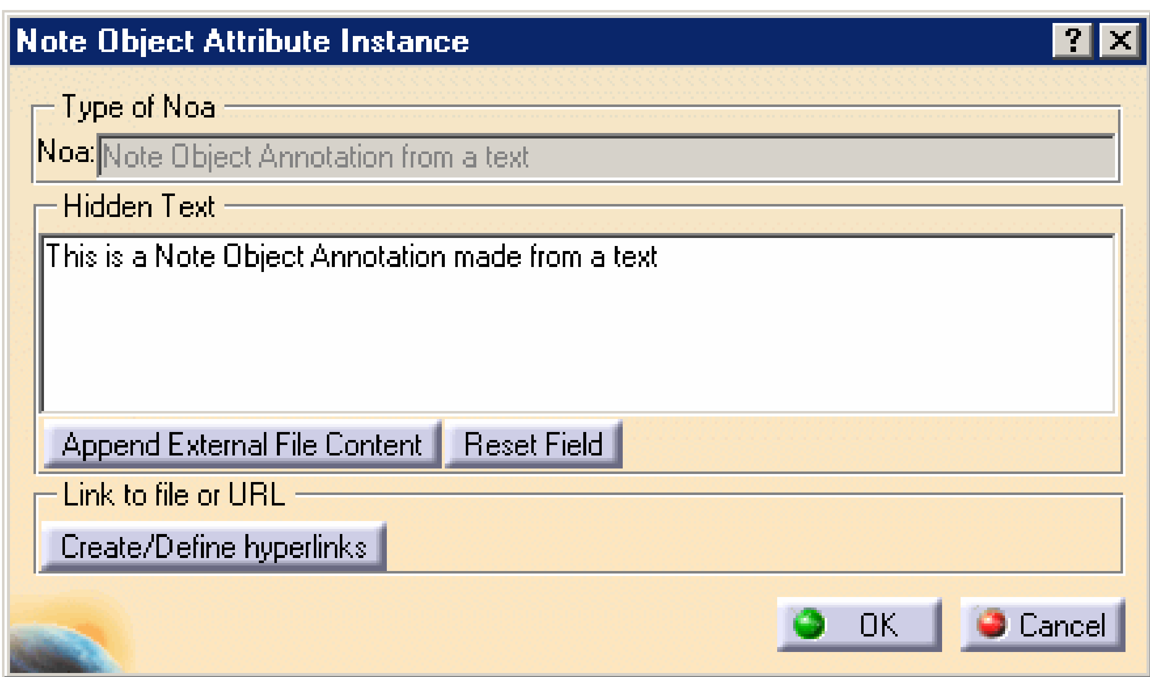
7. Click **OK** in the **Note Object Attribute Reference** dialog box.

The Note Object Attribute is created.



8. Double-click the annotation in the geometry or in the specification tree.

The **Note Object Attribute Instance** dialog box appears.
You can edit the link added to the Note Object Attribute and the comments.



9. Click **Cancel**.

Note Object Attribute From a Ditto

This task shows you how to create a Note Object Attribute from a ditto (a 2D component in a drawing).
See Instantiating a Note Object Attribute task and Note Object Attribute concept.

Two behaviors of Note Object Attribute created from a 2D component are available from the **Stick Ditto perpendicularly to geometry** option in the **Note Object Attribute Reference** dialog box.

- Unchecked, the 2D component is instantiated with a frame and a leader.
- Checked, the 2D component is instantiated without frame or leader and its origin point is stuck and set on the selected geometry. Its normal is defined by the V axis of the 2D component.

When the Note Object Attribute orientation is modified, text contained in the 2D component follows or not according to the text orientation reference.

To modify the orientation reference properties, right-click the text and select the **Properties...** command.
In the **Text** tab, select the **Reference** in the **Orientation** category: **Sheet** or **View / 2D Component**

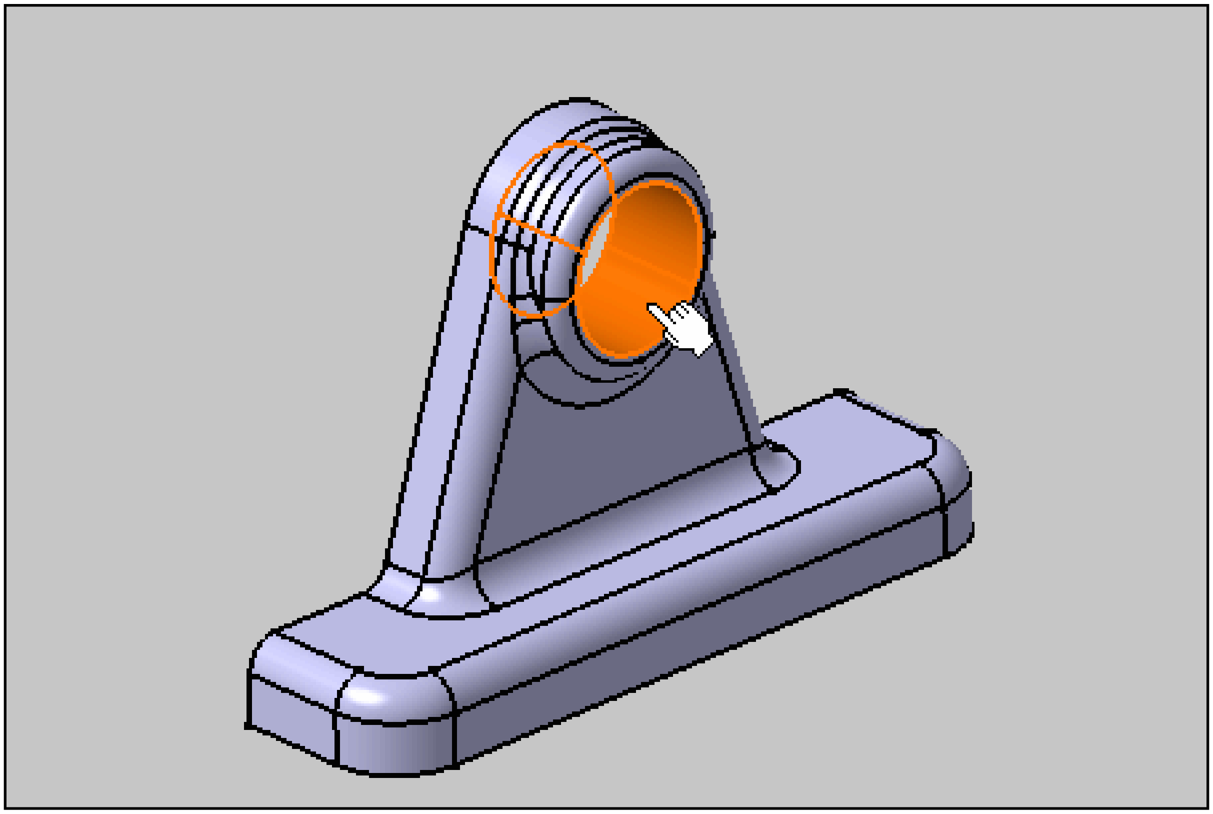
- With the **Sheet** option, the text does not follow the Note Object Attribute orientation.
- With the **View / 2D Component** option, the text follows the Note Object Attribute orientation.

Open the Tolerancing_Annotations_01 CATPart document:

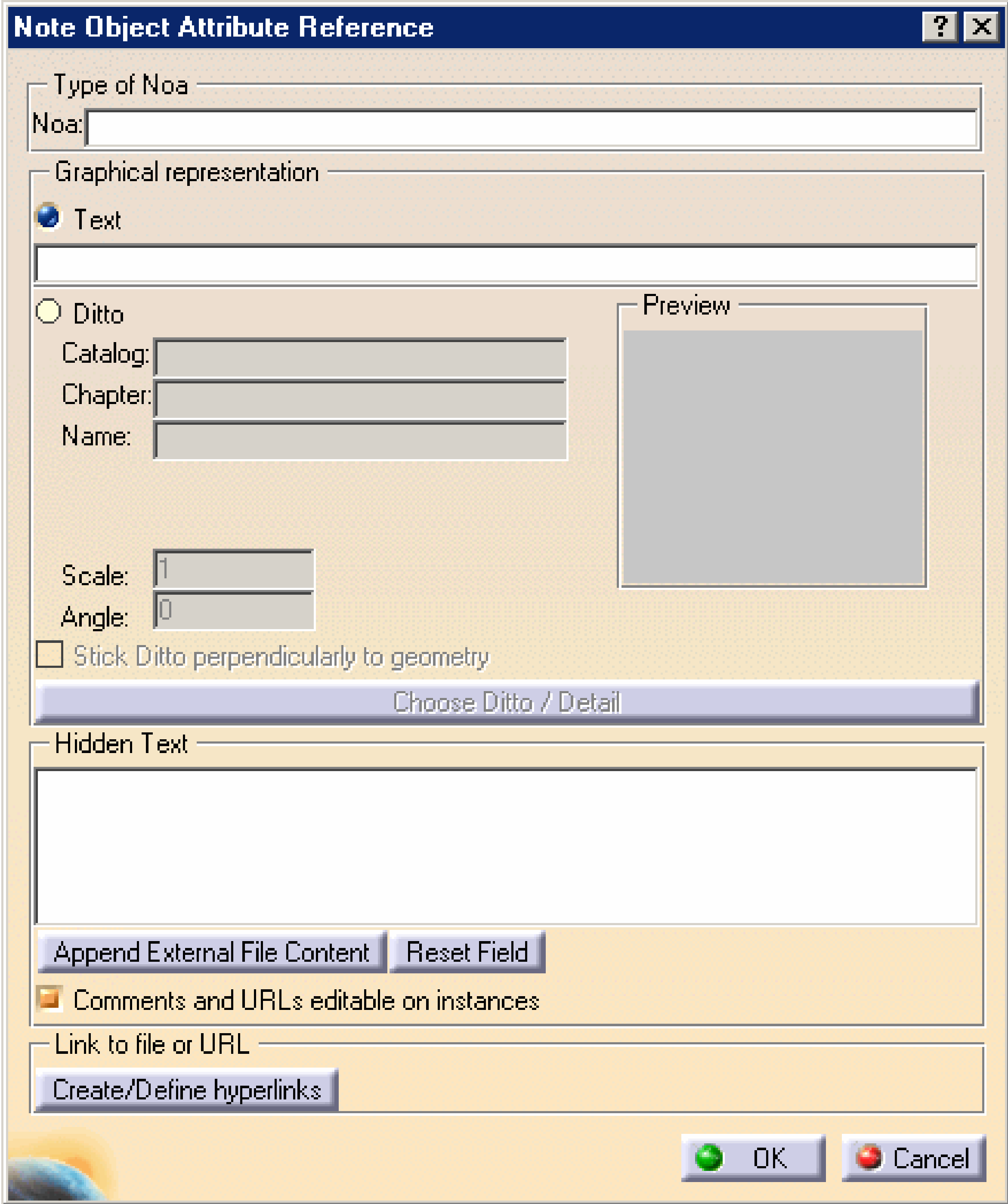
- Check the option allowing you to create a Note Object Attribute, see Tolerancing.
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Click the **Note Object Attribute** icon:

2. Select the surface as shown on the part.



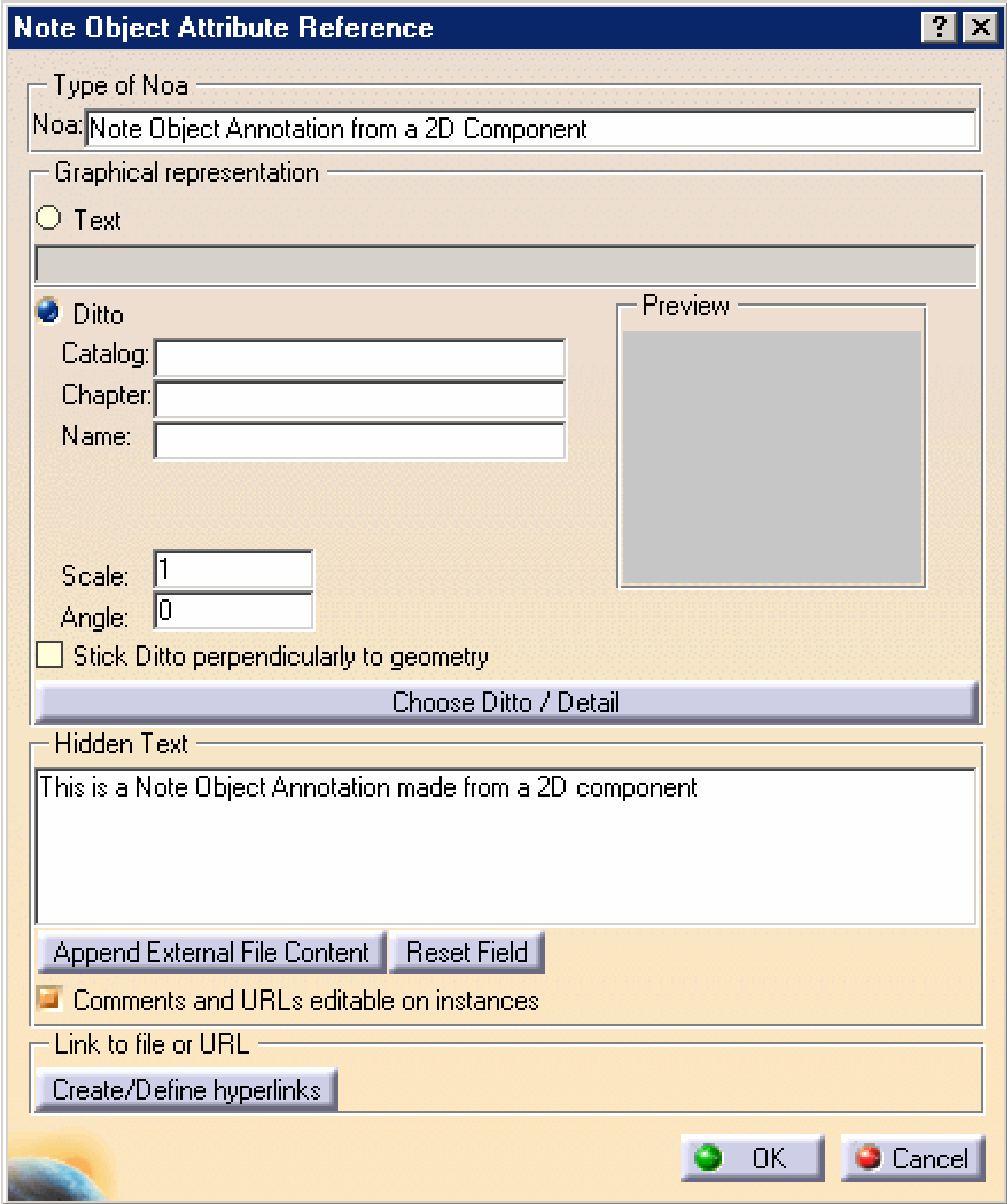
The **Note Object Attribute Reference** dialog box appears.



3. Select **Ditto** in **Graphical representation** and enter the following texts:

- The type of the Note Object Attribute: **Note Object Annotation from a 2D Component**
- A hidden text: **This is a Note Object Annotation made from a 2D component**

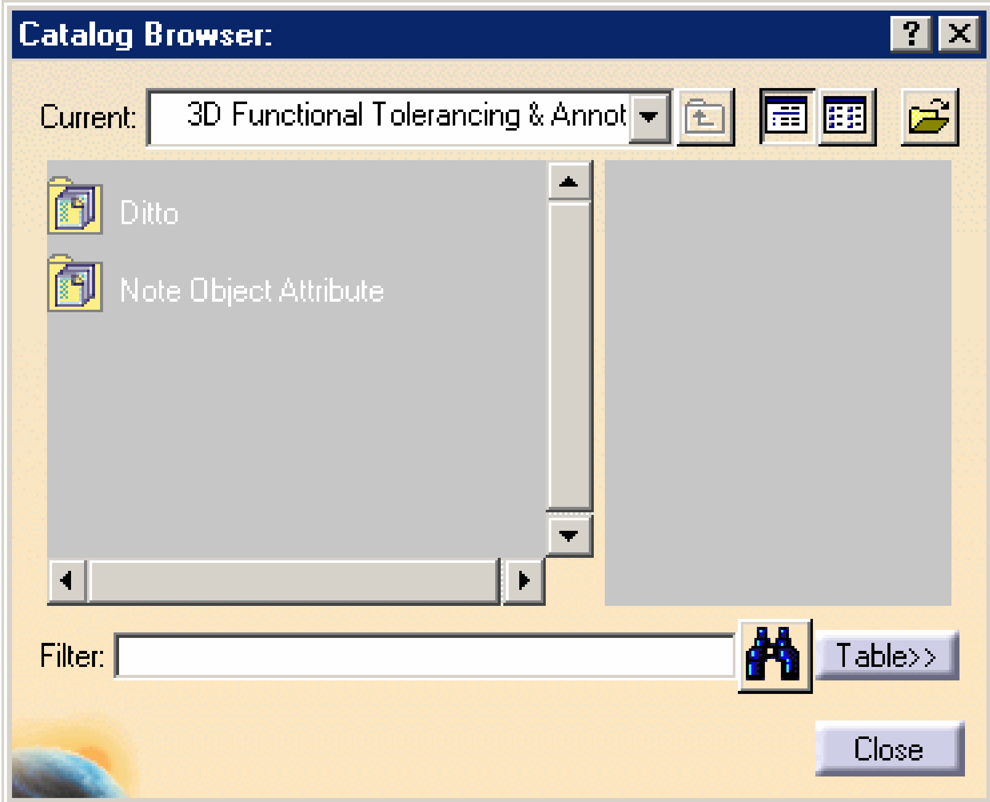
Hidden text in a Note Object Attribute is not extracted in drawing view.
The **Comments and URLs editable on instances** option allows user to modify the **Hidden Text** data during Note Object Attribute instantiation or modification.



4. Click **Choose Ditto / Detail**.

The **Catalog Browser** dialog box appears.

5. Browse and select for the Component document.



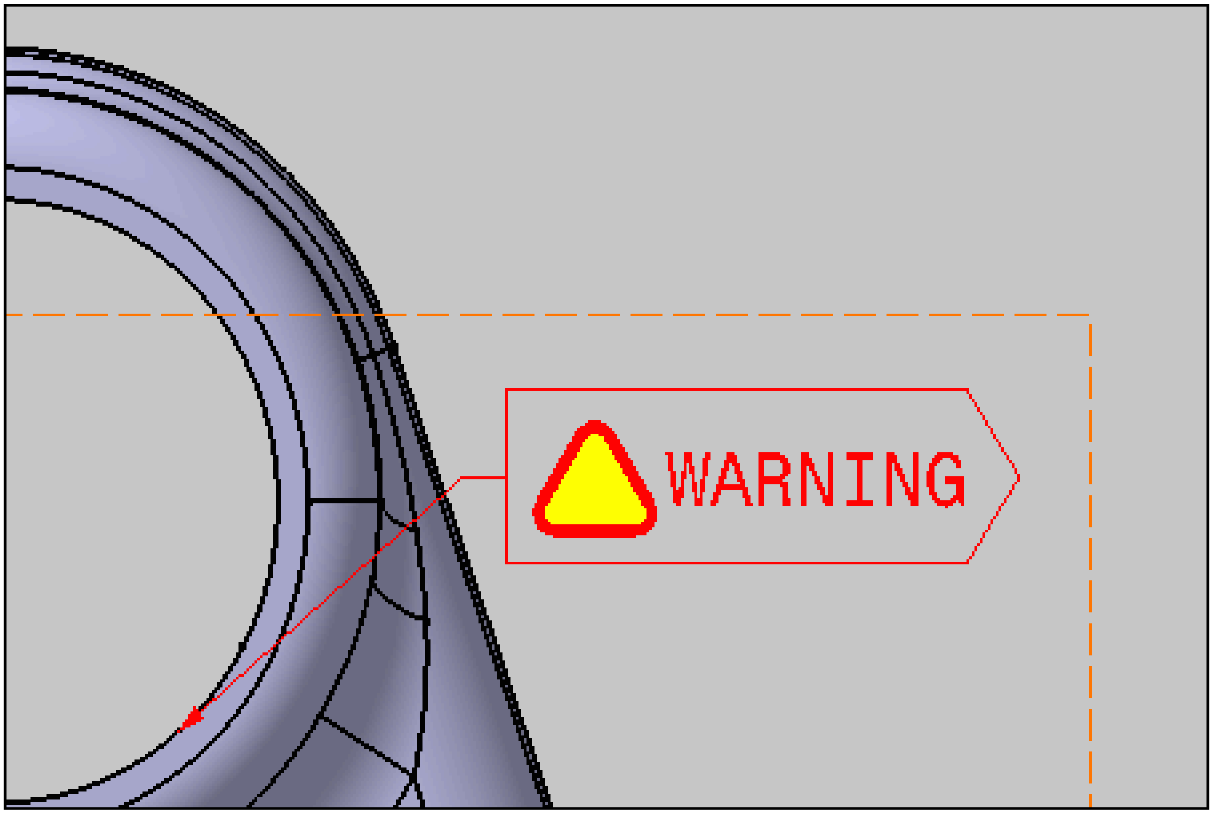
6. Double-click the **Ditto** component family item.

7. Select the **2D Component.2** component item.

8. Click **OK** in the **Catalog Browser** dialog box.

9. Click **OK** in the **Note Object Attribute Reference** dialog box .

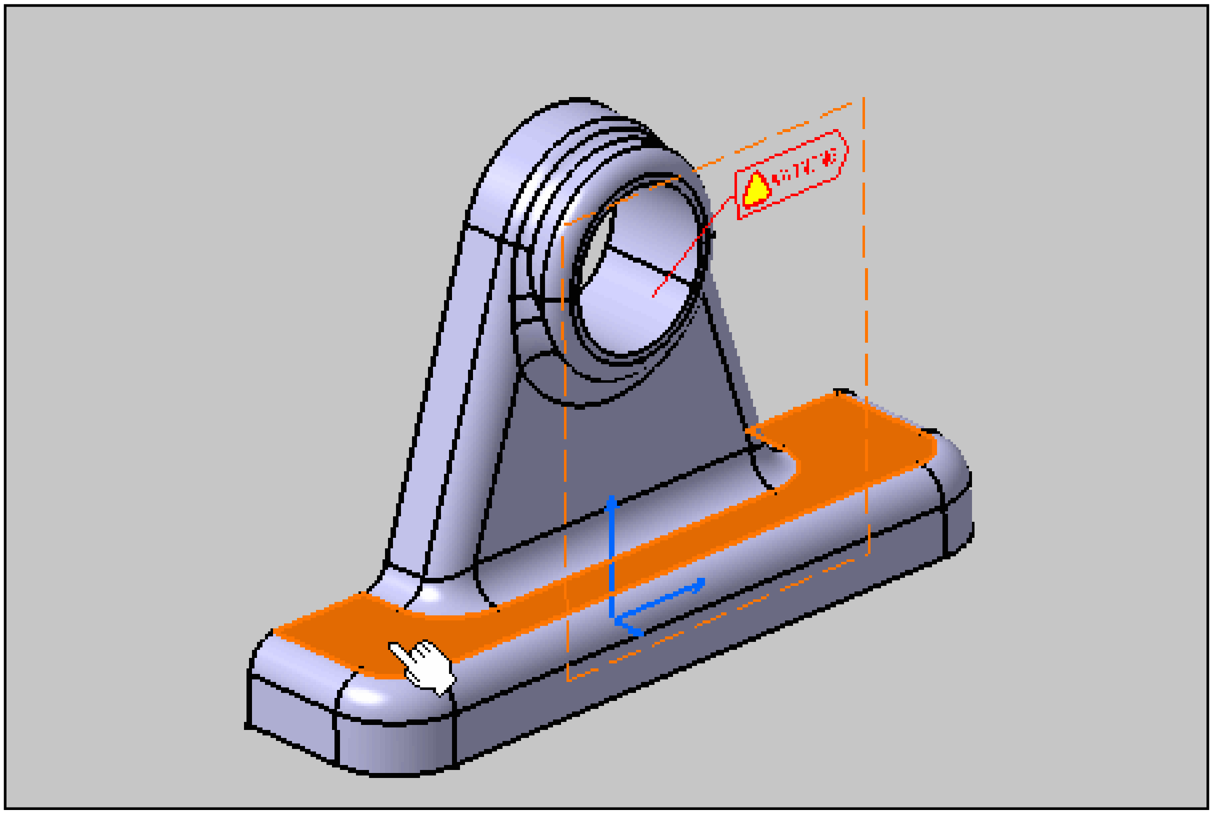
The Note Object Attribute is created.



When a text is created in a 2D component, by default is orientation is associated with the sheet. In this case, when the Note Object Attribute orientation change the text do not follow the new orientation.
To perform the text association orientation select the text in the 2D context, then right-click in the contextual menu **Properties**. In the Properties dialog box,

10. Click the **Note Object Attribute** icon: 

11. Select the surface as shown on the part.

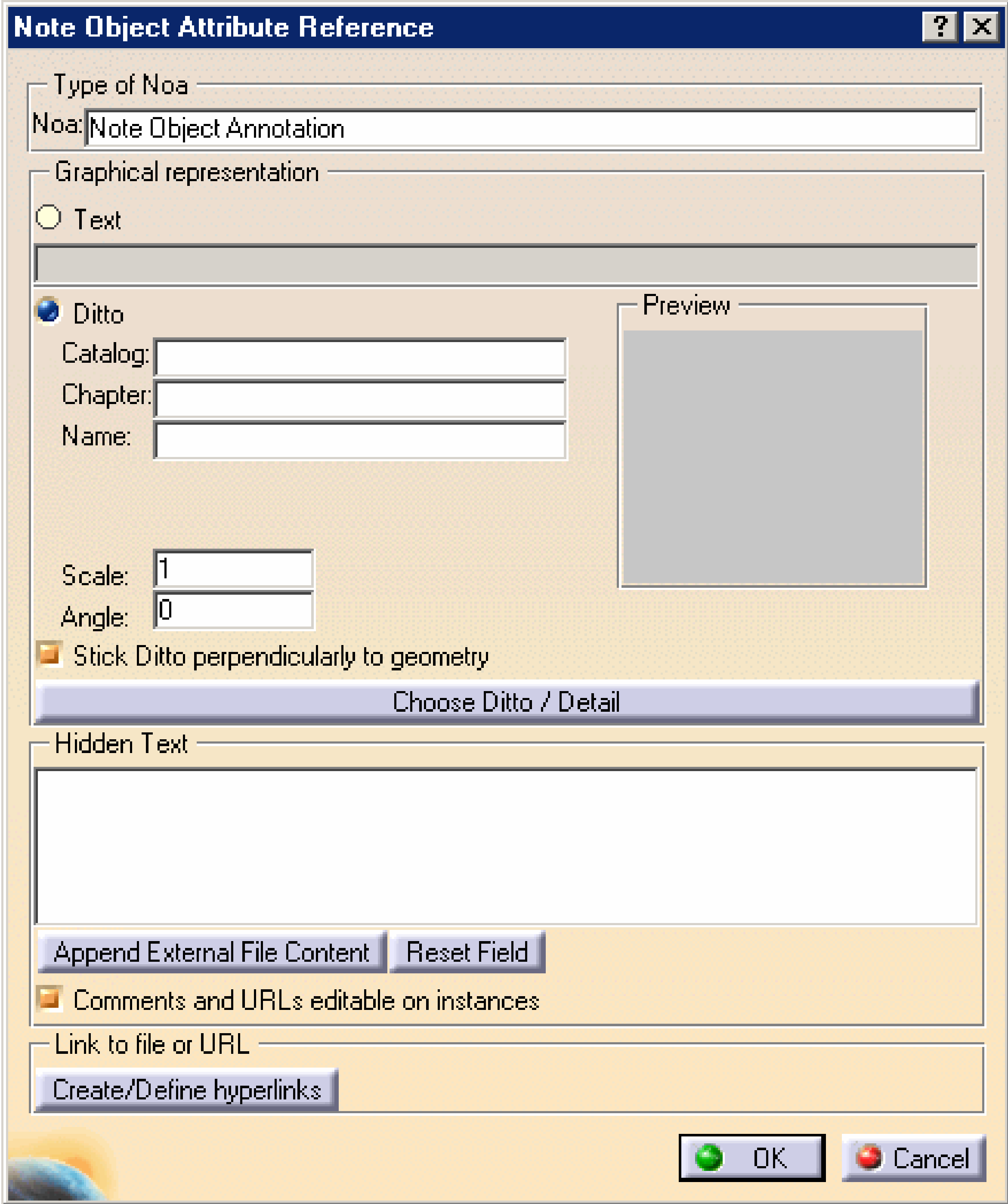


The **Note Object Attribute Reference** dialog box appears.

12. Select **Ditto** in **Graphical representation** and enter the following texts:

The type of the Note Object Attribute: **Note Object Annotation**

13. Check the **Stick Ditto perpendicularly to geometry** option



14. Click **Choose Ditto / Detail**.

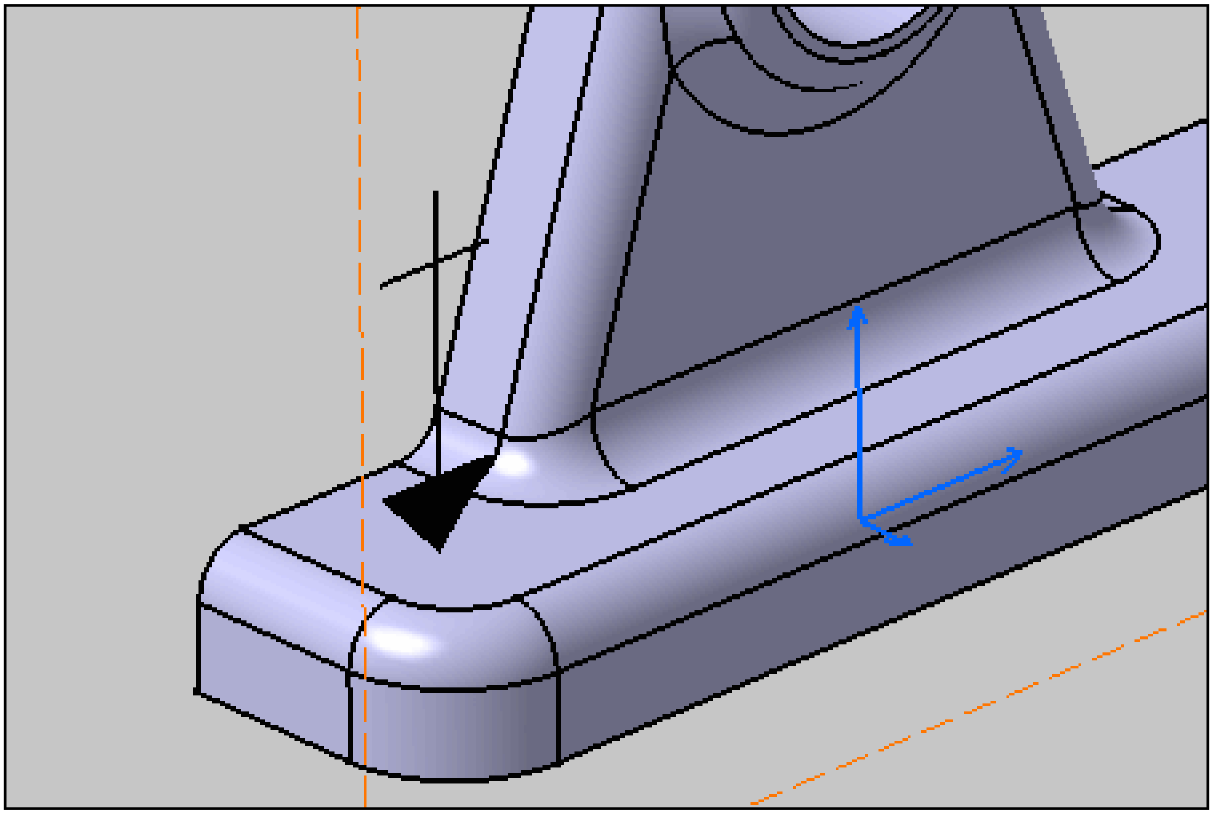
The **Catalog Browser** dialog box appears.

15. Select the **2D Component.1** component item.

16. Click **OK** in the **Catalog Browser** dialog box.

17. Click **OK** in the **Note Object Attribute Reference** dialog box .

The Note Object Attribute is created.

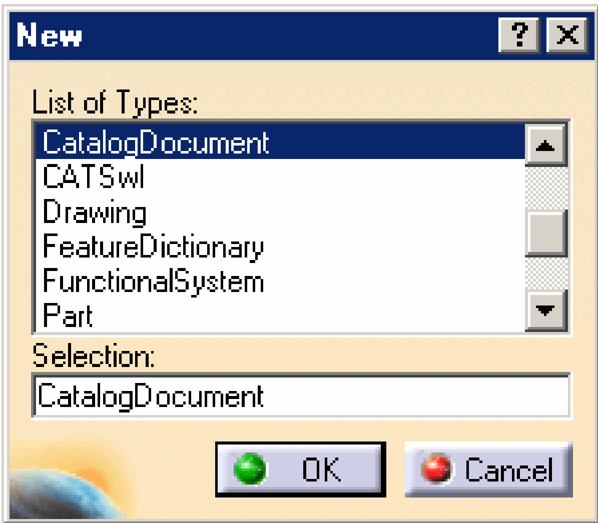


Storing a Note Object Attribute into a Catalog

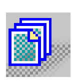
This task shows you how to store a Note Object Attribute into a catalog to be re-instantiated by user in another document.

Open the NoteObjectAttribute CATPart document.

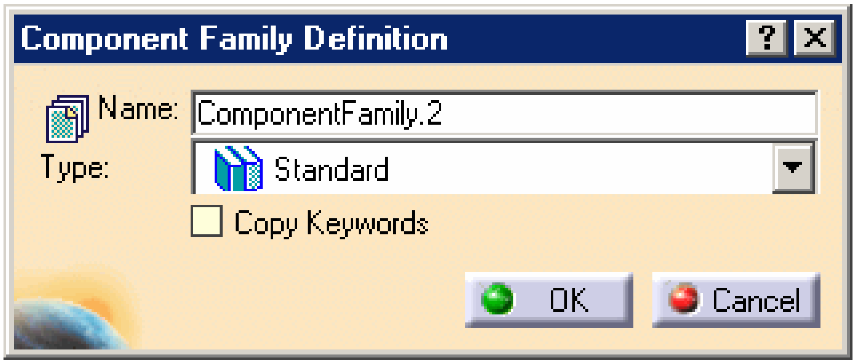
1. Create a new catalog document: **File -> New: CatalogDocument**



2. Check that Chapter.1 is activated in the catalog tree.

3. Click the **Add Family** icon: 

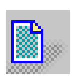
The **Component Family Definition** dialog box appears.



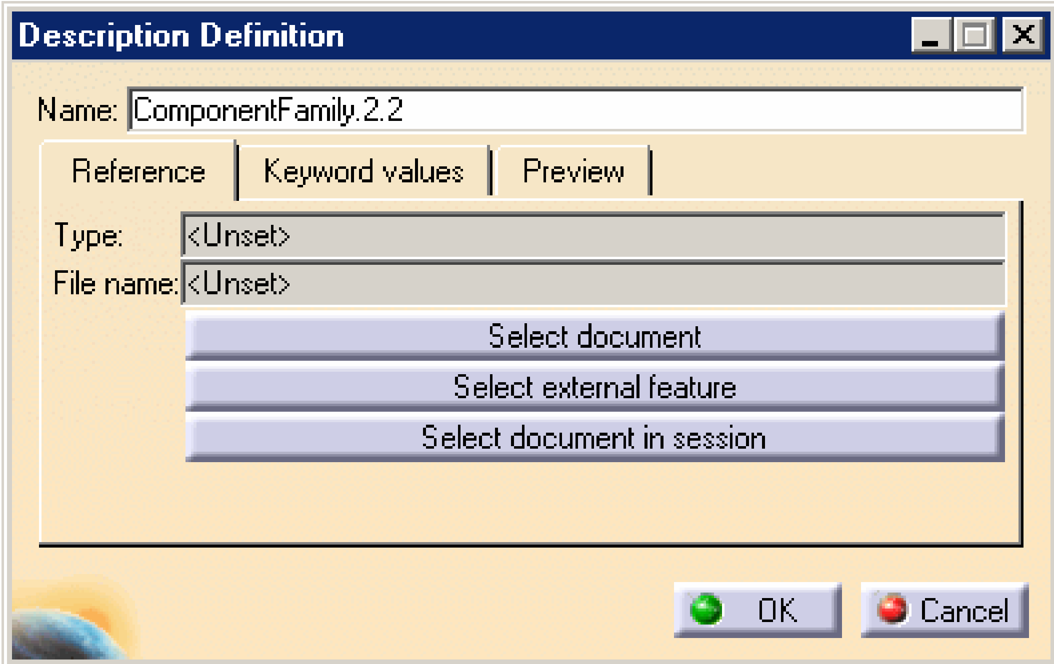
4. Click **OK**.

The **Component Family Definition** is added to the chapter.

5. Double-click the Component Family to activate it.

6. Click the **Add Component** icon: 

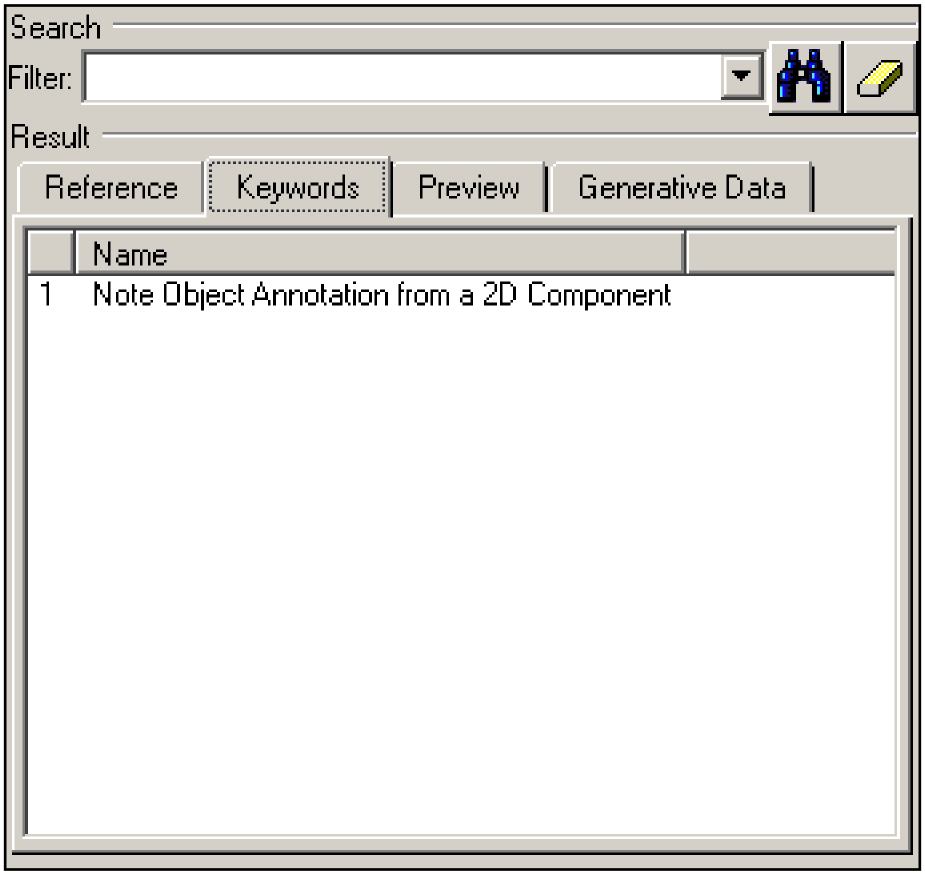
The **Description Definition** dialog box appears.



7. Click **Select external feature** command and select the **Note Object Attribute from 2D component** from the specification tree or the geometry.

8. Click **OK**.

The Note Object Attribute is added to the catalog.


Search
Filter:
Result
Reference
Keywords
Preview
Generative Data
Name
1 Note Object Annotation from a 2D Component

Managing Annotation Connections

Manage annotation connection offers to create, delete, modify or rename geometrical elements or user surfaces of an existing annotation:

Use the Scope Range: select the **Geometry Connection Management** command and select an annotation.

Add Geometry: select the **Geometry Connection Management** command and select an annotation.

Add Component: select the **Geometry Connection Management** command and select an annotation.

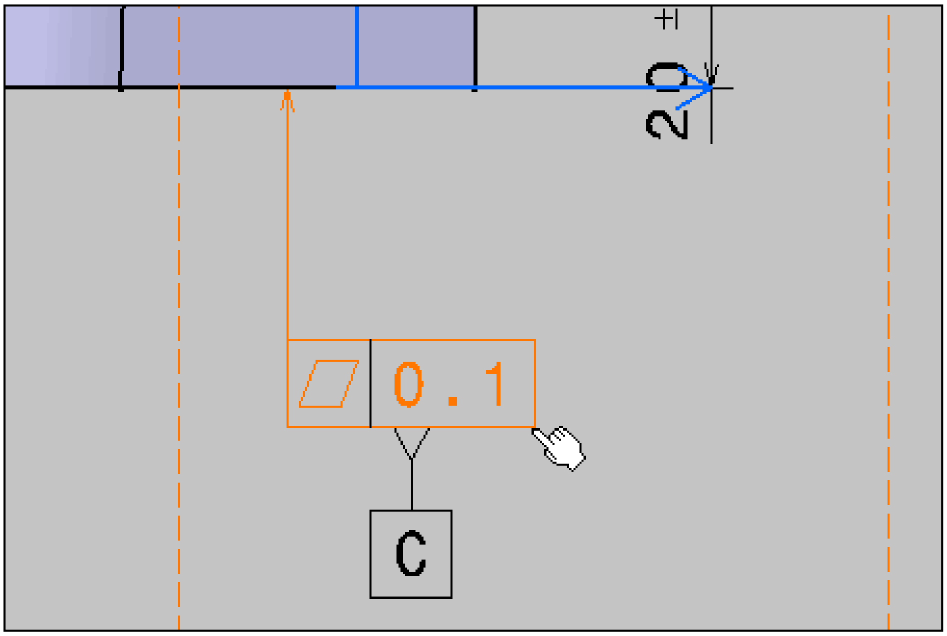
Using the Scope Range

This task shows you how to use the scope range when using the connection management and check the validity of the reconnected annotations according to the selected scope option.

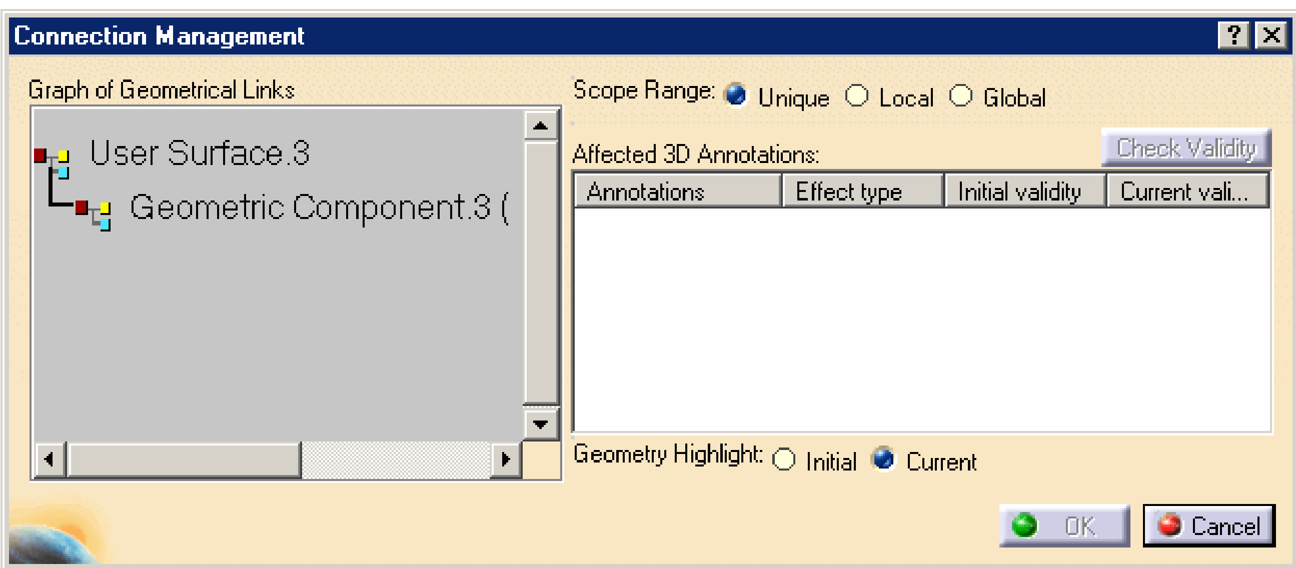
Open the Tolerancing_Annotations_03 CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Right-click the annotation as shown on the part and select the **Associated Geometry -> Geometry Connection Management** from the contextual menu.



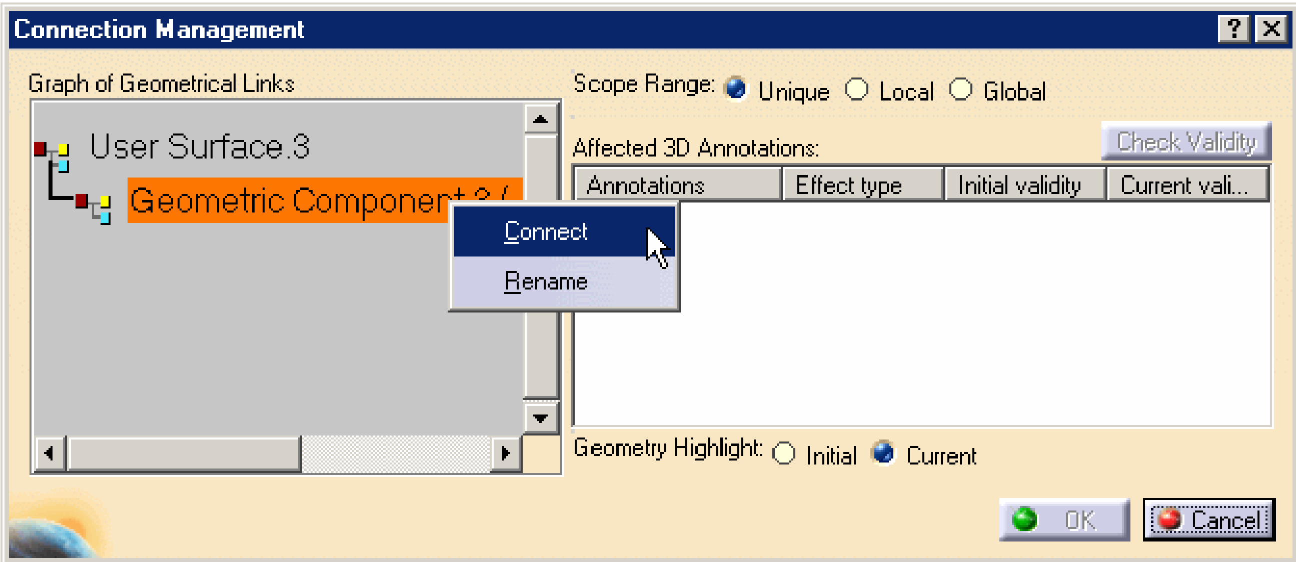
The **Connection Management** dialog box appears.



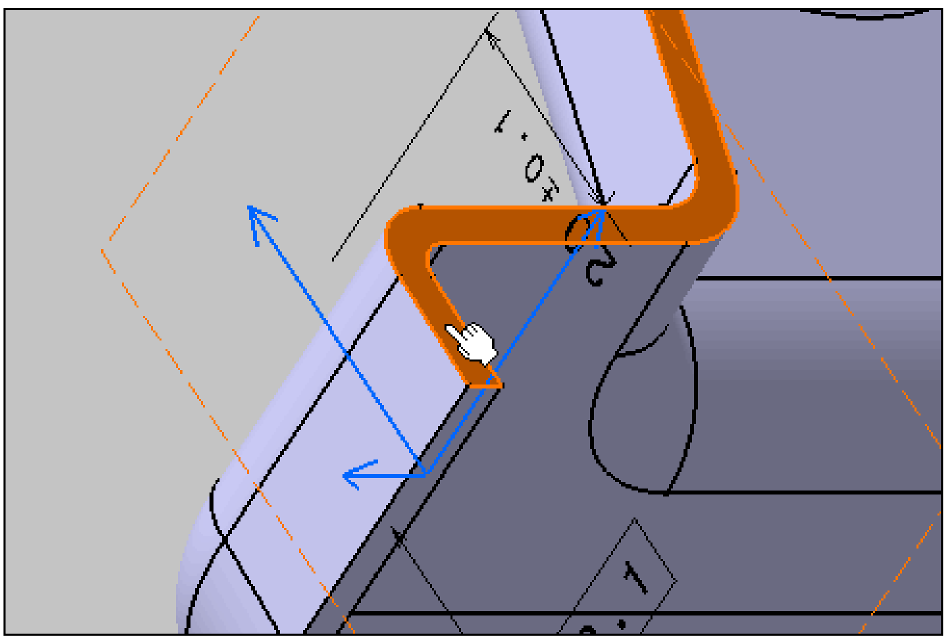
2. Check that **Unique** option is activated in **Scope Range**.

The geometry connection modification will only affect the selected annotation.

3. Right-click Geometric Component.1 in the **Graph of Geometrical Links** as shown and select the **Reconnect** contextual menu.



4. Select the surface as shown on the part.



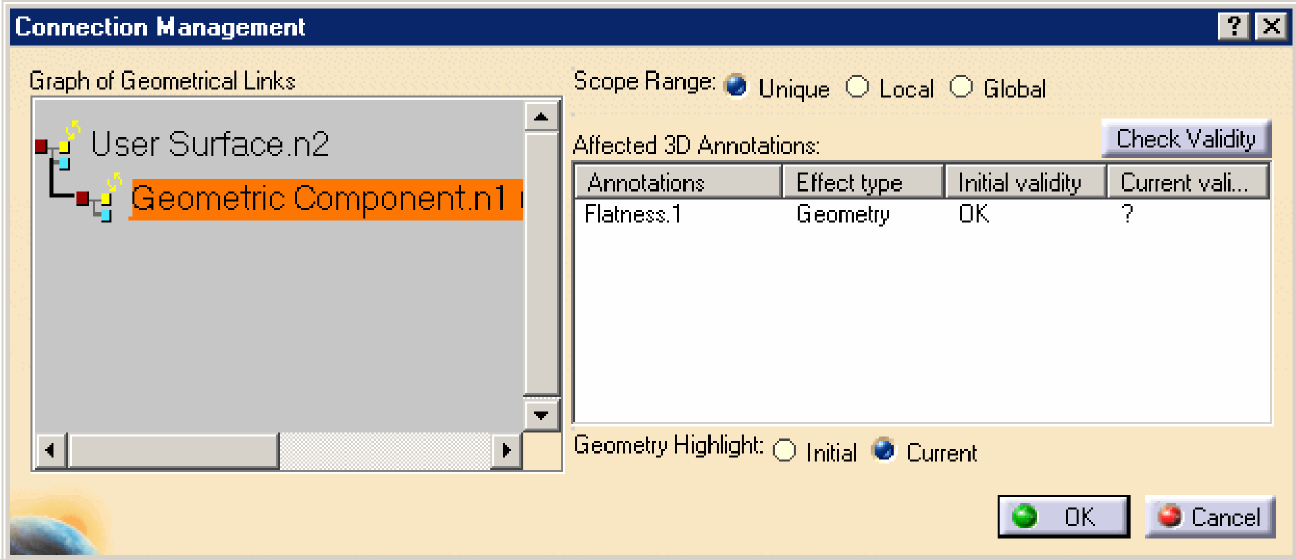
The **Connection Management** dialog box displays:

The selected annotation

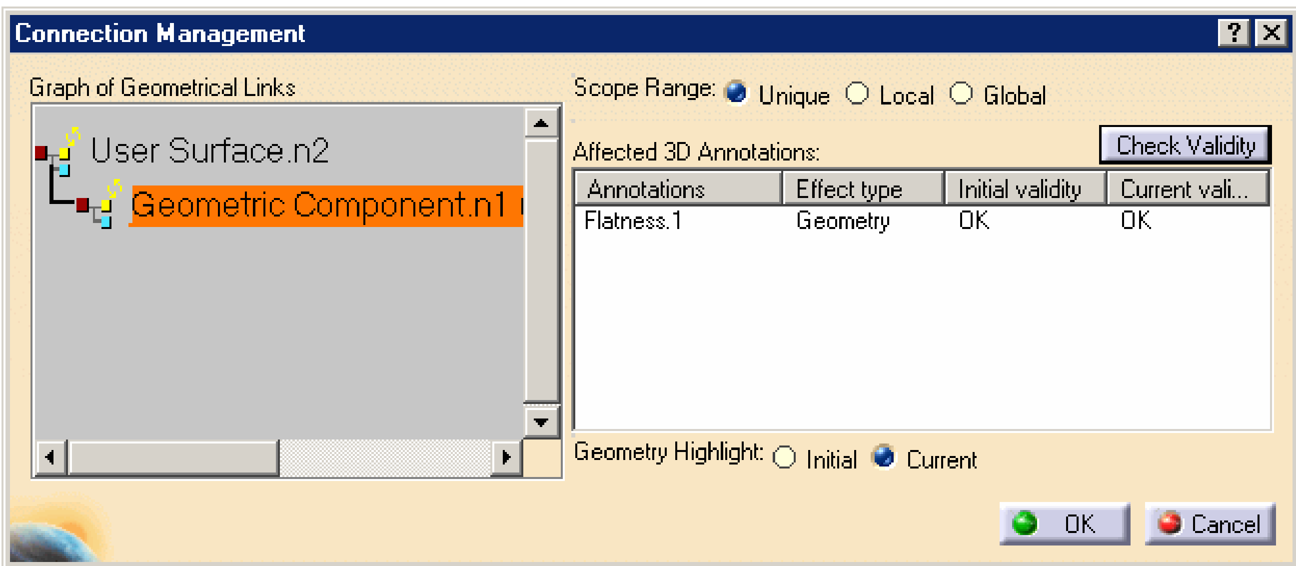
The effect type

The initial validity of the selected annotation

The current validity of the selected annotation after the reconnection.

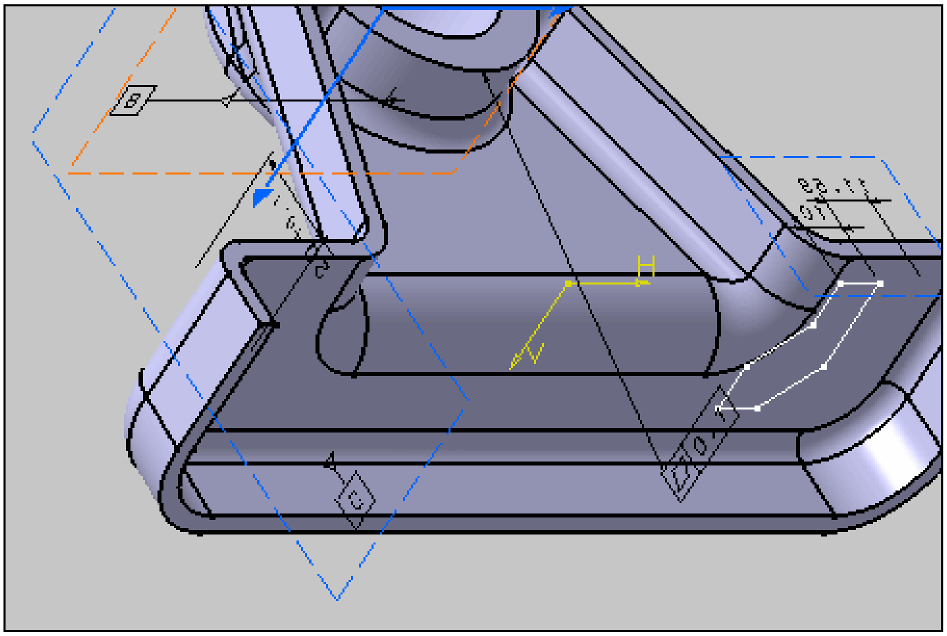


5. Click the **Check Validity** command to check the new geometry component validity relative to the selected annotation.



6. Click **OK**.

The annotation is now connected to the new surface

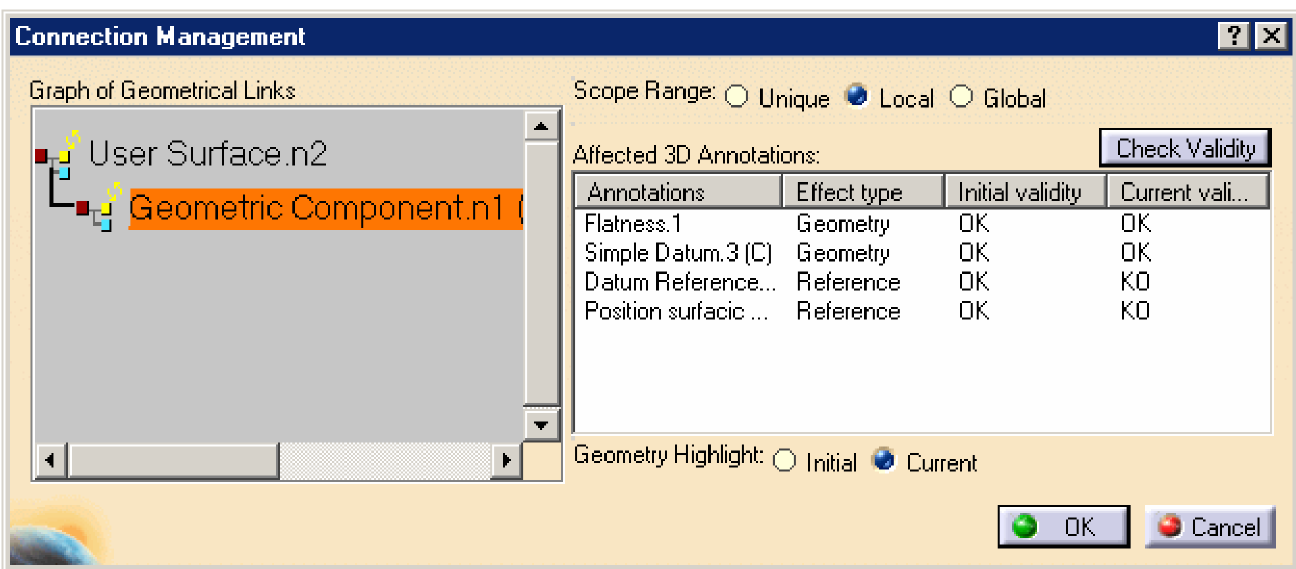


The connection of the annotation on the new geometrical element is computed according to its local axis, not with the point where you make the selection.

7. Close the part document, re-open it and redo step 1 to 5 with the **Local** option in **Scope Range**.

   The geometry connection modification will only affect all the annotation that are directly applied to the User Surface.2 feature.

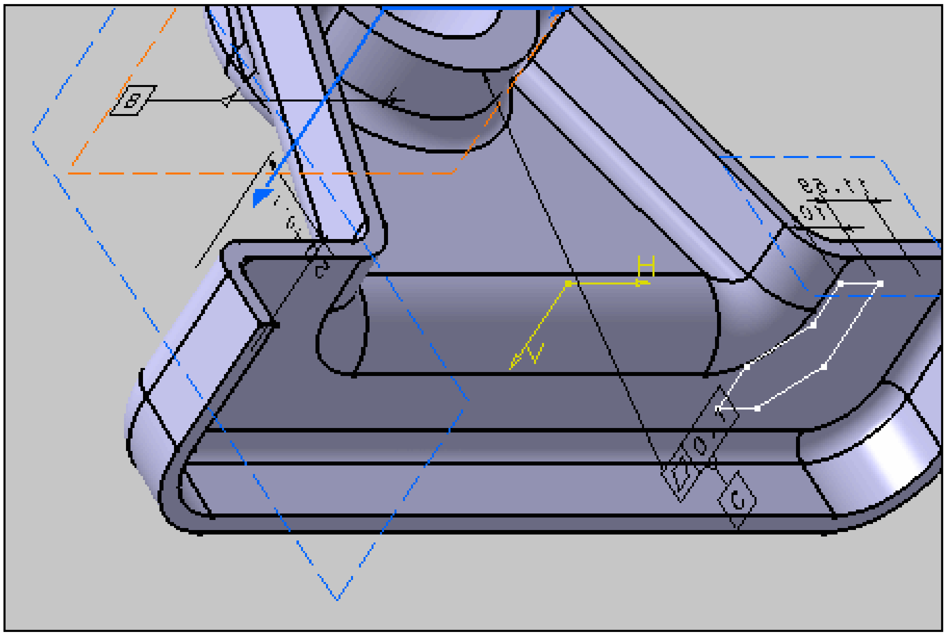
   The **Connection Management** dialog box displays now the selected annotation and all annotations related with it.



   The validity status is KO for the **Datum Reference Frame.1** and **Position surfacic profile.1** annotations because they will not have any meaning if the modification is applied.

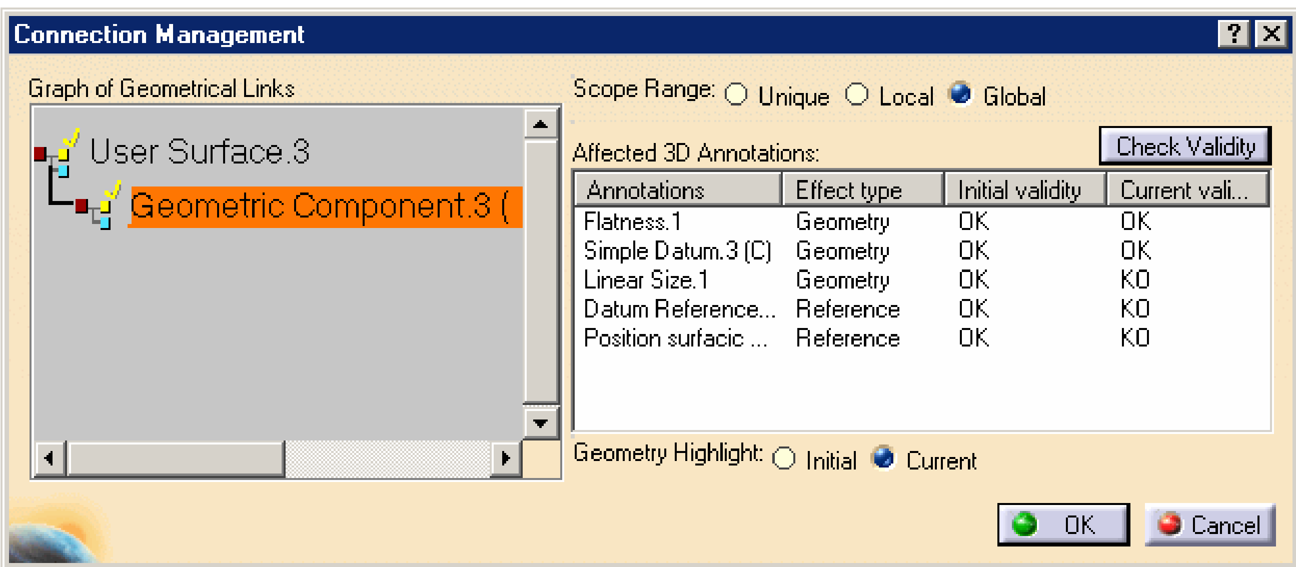
8. Click **OK**.

   The annotations are now connected on the new surface.



9. Close the part document, re-open it and redo step 1 to 5 with the **Global** option in **Scope Range.**

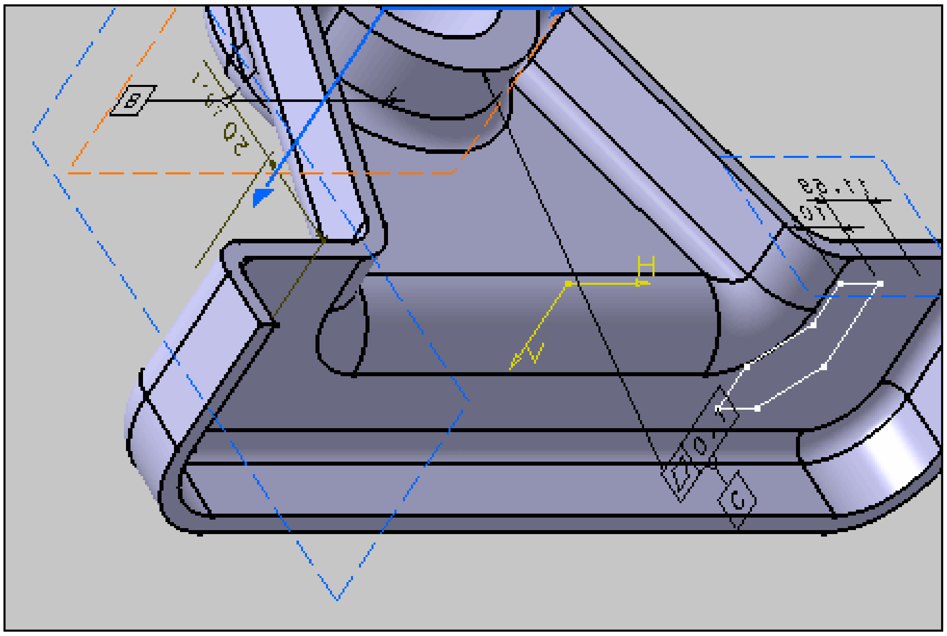
The geometry connection modification will only affect all the annotation that are directly or indirectly applied to the User Surface.1 feature.



The validity status is KO for the **Linear Size.1**, **Datum Reference Frame.1** and **Position surfacic profile.1** annotations because they will not have any meaning if the modification is applied.

10. Click **OK**.

The annotations are now connected on the new surface and the dimension annotation is turned to invalid because it has no more meaning for the new geometry.



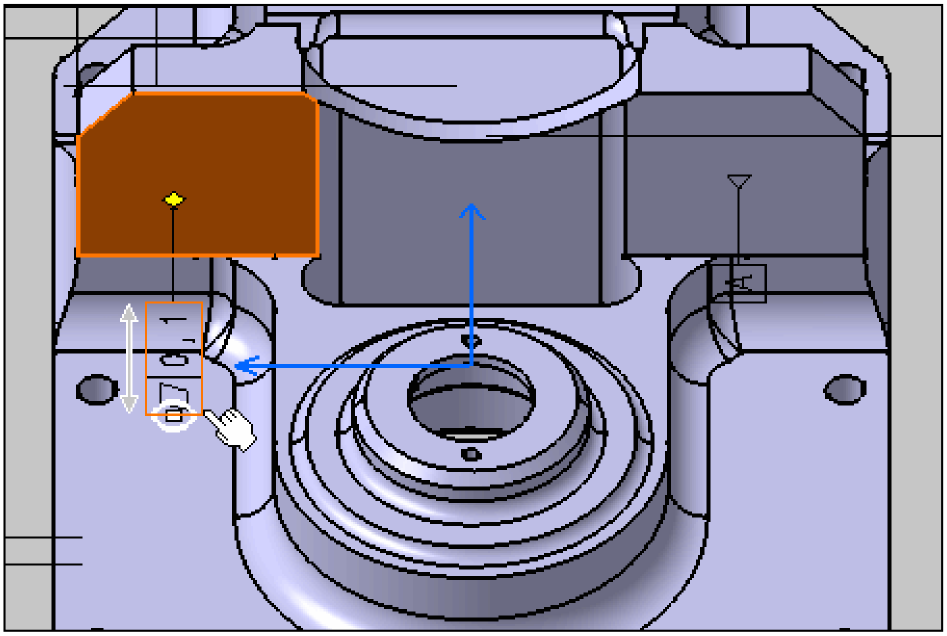
Adding Geometry

This task shows you how to add geometry to the user surface of an annotation.
See 3D Annotations and Annotation Planes concept.

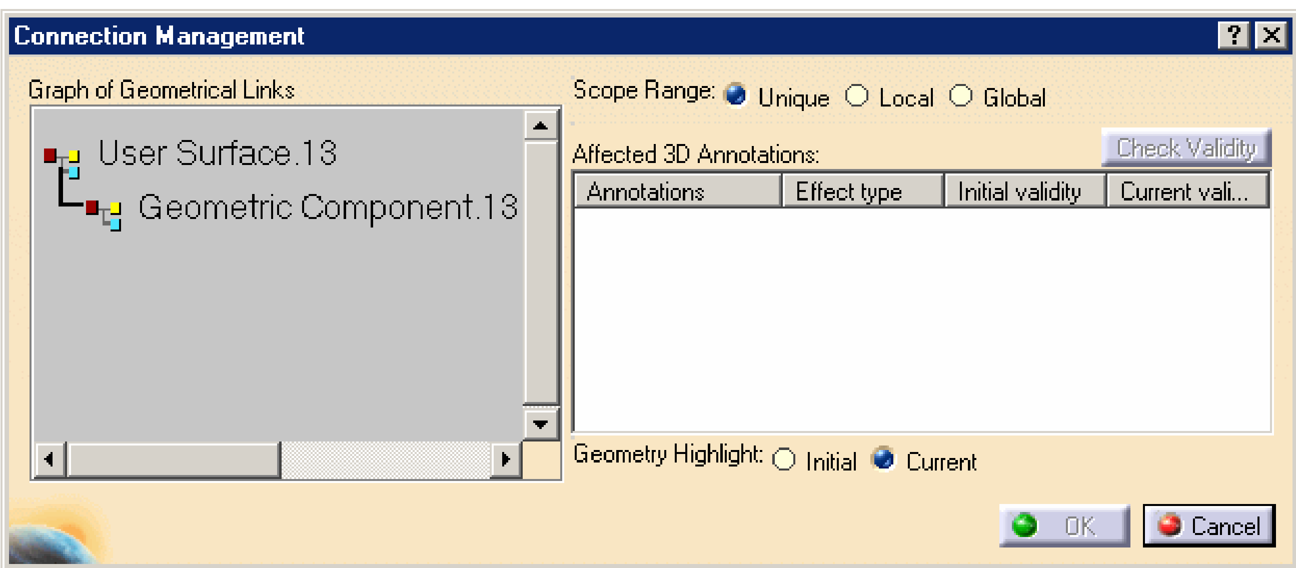
Open the Annotations_Part_02.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Right-click the annotation as shown on the part and select the **Associated Geometry ->Geometry Connection Management** from the contextual menu.



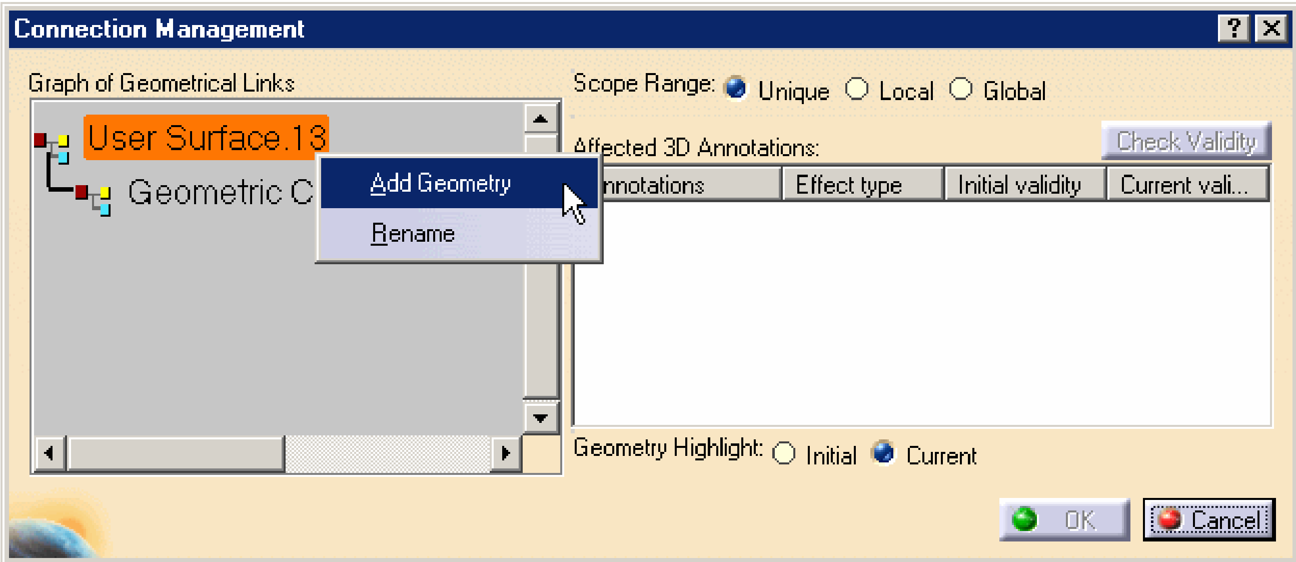
The **Connection Management** dialog box appears.



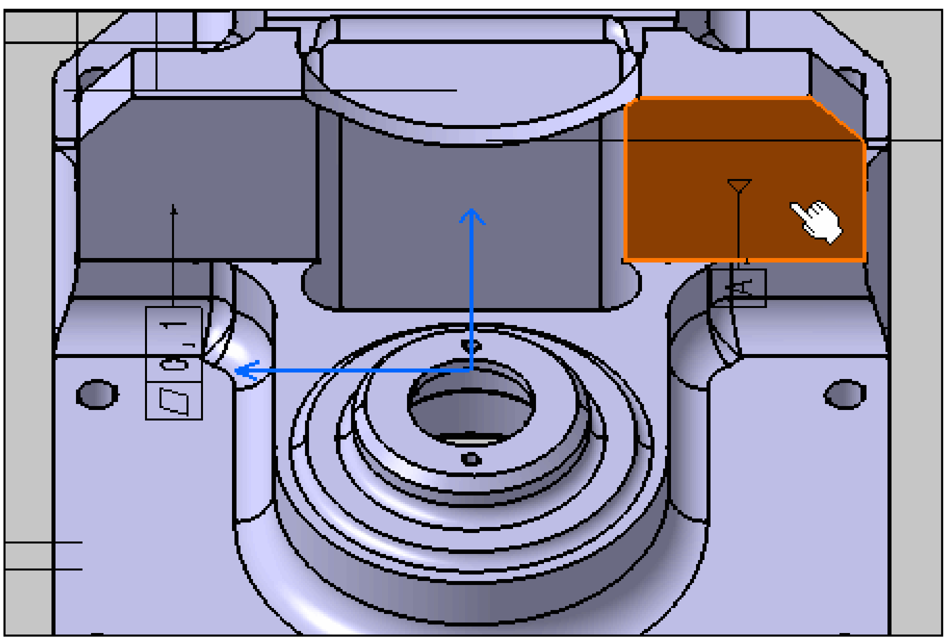
2. Check that **Unique** option is activated in **Scope Range**.

3. Right-click **User Surface.13** in the **Graph of Geometrical Links** as shown and select the **Add Geometry** from the contextual menu.

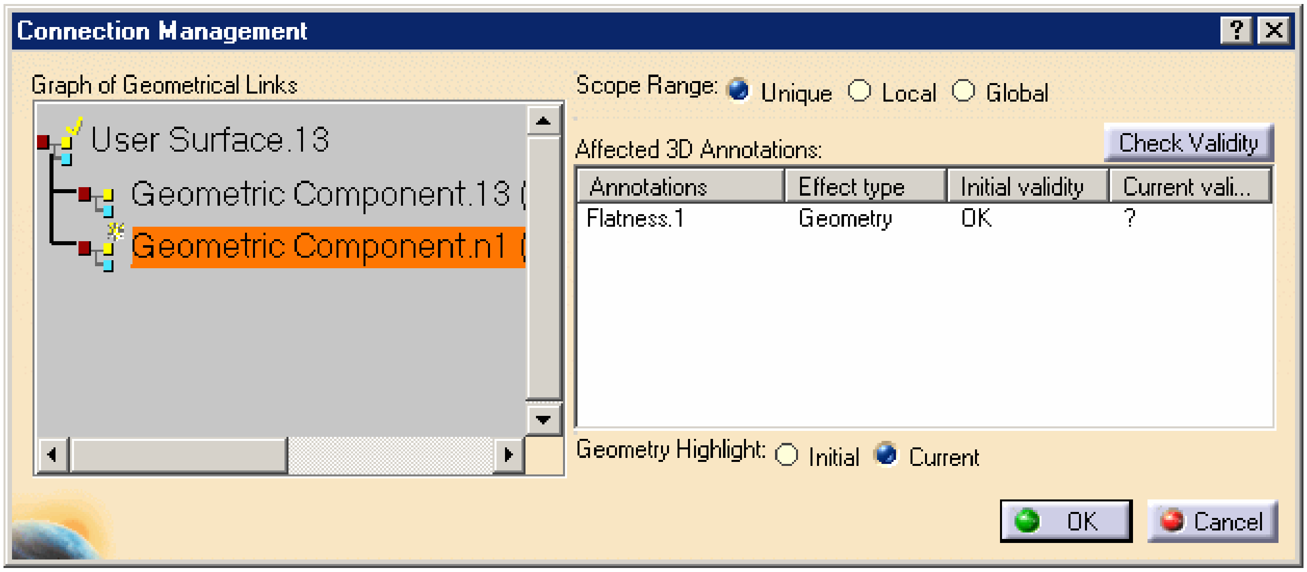
It will add a new geometric component to the user surface **User Surface.13** feature and prompt you to select the new geometrical element to be linked to the annotations.



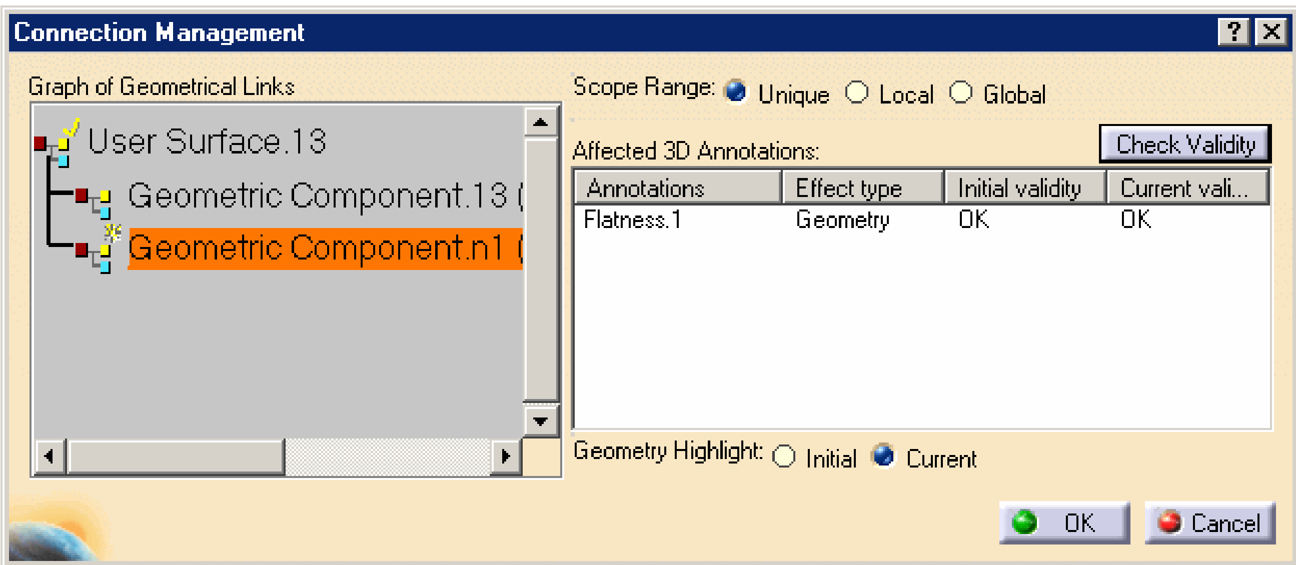
4. Select the surface as shown on the part.



The **Connection Management** dialog box displays the new added geometry to the structure: Geometric Component.1

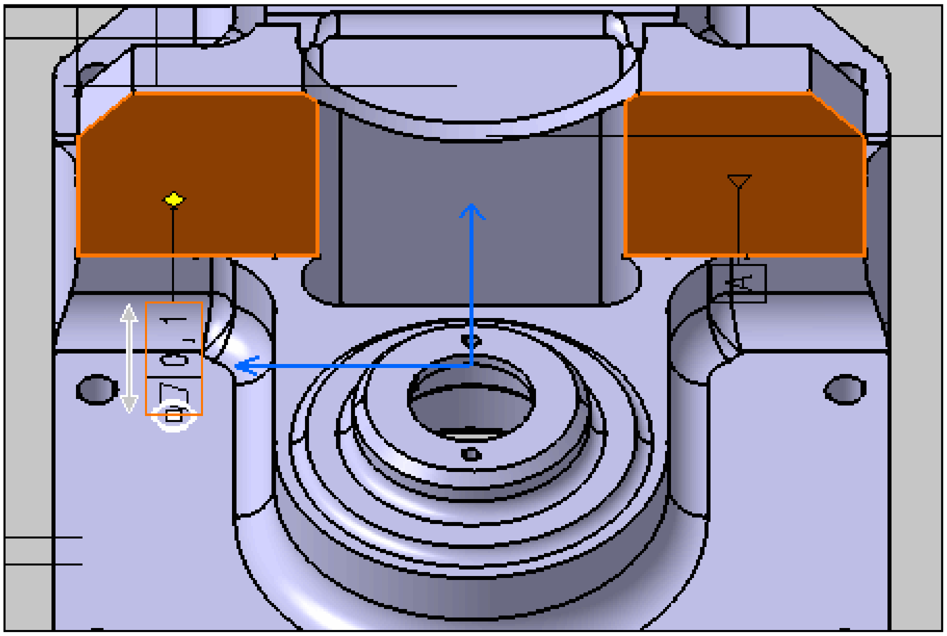


5. Click the **Check Validity** command to check the new geometry component validity relative to the selected annotation.



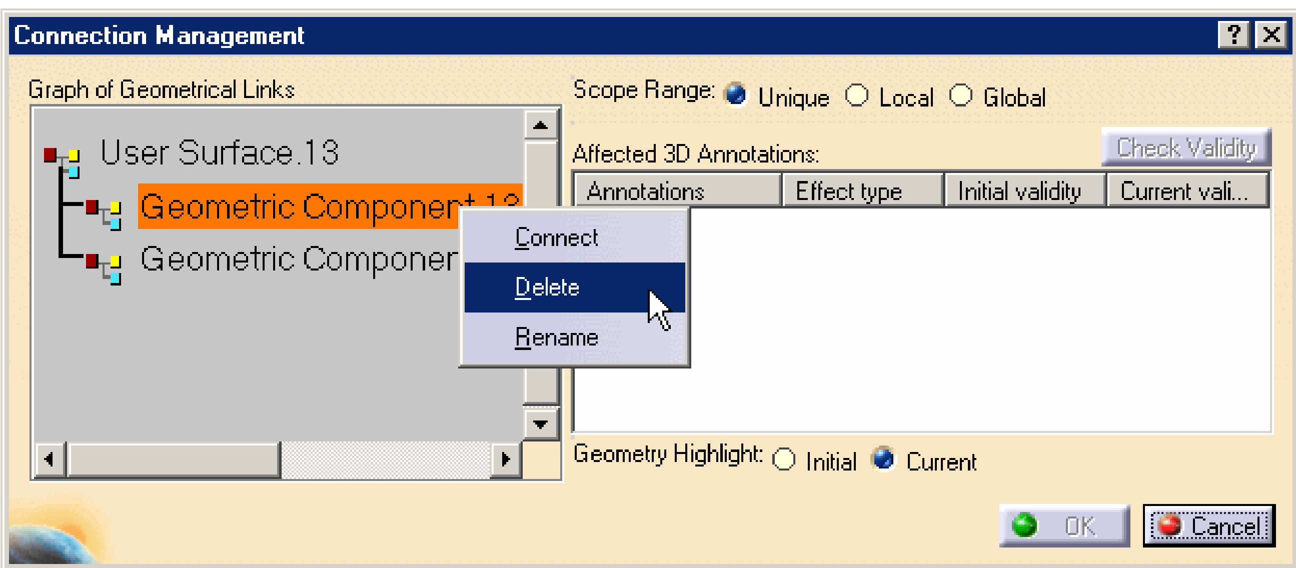
6. Click **OK**.

The two geometric components are now link to the annotation.



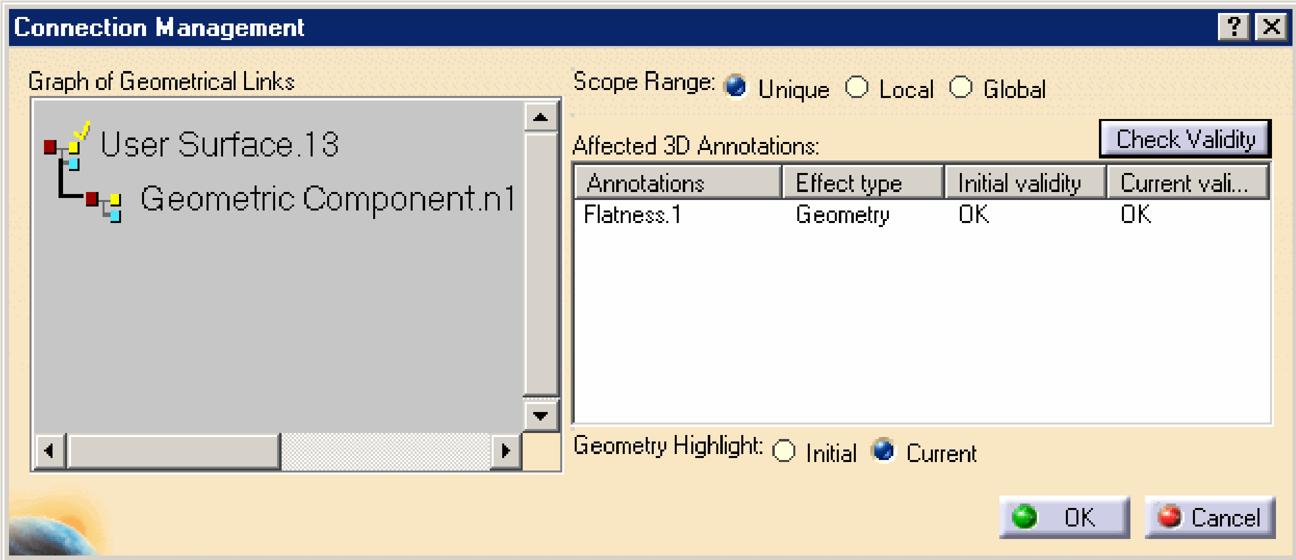
7. Right-click the annotation again and select the **Geometry Connection Management** contextual menu.

8. Right-click **Geometric Component.2** in the **Graph of Geometrical Links** as shown and select the **Delete** contextual menu.



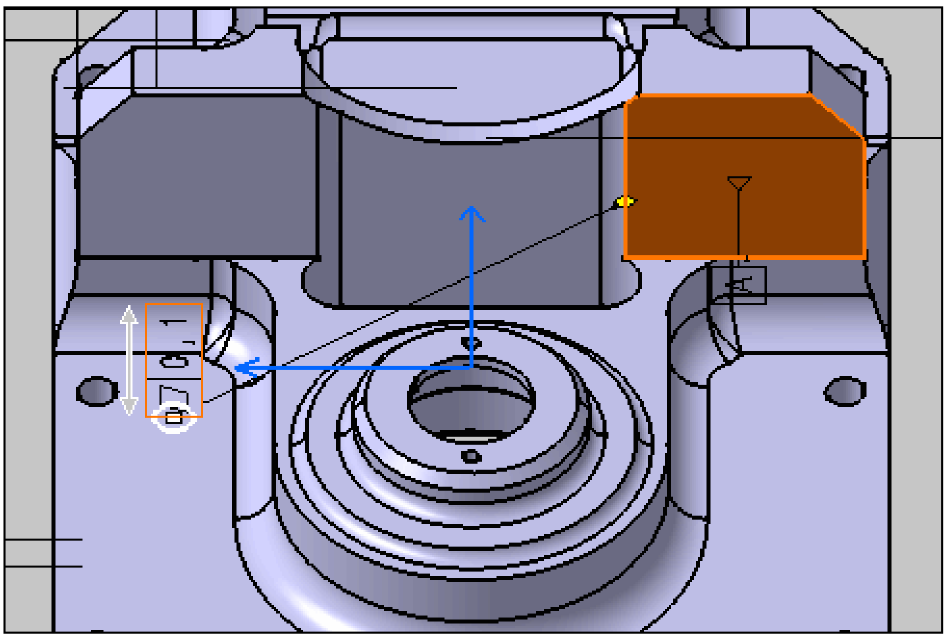
9. Click the **Check Validity**

The **Connection Management** dialog box displays the updated structure.



10. Click **OK**.

Only one geometric component is now linked to the annotation.



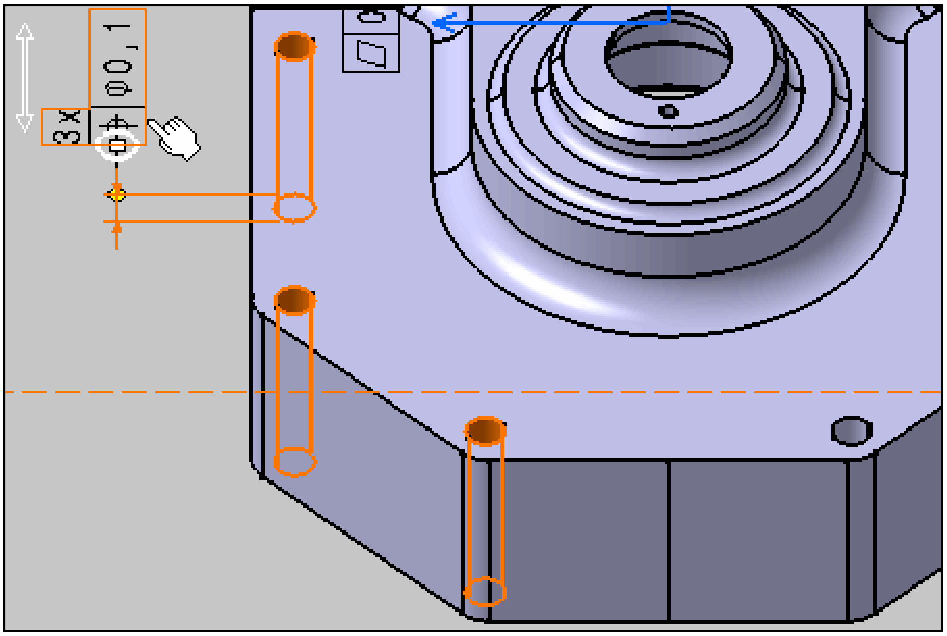
Adding Components

This task shows you how to add a user surface to the group of surfaces of an annotation.
See 3D Annotations and Annotation Planes concept.

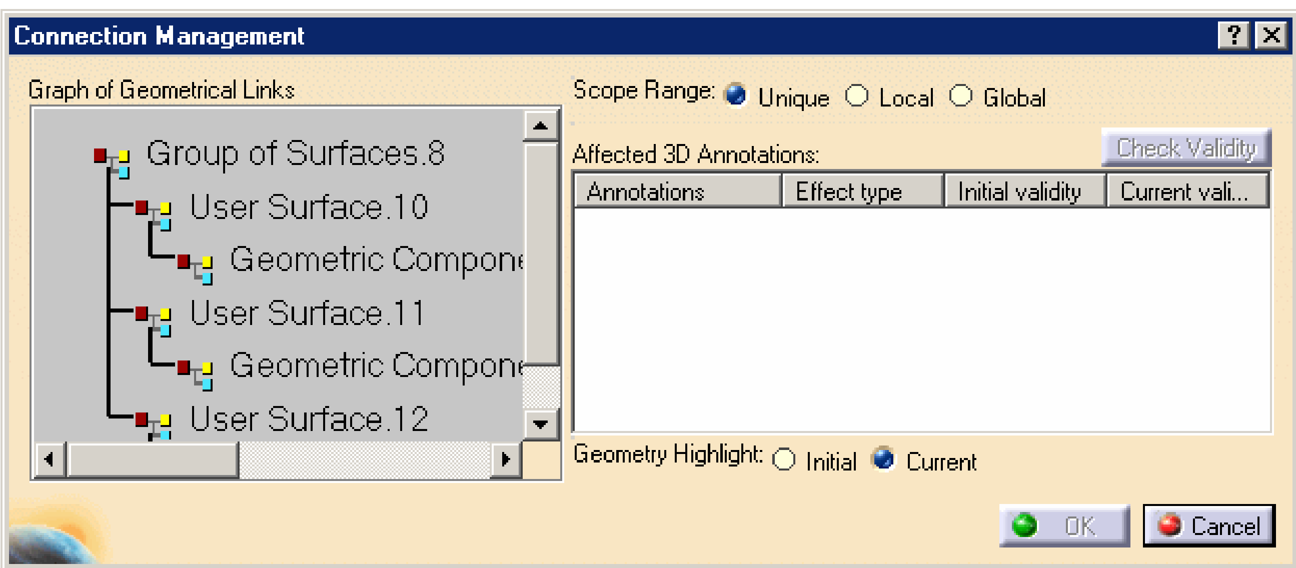
Open the Annotations_Part_02.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Right-click the annotation as shown on the part and select the **Associated Geometry ->Geometry Connection Management** from the contextual menu.



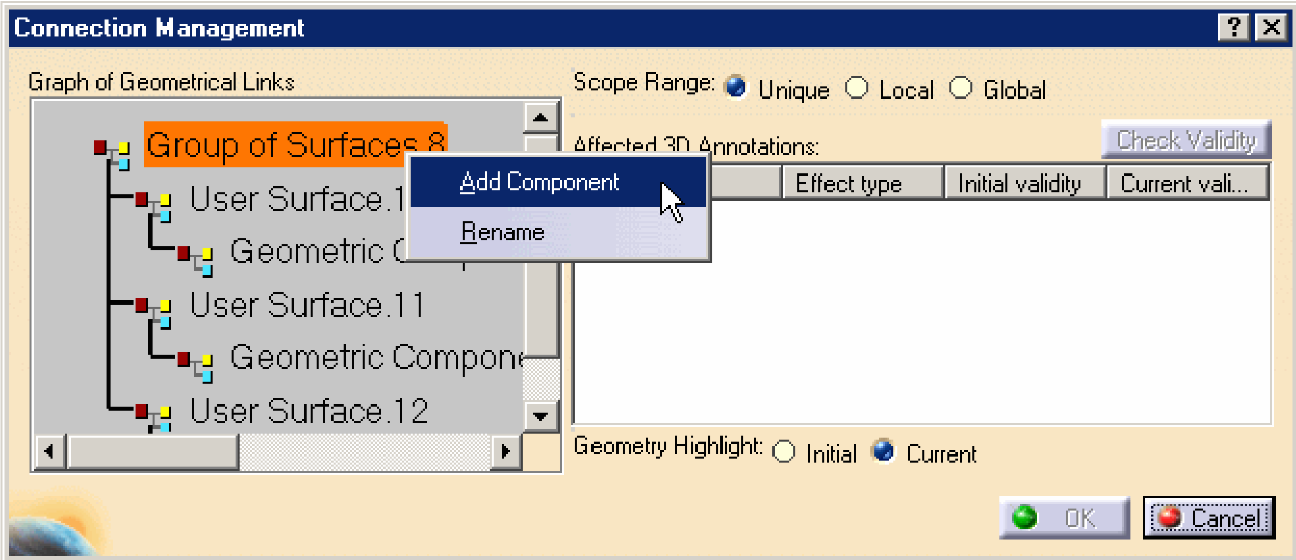
The **Connection Management** dialog box appears.



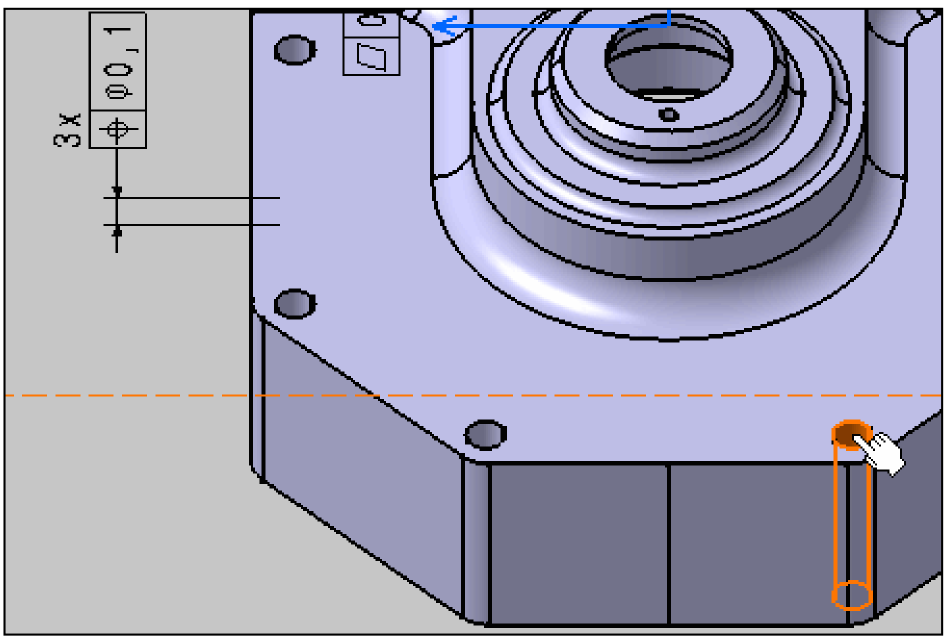
2. Check that **Unique** option is activated in **Scope Range**.

3. Right-click **Group of Surfaces.8** in the **Graph of Geometrical Links** as shown and select the **Add Component** contextual menu.

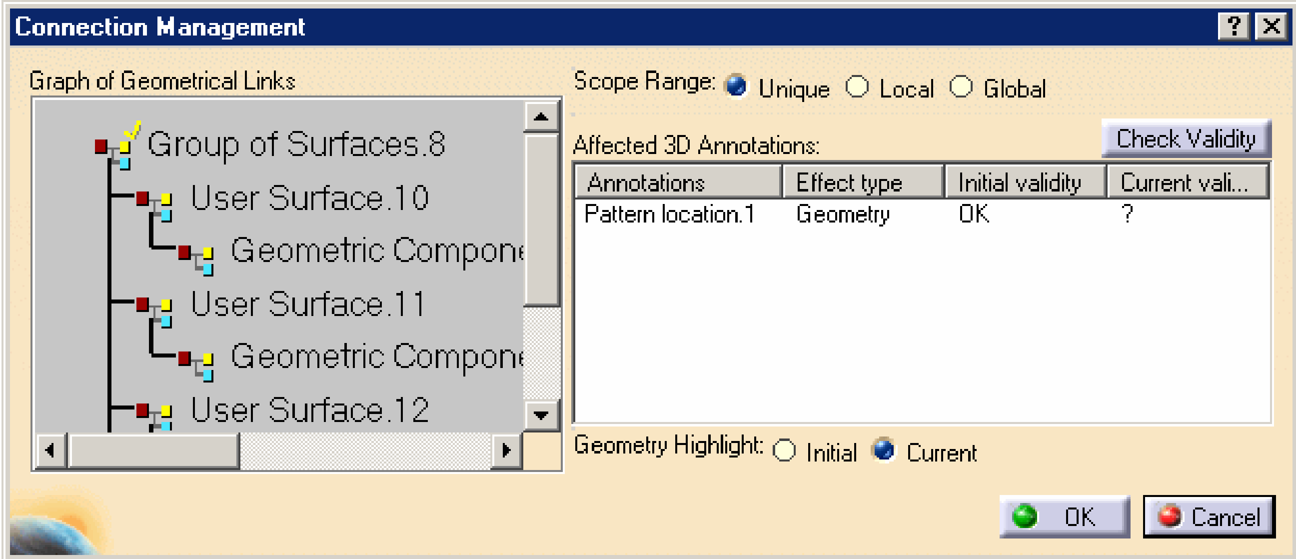
It will add a new component to the group of surfaces **Group of Surfaces.10** feature and prompt you to select the new geometrical element to be linked to the annotations.



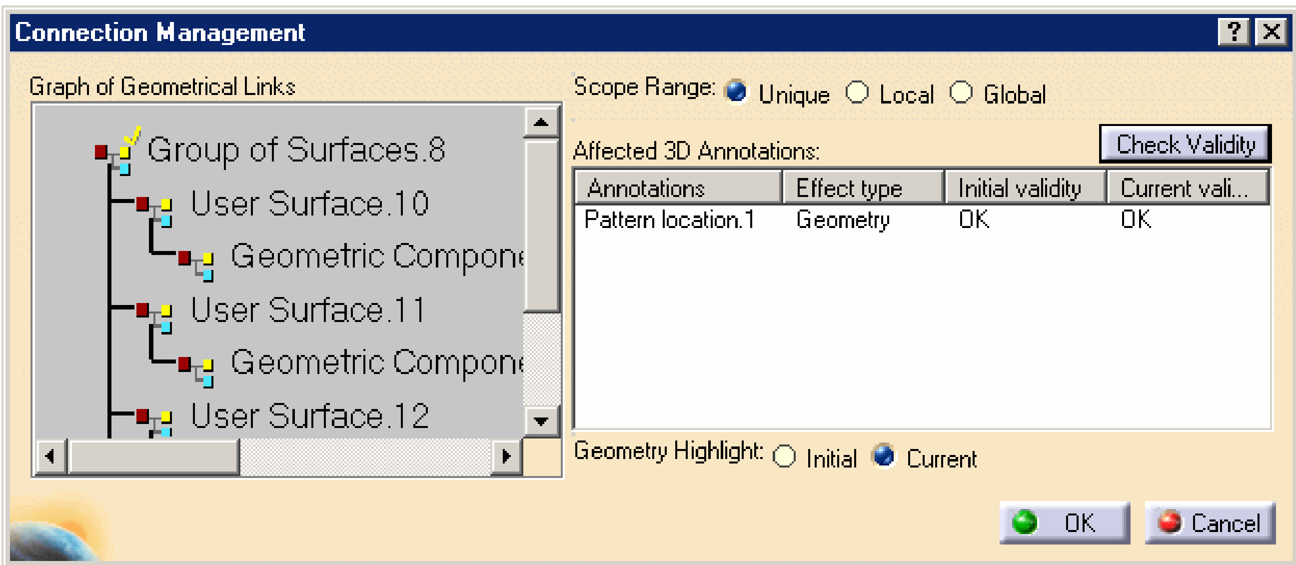
4. Select the hole surface as shown on the part.



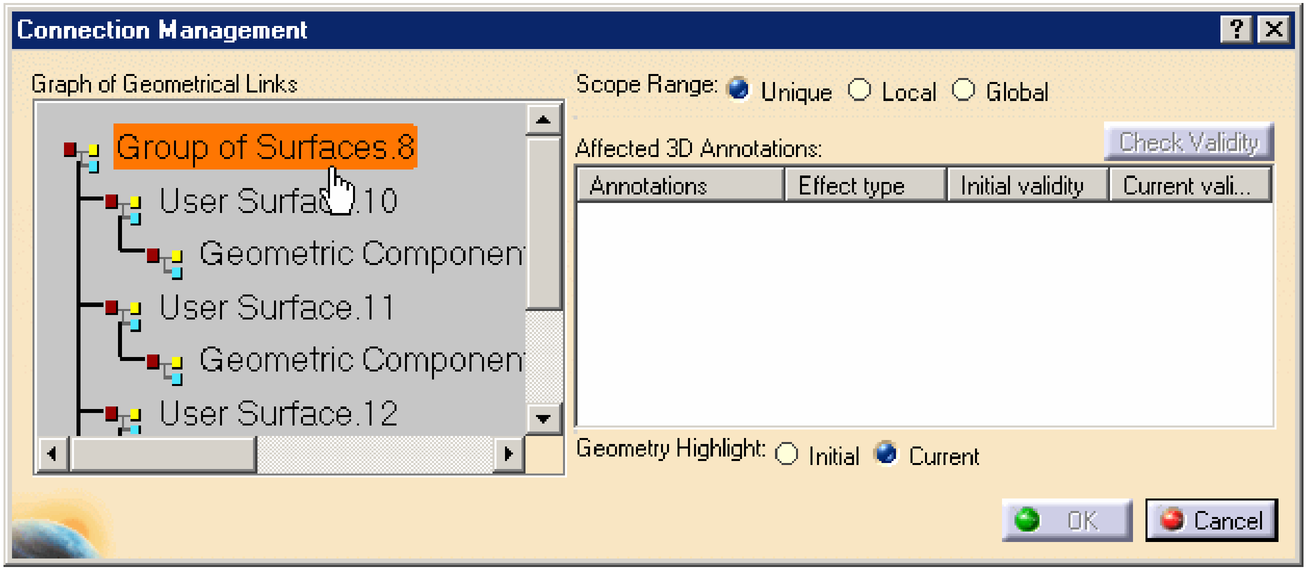
The **Connection Management** dialog box displays the new connected geometry: Geometric Component.1



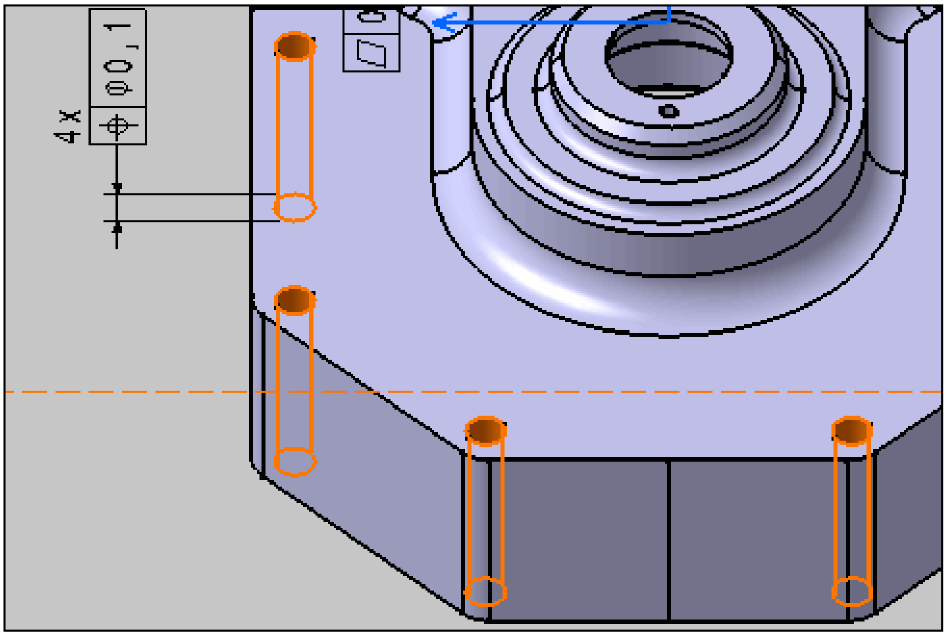
5. Click the **Check Validity** command to check the new geometry component validity relative to the selected annotation.



6. Select **Group of Surfaces.8** in the **Graph of Geometrical Links** as shown and select **Current** option in **Geometry Highlight**.

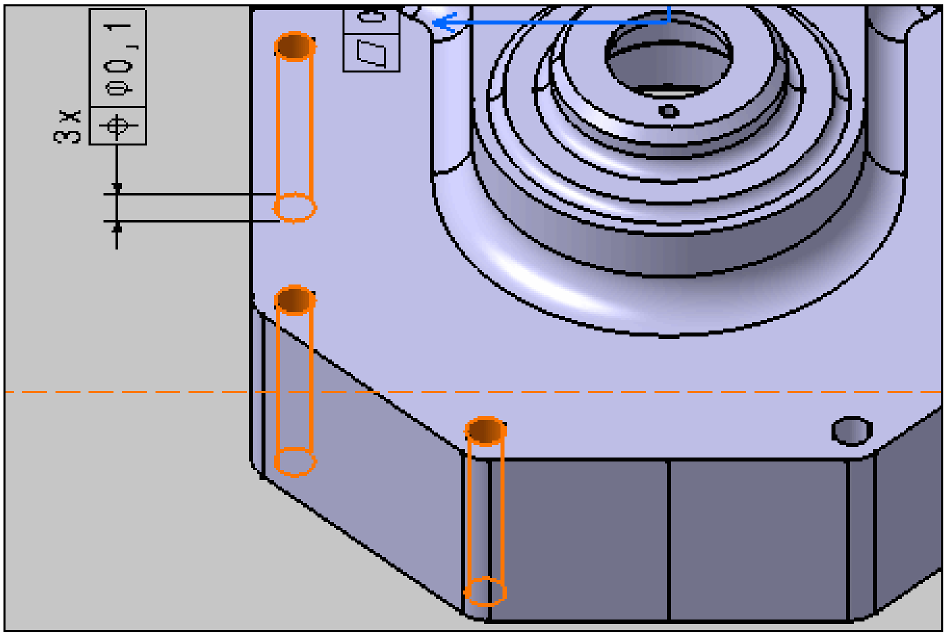


This option shows you the current group of surfaces of the selected annotation after the modification.



7. Select now **Initial** option in **Geometry Highlight**.

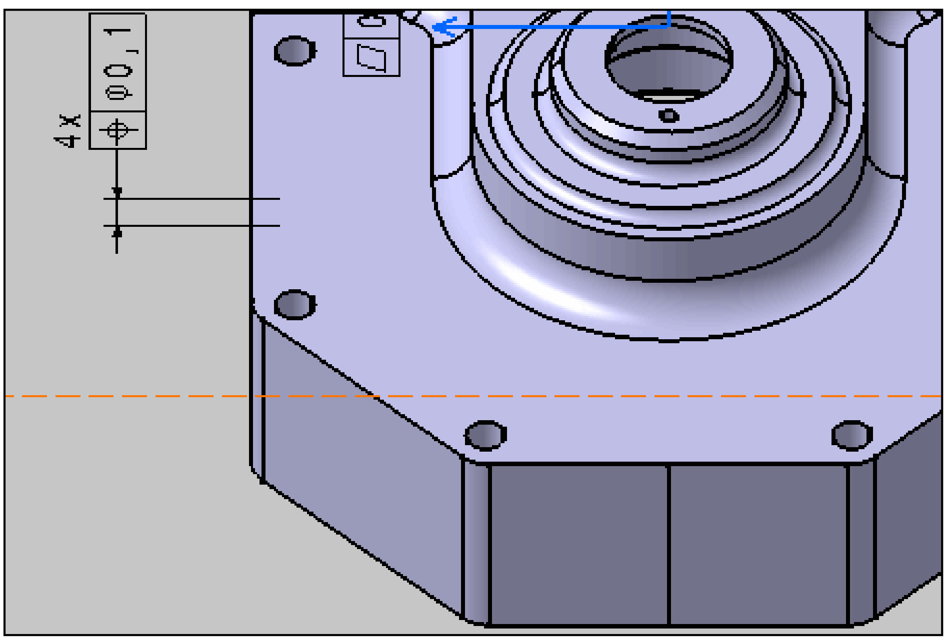
This option shows you the initial group of surfaces of the selected annotation before the modification.



Options in **Geometry Highlight** are always applied to the selection in the **Graph of Geometrical Links**.

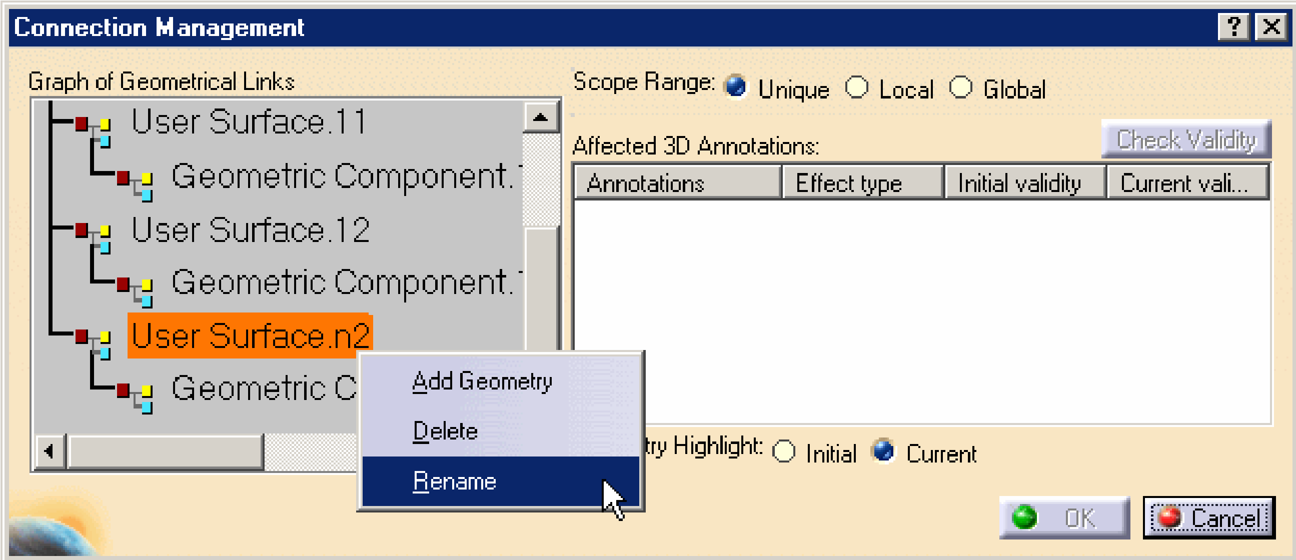
8. Click **OK**.

The four geometric components are now linked to the annotation.



9. Right-click the annotation again and select the **Geometry Connection Management** contextual menu.

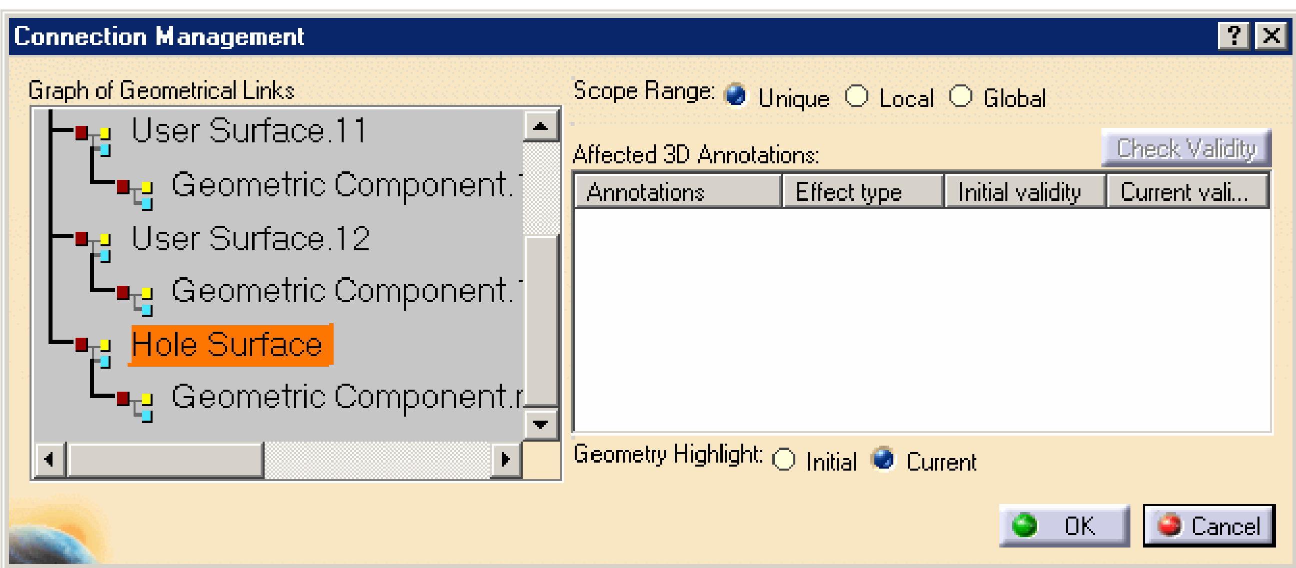
10. Right-click **Geometric Component.1** in the **Graph of Geometrical Links** as shown and select the **Rename** contextual menu.



The **Rename** dialog box appears.

11. Enter the new name: Hole Surface and Click **OK**.

The feature has been renamed.



12. Click **OK**.

Re-specifying Geometry Canonicity

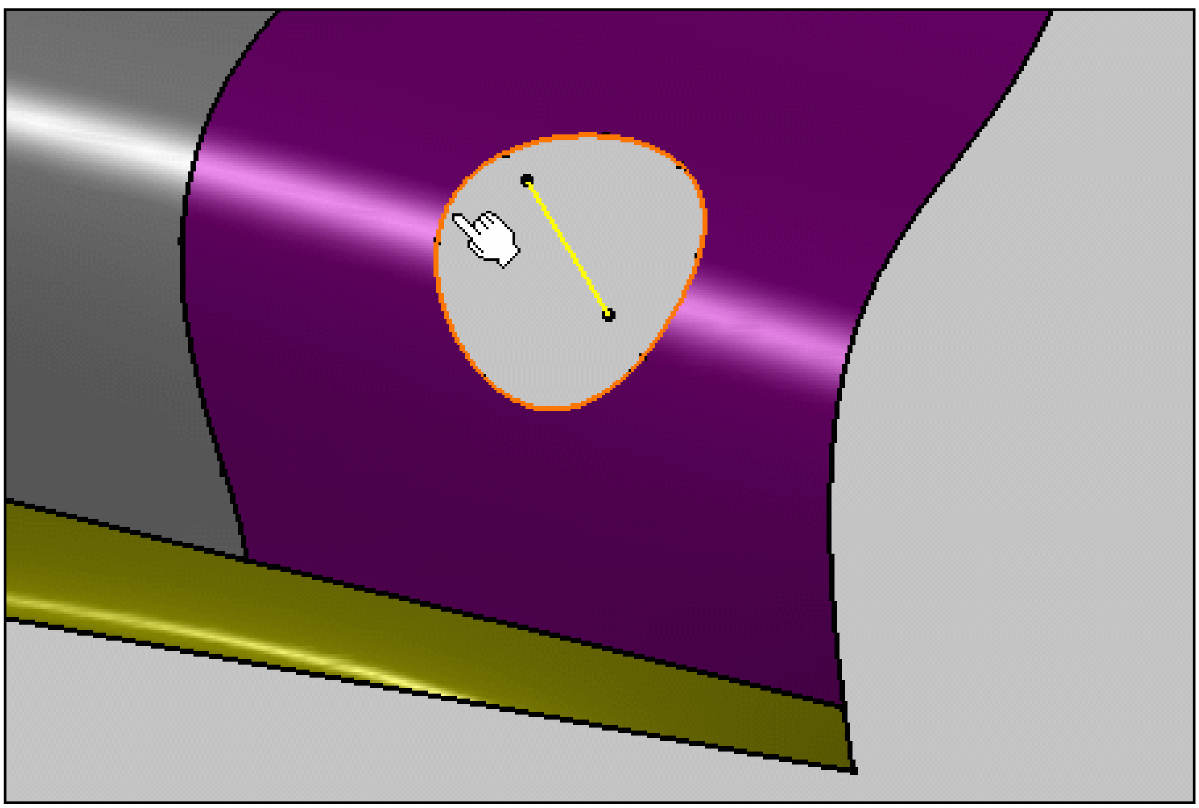
This task shows how to re-define a complex edge into a cylindrical surface and apply a size tolerance to the previous redefined geometry.

Re-specifying geometry canonicity offers to re-define user surface or group of surface properties to avoid recognition problem when selecting a geometry. You can re-define the geometry through an existing annotation only.

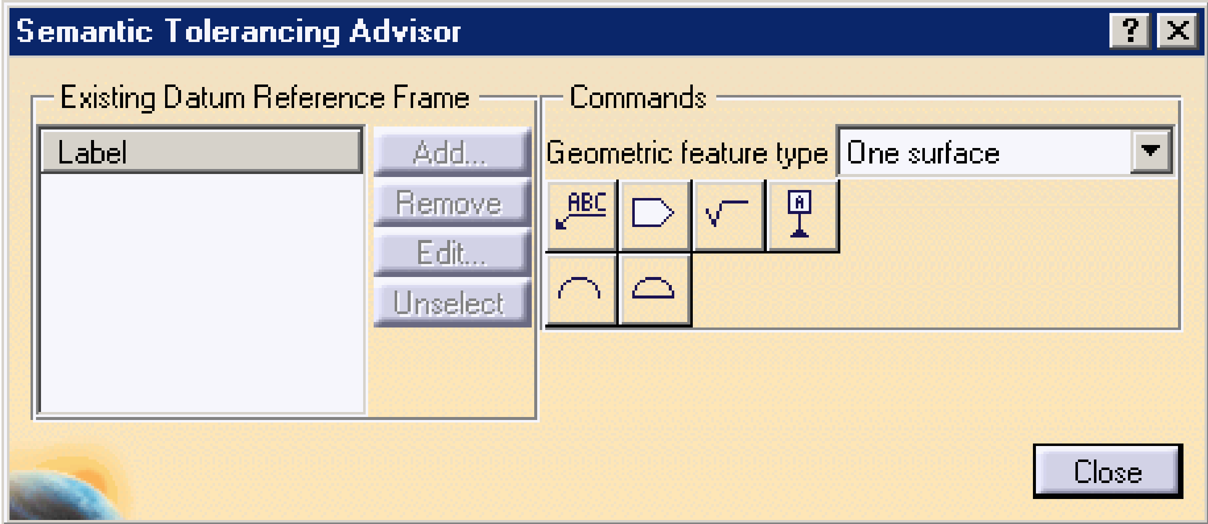
Open the Tolerancing_Annotations_07 CATPart document.

1. Click the **Tolerancing Advisor** icon:

2. Select the complex edge as shown on the part.



The **Semantic Tolerancing Advisor** dialog box appears. Note that no command related to a cylindrical surface is displayed.

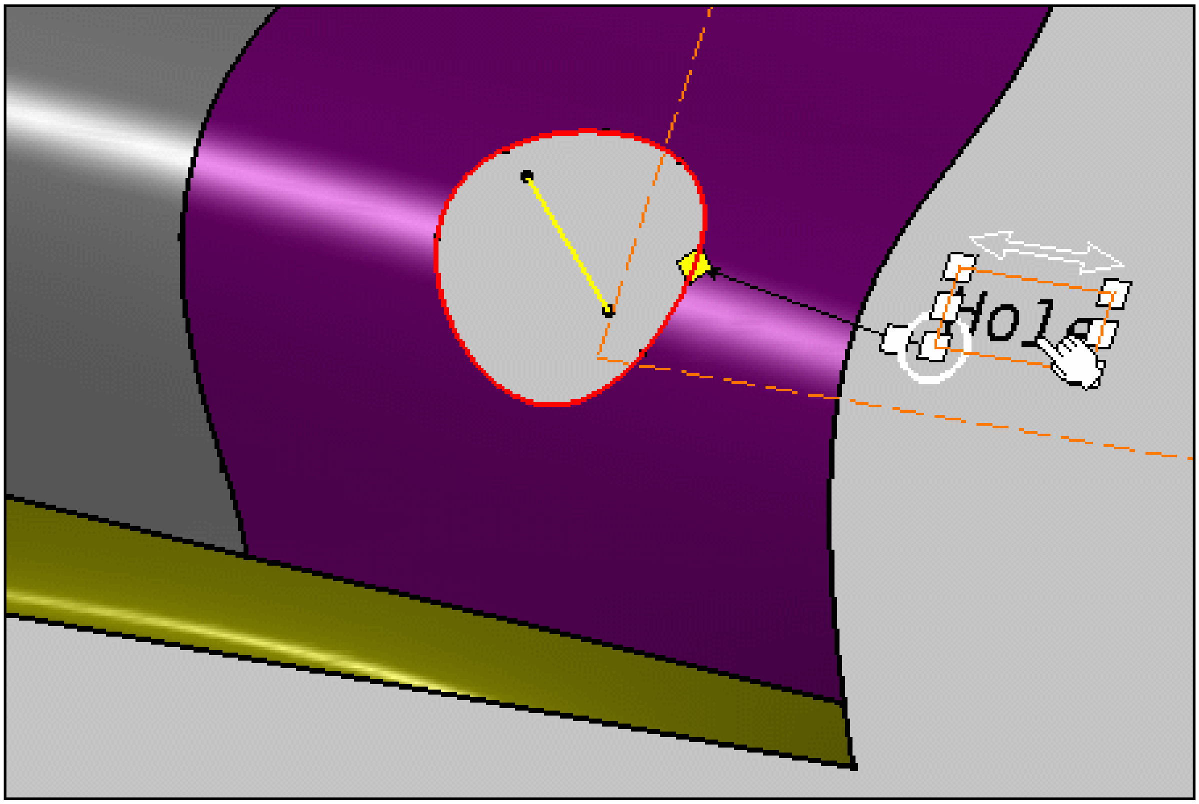


3. Click the **Text with Leader** icon (One surface): 

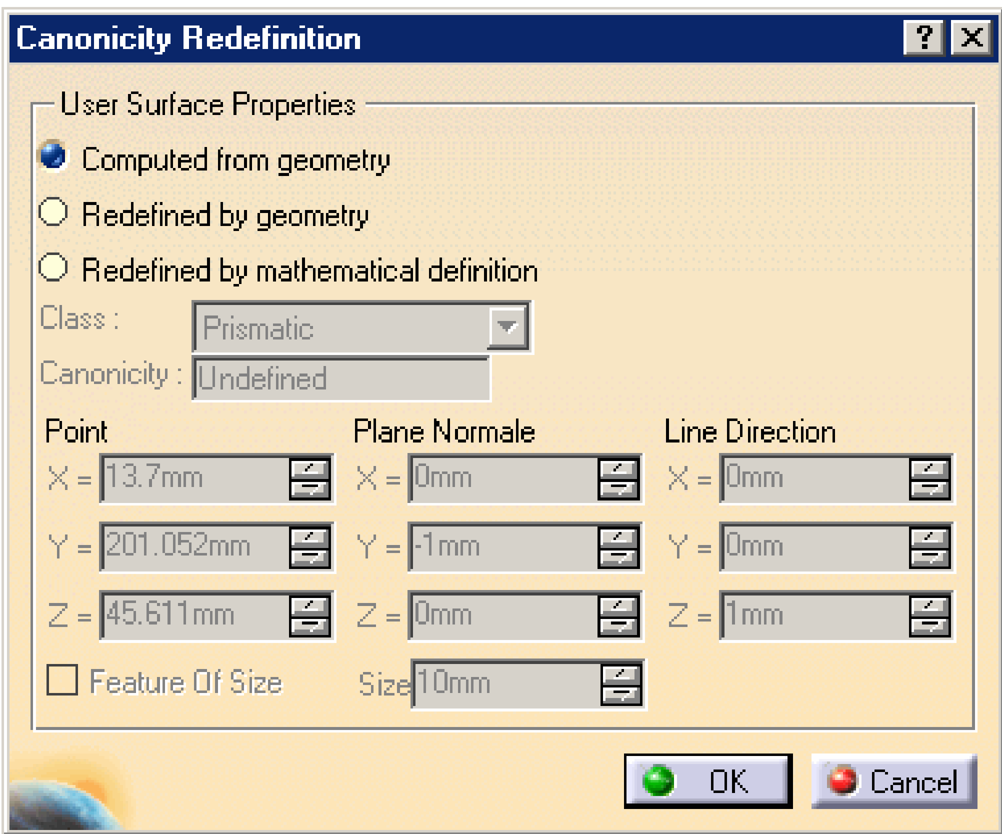
4. Enter Hole in the **Text Editor** dialog box field when appears.

5. Click **OK** in the **Text Editor** dialog box and **Close** in the **Semantic Tolerancing Advisor** dialog box.

6. Right-click the annotation as shown on the part and select the **Associated Geometry -> Geometry Canonicity Re-specification** from the contextual menu.

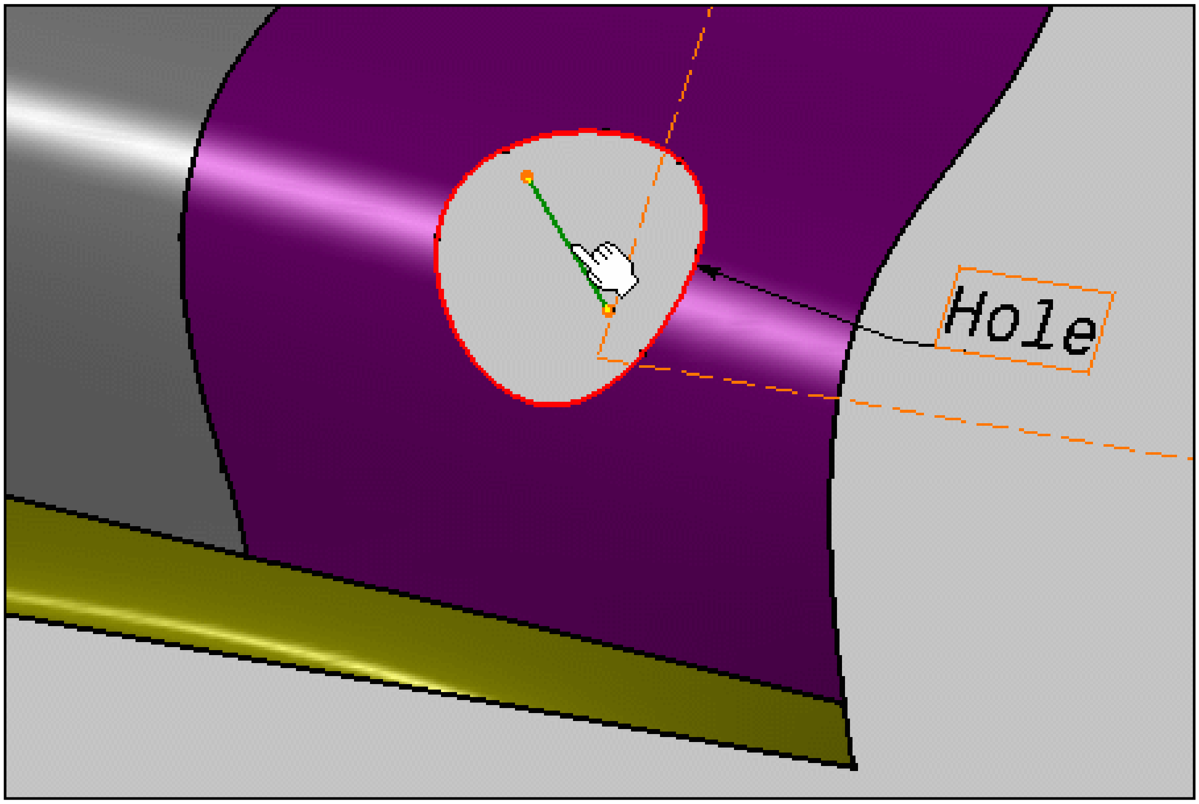


The **Canonicity Redefinition** dialog box appears.

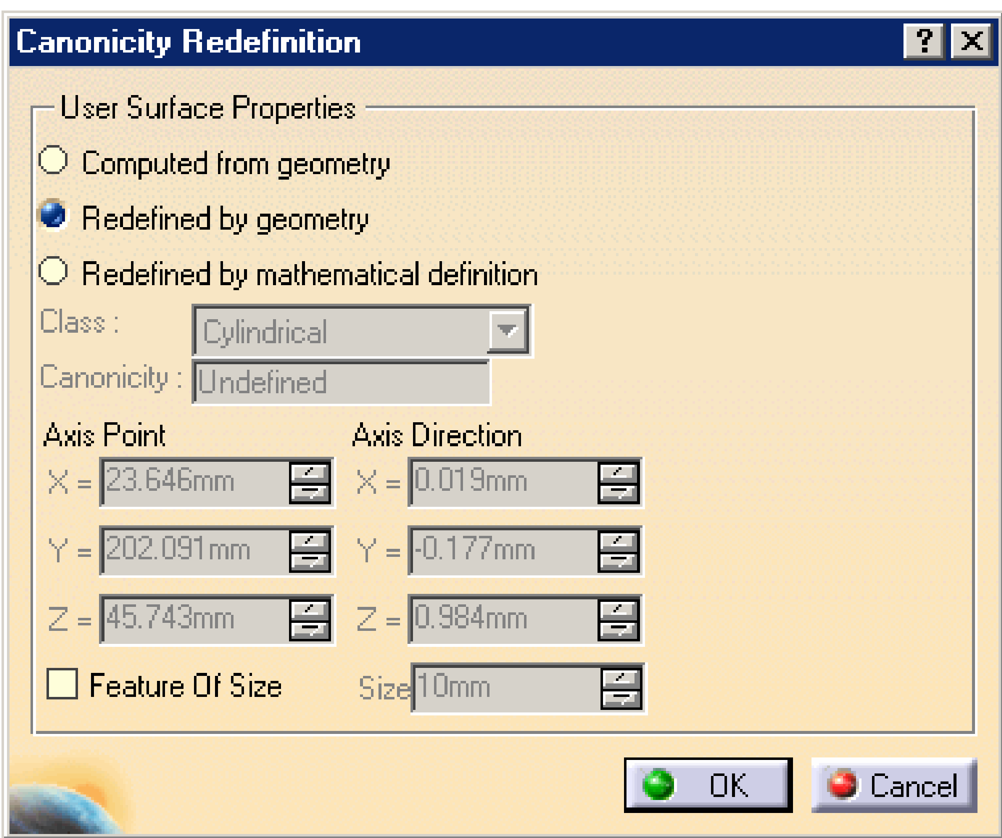


7. Select the **Redefined by geometry** option.

8. Select the line as shown on the part as axis line associated with the cylindrical surface definition.

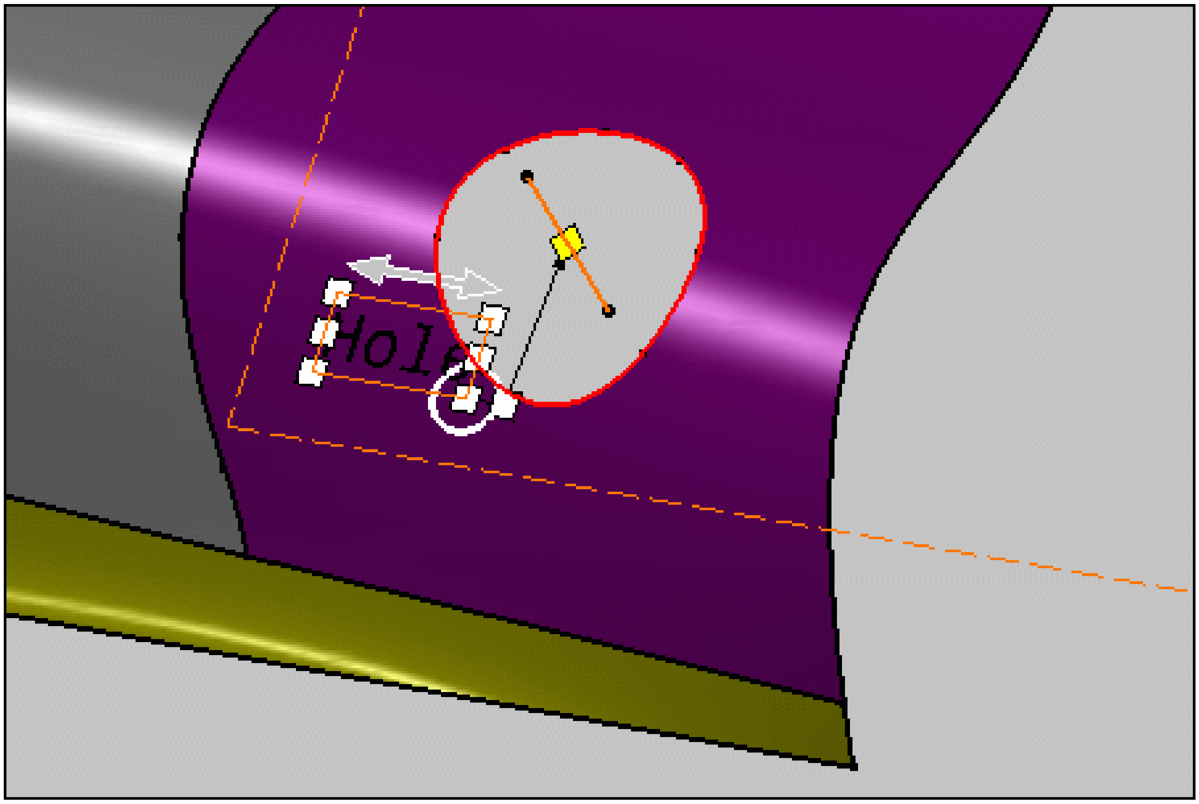


The **Canonicity Redefinition** dialog box is updated.



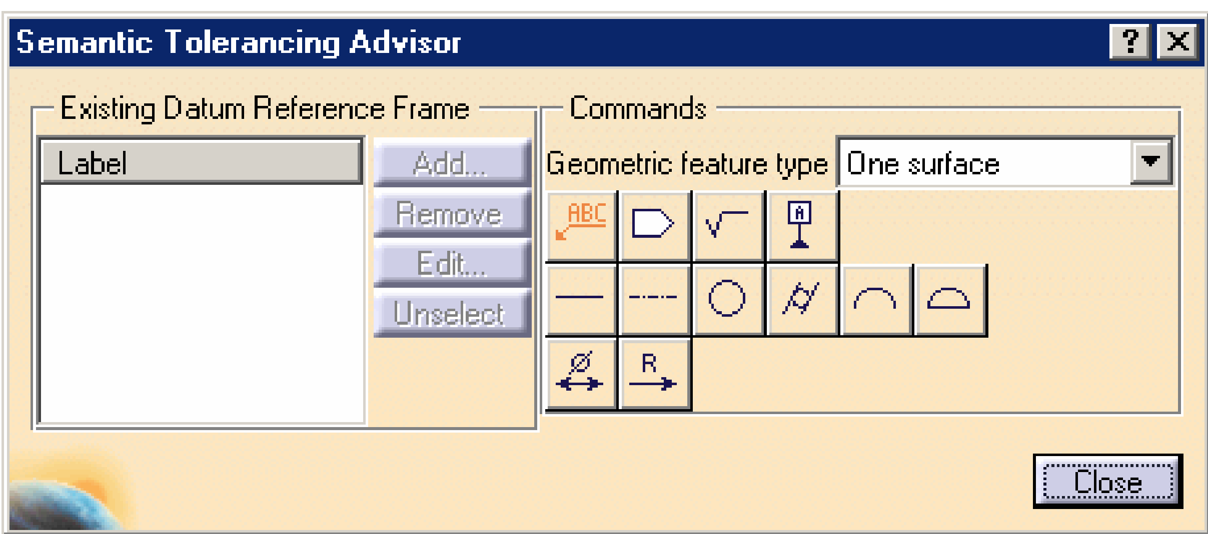
9. Select the **Feature of Size** option, the Size field is now enabled and set its value to 20mm (We "know" the cylindrical diameter) and click **OK**.

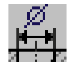
The annotation is now attached to the axis.



10. Click the **Tolerancing Advisor** icon: , the annotation is still selected.

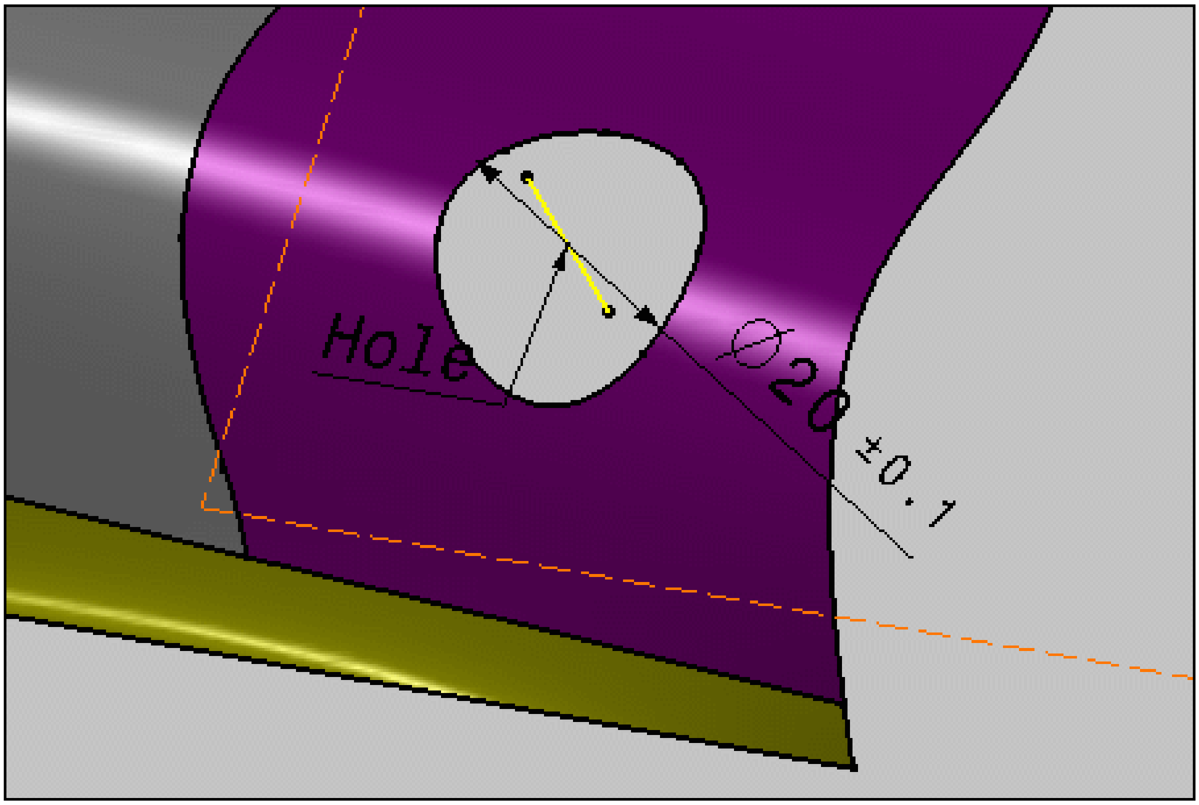
The **Semantic Tolerancing Advisor** dialog box displays now command related to a cylindrical surface.



11. Click the **Diameter** icon (One surface): 

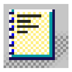
12. Click **OK** in the **Limit of Size Definition** dialog box when appears.

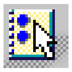
The diameter dimension annotation is created.



13. Click **Close** in the **Semantic Tolerancing Advisor** dialog box.

Reporting Annotations

 Generate a Check Report: click this icon to generate the report.

 Customize the Reporting: click this icon and select the desired options.

Generating a Check Report

This tasks shows you how to generate a report checking whether tolerancing rules are respected or not. These rules depend on the standard you are using. See Tolerancing settings.

Open the Annotations_Part_02.CATPart document.

1. Make sure the options **Html** and **Both** are activated in the **Tolerancing Rule Settings** dialog box.

   For more information, refer to Customizing the Reporting.

2. Click the **Report** icon

   The application generates the report in the browser you usually use and displays it onscreen using the options as specified in the Custom Report command.

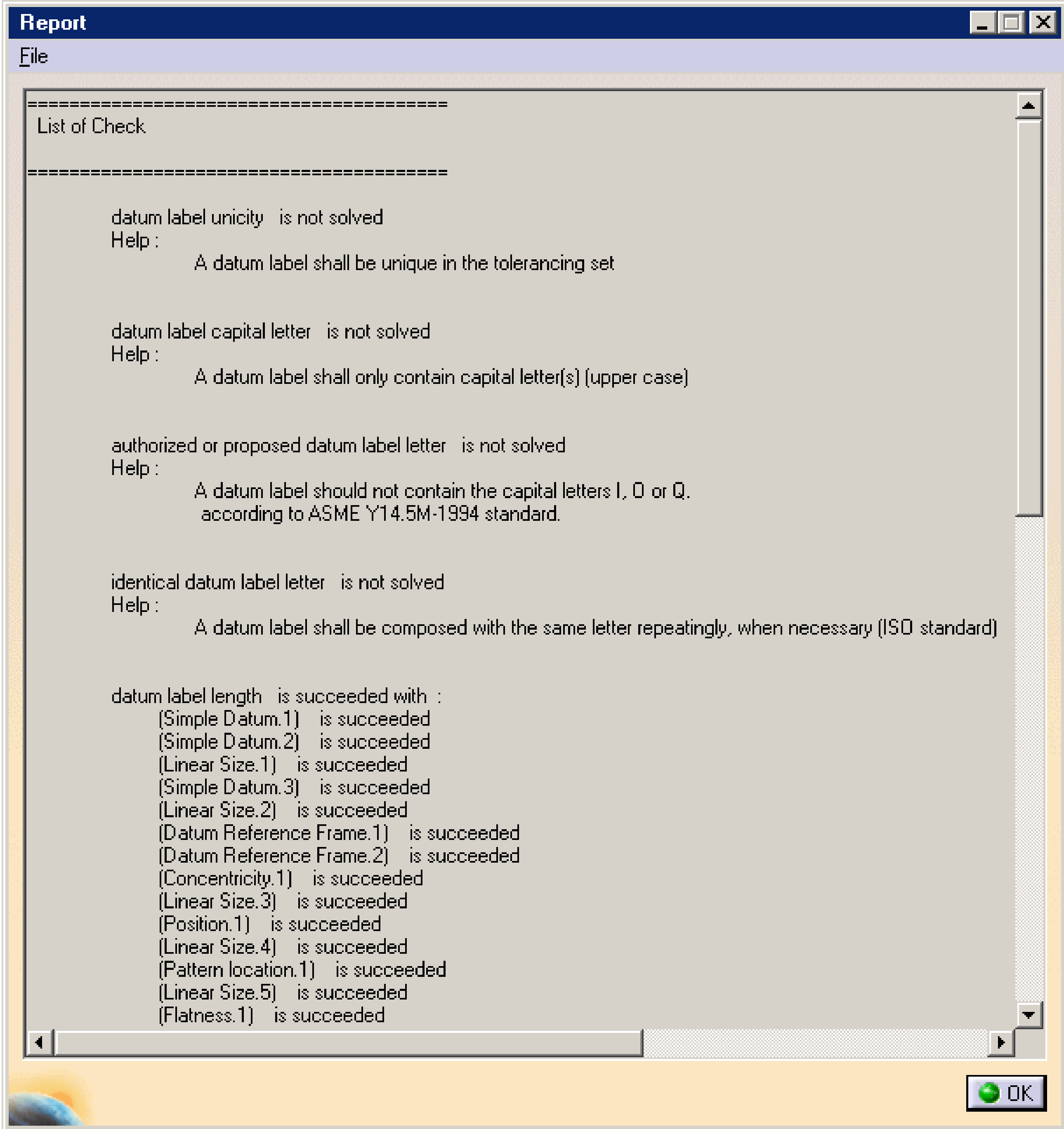
   The file provides you with the path of the CATPart document you are using and the date of generation. In our example, all the rules are respected as mentioned by the green symbols and the 100% success message.

3. Click any rule name to obtain detailed information

4. To generate the other type of report, use the Customize Report command to set the options **File** and **Both**.

5. Click the **Report** icon

   The application generates the report.

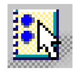


Note that you cannot edit tolerancing rules.

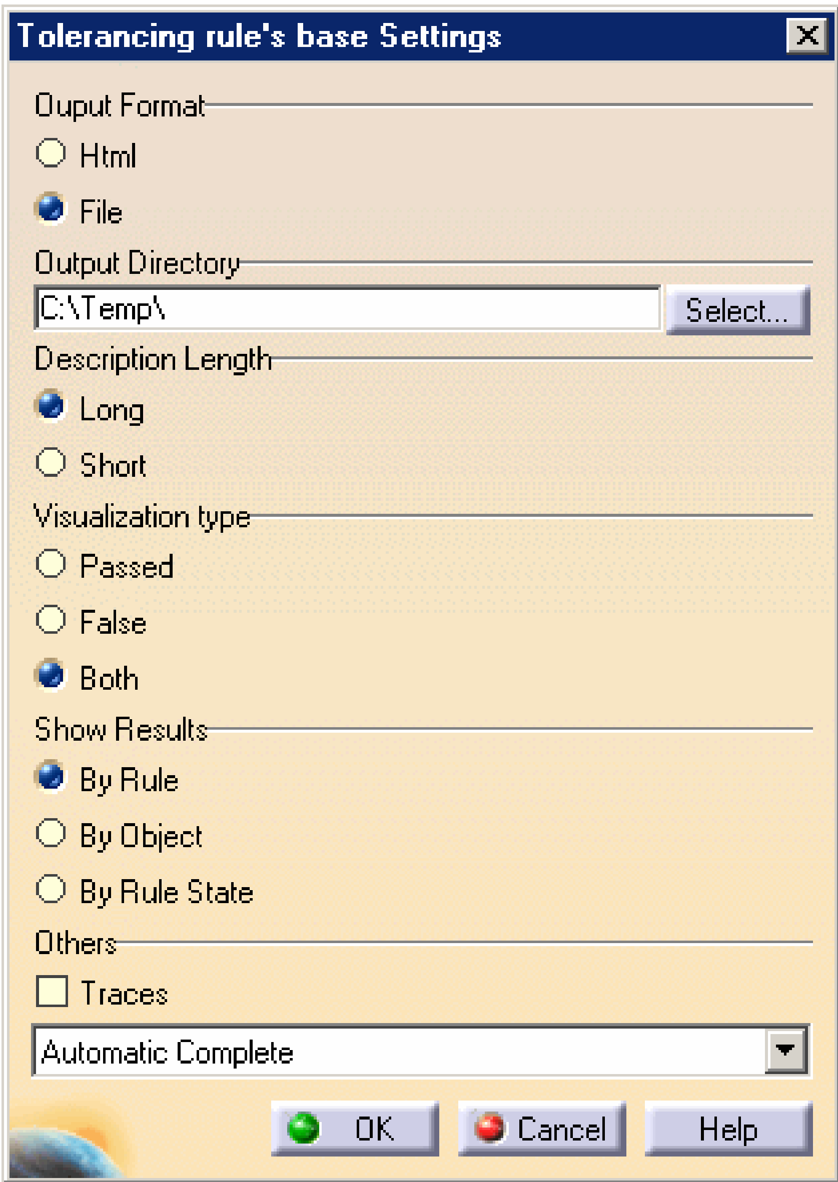
Customizing the Reporting

The data logged in the generated report as well as the report format depend on the rule base settings. This task explains how to specify these settings.

Open any document.

1. Click the **Custom Report** icon 

The **Tolerancing rule's base settings** dialog box appears.



Output Format option:

Html: to generate the report in html format.

File: to generate the report in text format.
In this mode, the **Description Length** and the **Show results** options are activated by default.

Description Length option:

Long: to insert the Help message specified at the check creation.

Short: if you do not need the Help message.

Visualization type option:

Passed: to include in the report only information about the features for which the checks are valid.

False: to include in the report only information about the features for which the checks are invalid.

Both: to include in the report information about all the features on which a check has been applied.

Show Results option:

By Rule: to organize your report data by rule in the file.

By Object: to organize your report data by object.

By Rule: State to organize your report data by rule state.

Others option:

Traces: to display the steps of the solve process.

Automatic Complete: to perform an initialization and a solve operation on the objects whenever the part is updated.

Automatic Optimized: to perform a new solve on the last changes.

Manual Solve: to perform a manual solve.

2. Click **OK** to apply the settings to the rule base.

Unless you want to modify the check report characteristics, you don't have to re-specify the settings each time you generate a report.

Annotation Associativity

Different types of associativity characterize the Functional Tolerancing & Annotation application, such as associativity between the 3D part and the navigation tree for example.

Annotation associativity lets you highlight the geometrical element, Part Design feature or Generative Shape Design feature that is related to an annotation. You can turn an annotation into the default annotation.

Query 3D Annotations: activate or deactivate this icon.

Create an Automatic Default Annotation: select the annotation.

Querying 3D Annotations

This task shows you how to highlight the geometrical element, Part Design feature or Generative Shape Design feature that is related to an annotation, and vice-versa. This enables you to know the relationships between annotations and geometry.

You can activate/deactivate the highlight via the **3D Annotation Query Switch On / Switch Off** icon:

This icon is activated by default when opening the workbench for the first time.

Selecting a 3D annotation will highlight:

- The 3D annotation itself.
- All the geometric element that are components of the tolerancing feature (user surface or group of user surfaces) it is applied to.
- All the construction geometry attached to the tolerancing feature and all its components.
- All the framed (basic) dimensions that are related to the specification.
- All the datum feature annotations that are related to the specification (for semantic geometric tolerance only).

Selecting a geometrical element will highlight:

- The geometrical element itself.
- All the annotations that are applied (directly or indirectly) to it.
- All the framed (basic) dimensions that are related to the corresponding specifications.
- All the datum feature annotations that are related to the specifications (for semantic geometric tolerance only).

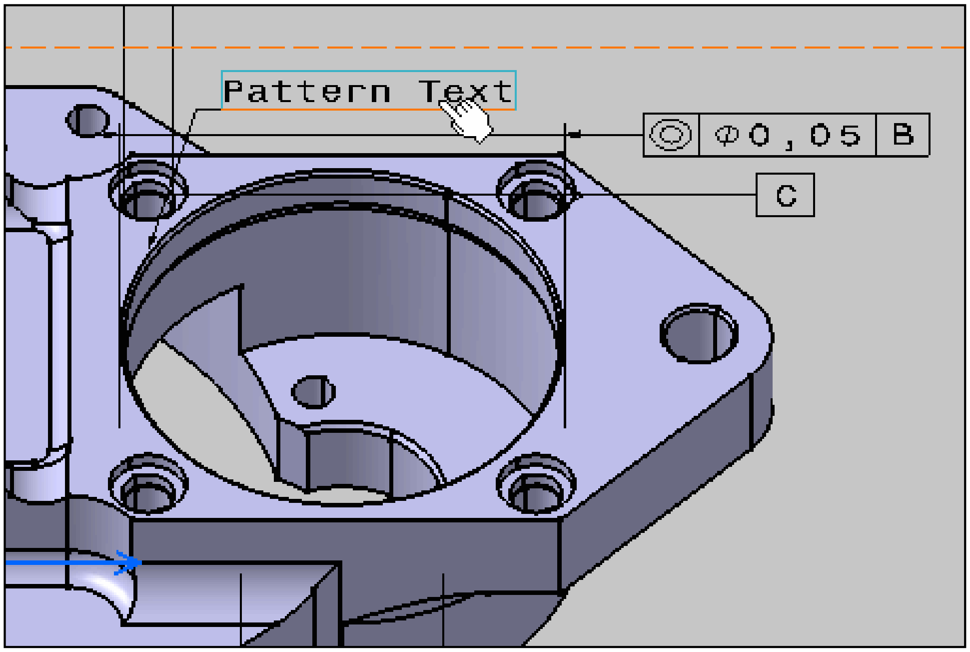
Selecting a Part Design or Generative Shape Design feature or a restricted area in the specification tree will highlight:

- The feature itself.
- All the annotations that are applied (directly or indirectly) to all its geometrical components.
- All the framed (basic) dimensions that are related to the specification.
- All the framed (basic) dimensions that are related to the corresponding specifications.
- All the datum feature annotations that are related to the specifications (for semantic geometric tolerance only).

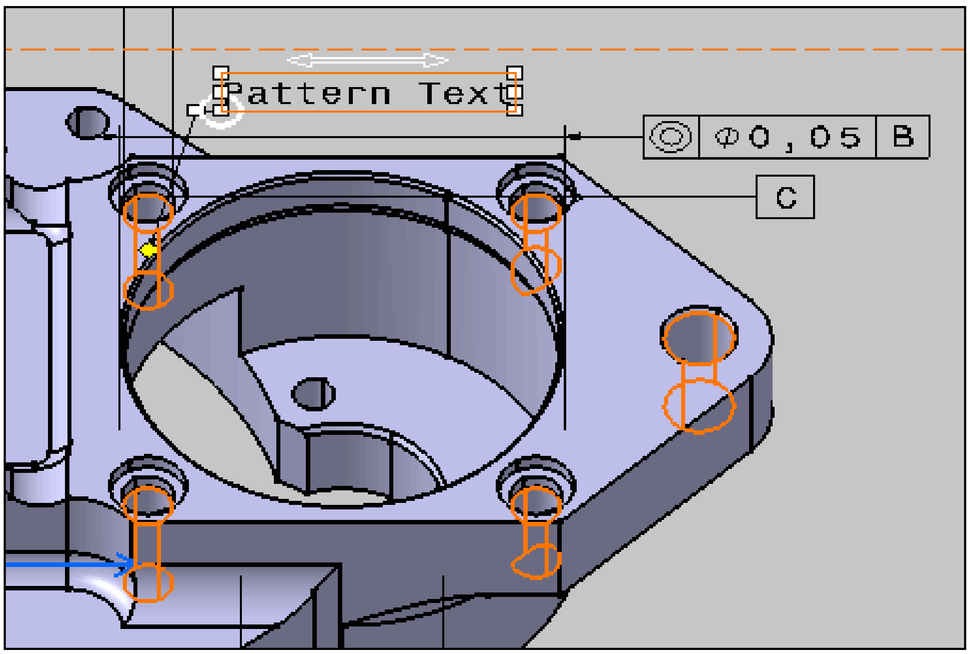
Open the Annotations_Part_02.CATPart document:

- Make sure that the **3D Annotation Query Switch On / Switch Off** icon is activated.
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

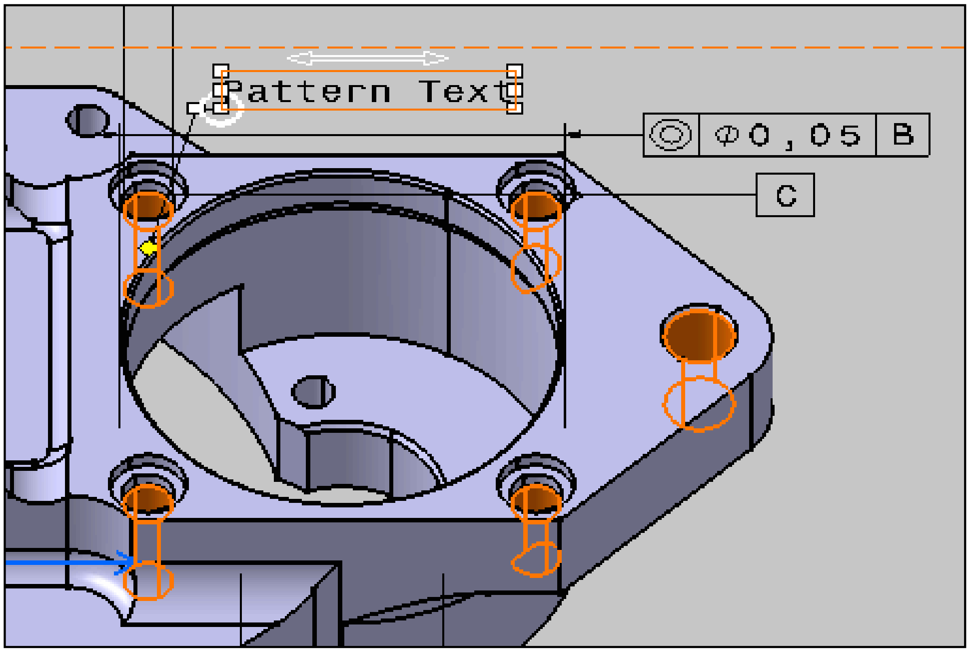
1. Select the annotation as shown:



This location specification is defined on a pattern of holes. Any toleranced hole of the pattern is then highlighted.

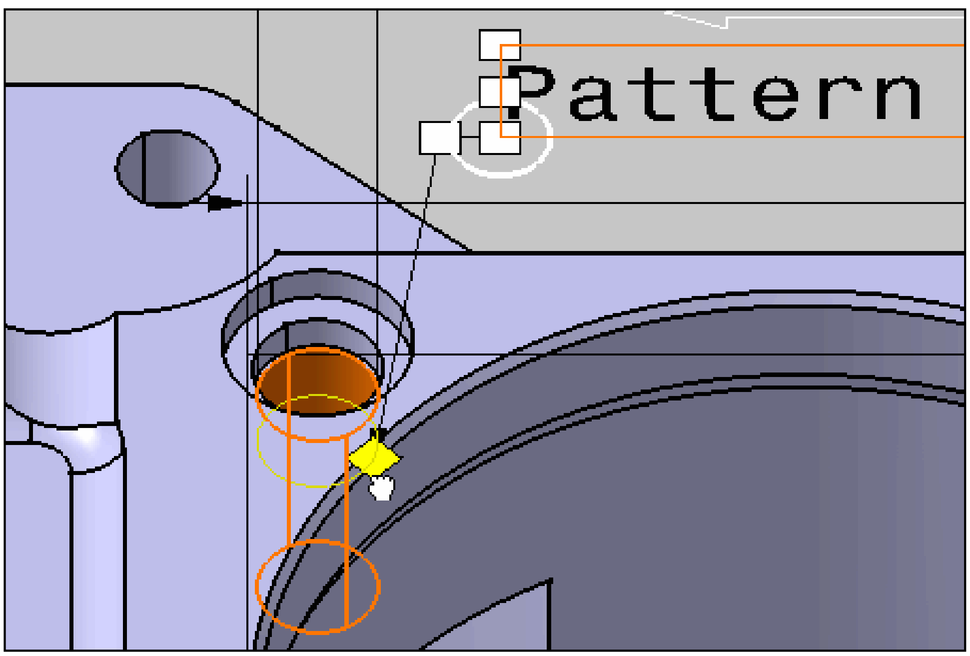


Any toleranced face hole of the pattern is then highlighted.

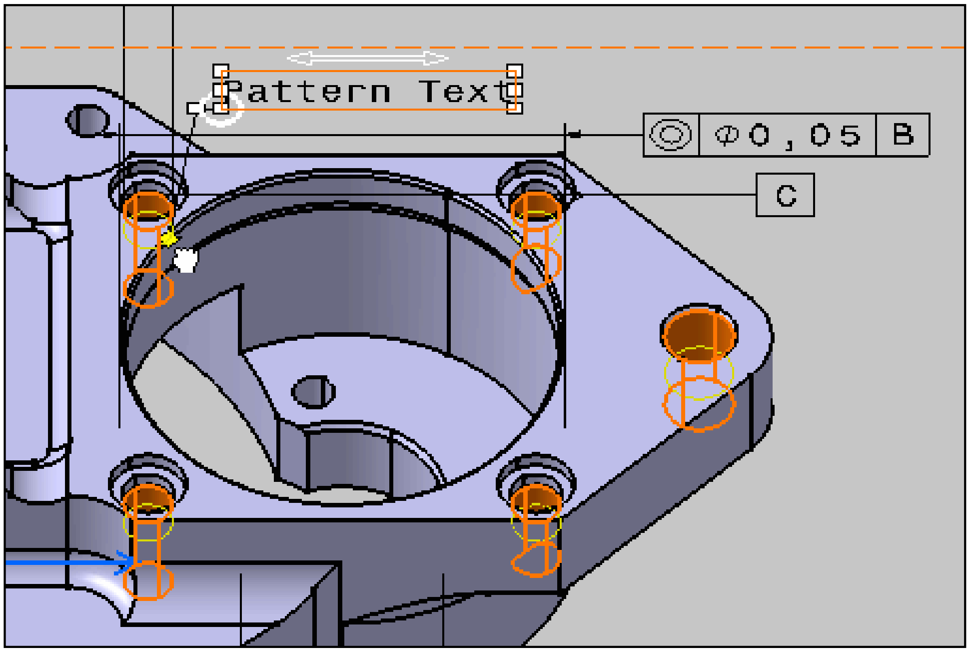


2. Select the yellow manipulator and start dragging the arrow head.

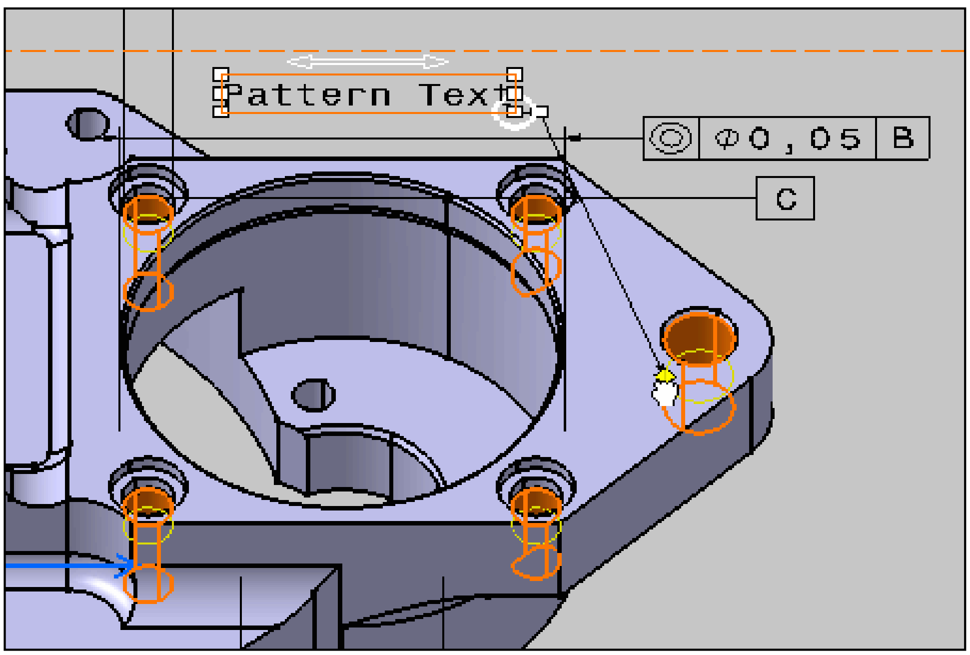
A yellow temporary trace is displayed. It corresponds to the intersection between the annotation plane and the corresponding toleranced elements. This trace defines all the possible positions for the arrow head that are semantically correct for the selected tolerancing annotation.



In our example, the toleranced elements are a group of five holes, this is why we obtain five yellow traces:

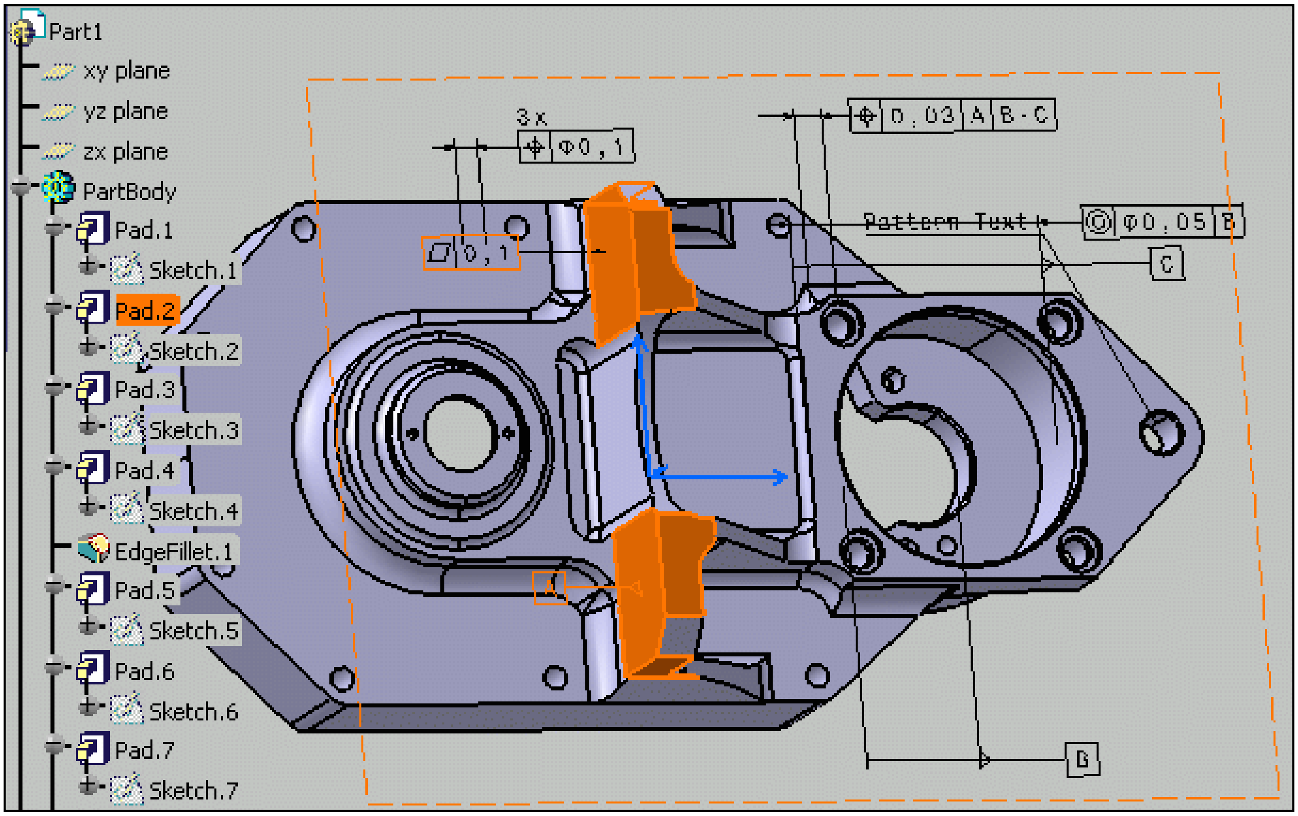


3. For example, drag and drop the arrow head on another trace to the right. You can obtain this:



4. Now, select Pad.2 in the specification tree. As you can see, the pad is highlighted in the geometry area, as well as all annotations which are applied to it.

This functionality is demonstrated here using a Part Design feature, but it is also available for Generative Shape Design features.



You can add the **3D Annotation Query Switch On/Switch Off** command to another workbench via **Tools -> Customize**. For more information, refer to the Customizing Toolbars and Workbenches chapter in the Infrastructure User's Guide.

Creating an Automatic Default Annotation

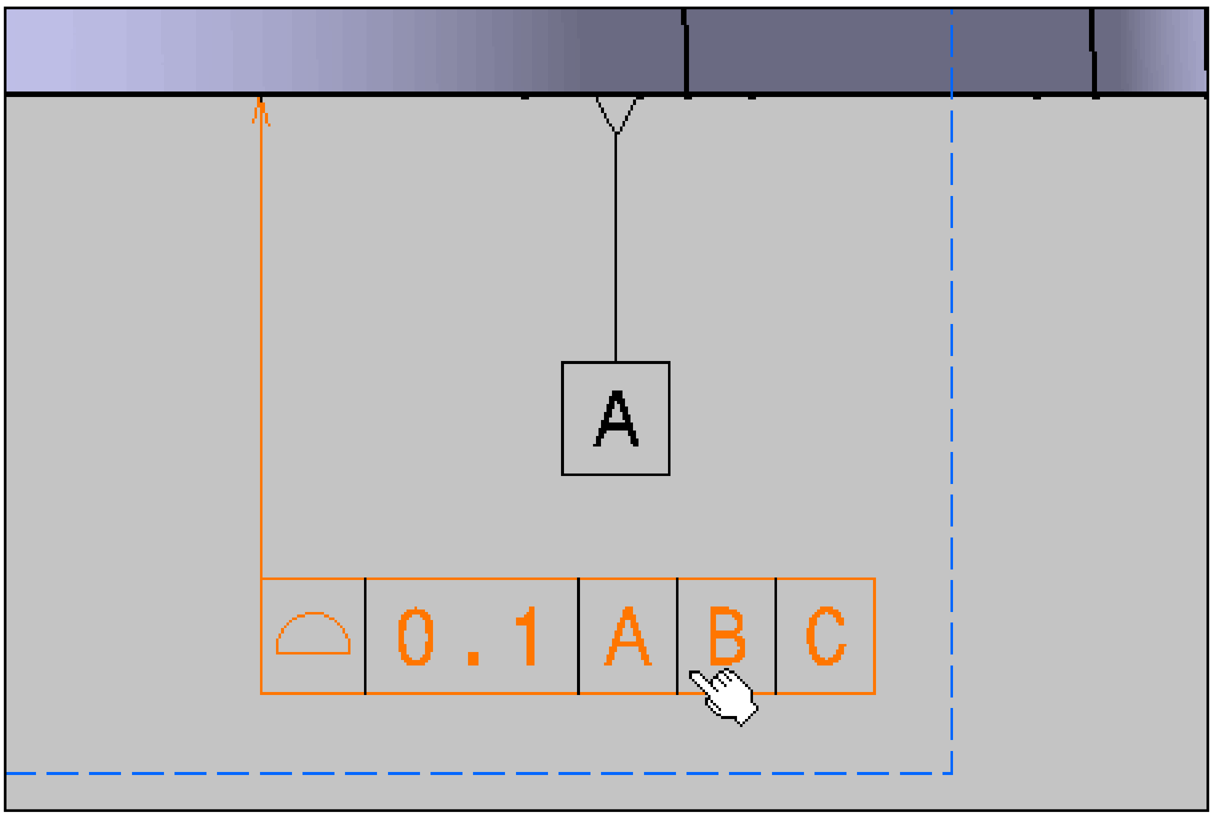
This task shows you how to apply an existing annotation to several geometrical elements of a part by making it the default annotation.

Open the Tolerancing_Annotations_03 CATPart document:

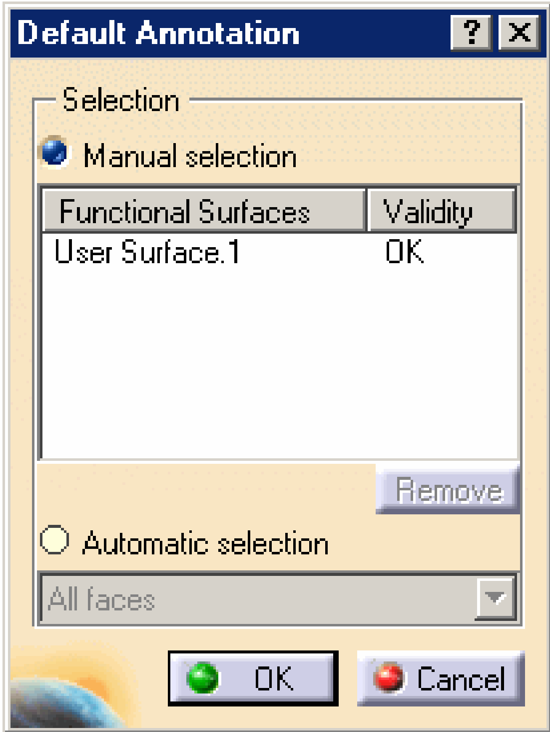
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

1. Select the **Insert -> Annotations -> Default Annotation** menu.

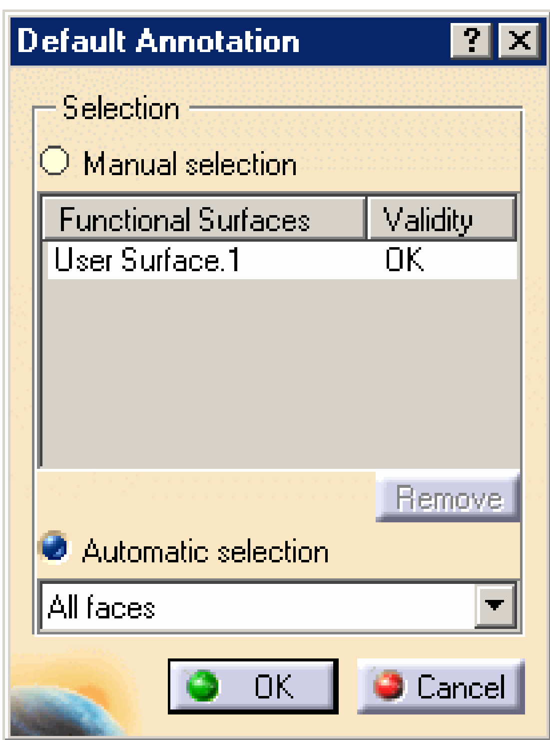
2. Select the **Position surfacic profile.1** annotation as shown



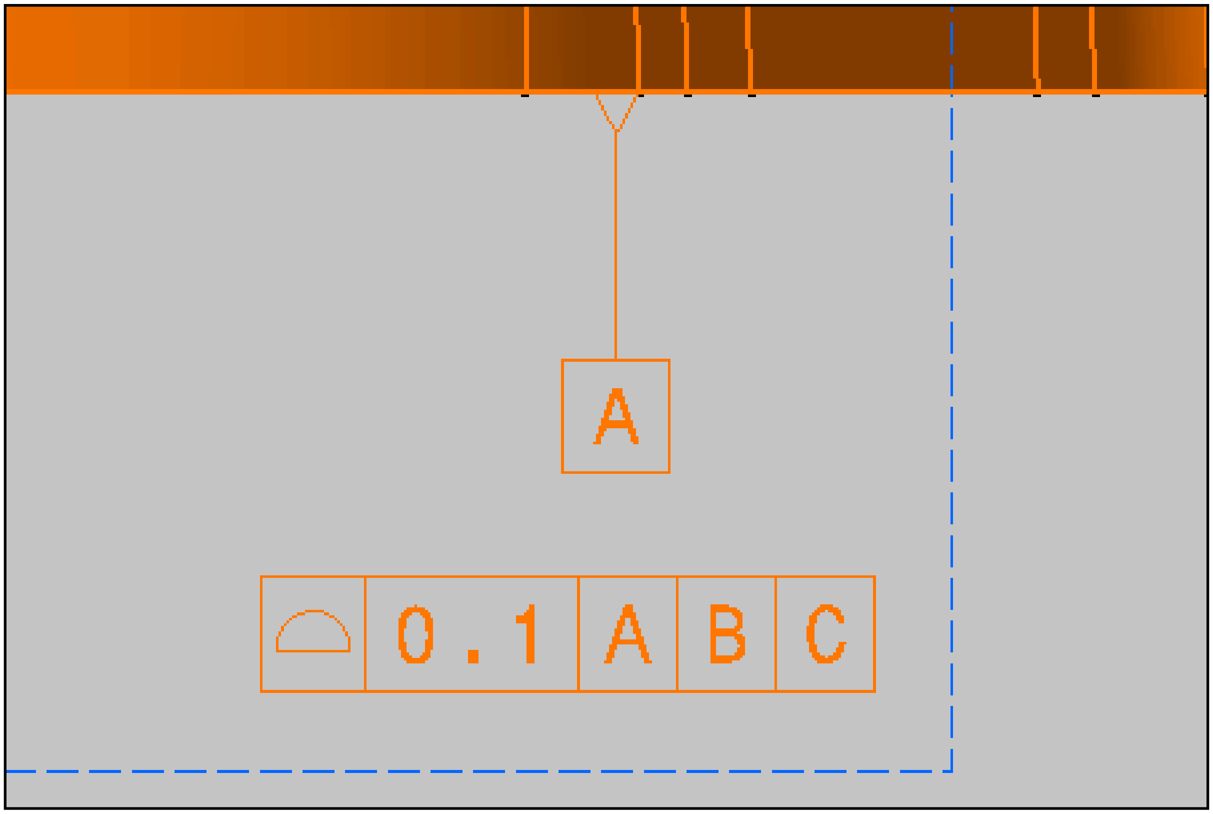
The **Default Annotation** dialog box appears.

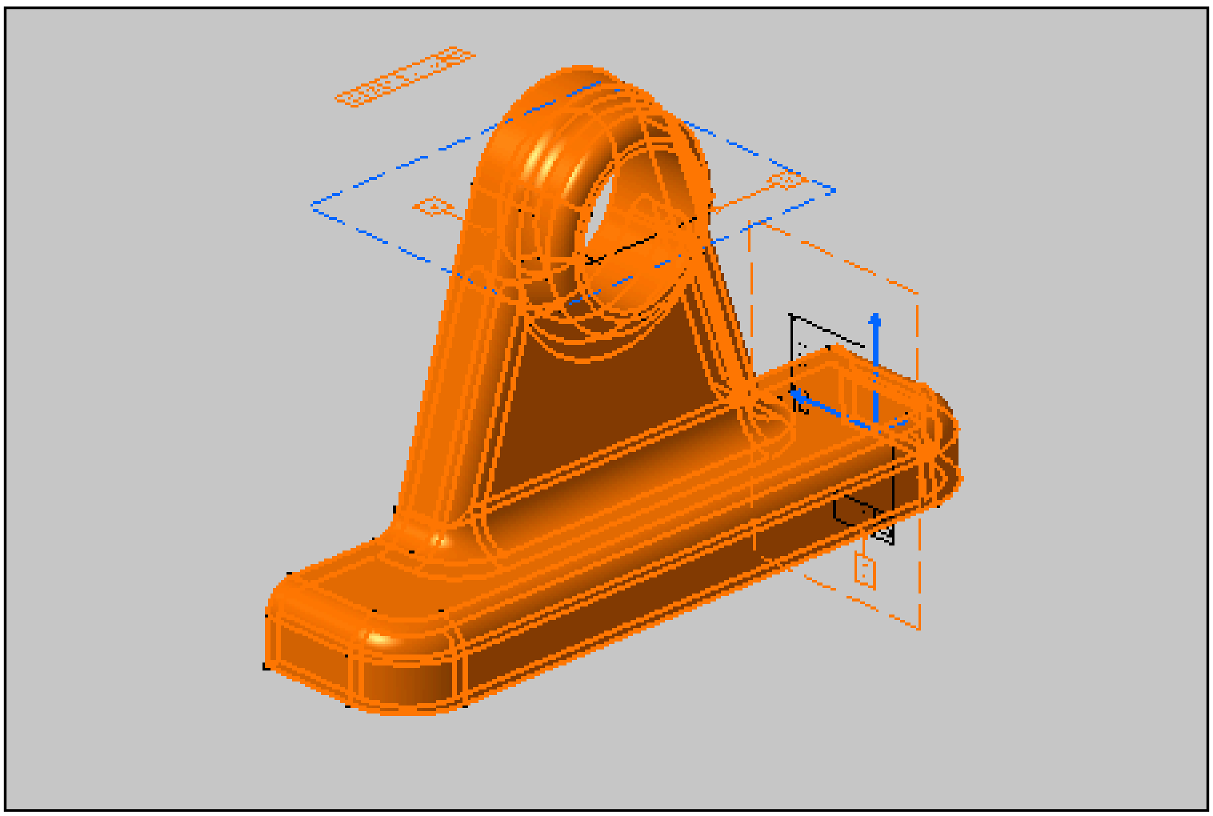


3. Select **Automatic** in the **Default Annotation** dialog box, **All faces** option in the combo box and click **OK**.



The annotation is now applied to all the faces of the part.





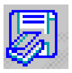
According to the selected annotation you will find the following option in the combo box.

Annotation and Tolerance types	Options
Roughness Non-semantic Geometrical Dimension and Tolerance Text Flagnote Note Object Attribute (NOA)	All faces Planar faces Cylindrical faces Spherical faces Non-canonical faces Fillet faces
Profile of a surface with Datum Reference Frame	All faces Planar faces Cylindrical faces Spherical faces Non-canonical faces Fillet faces
Profile of a surface without Datum Reference Frame	All faces Planar faces Cylindrical faces Spherical faces Non-canonical faces Fillet faces
Flatness	Planar faces
Generator straightness	Cylindrical faces
Cylindricity	Cylindrical faces
Circularity	Cylindrical faces
Radius size	Cylindrical faces Spherical faces Fillet faces
Diameter size	Cylindrical faces Spherical faces

Managing Power Copies

Create Power Copy: select the **Insert ->Advanced Replication Tools -> Powercopy Creation** command,
select the elements making up the Power Copy from the specification tree,
define a name for the Power Copy and its reference elements then choose an icon for identifying it.

Instantiate Power Copy: select the **Insert -> Instantiate From Document...** command,
select the document or catalog containing the Power Copy ,
complete the Inputs within the dialog box selecting adequate elements in the geometric area.



Save Power Copy into a Catalog: select the Power Copy from the specification tree,
select the **Insert -> Advanced Replication Tools -> Save In Catalog...** command,
enter the catalog name and click Open.

Creating Power Copies

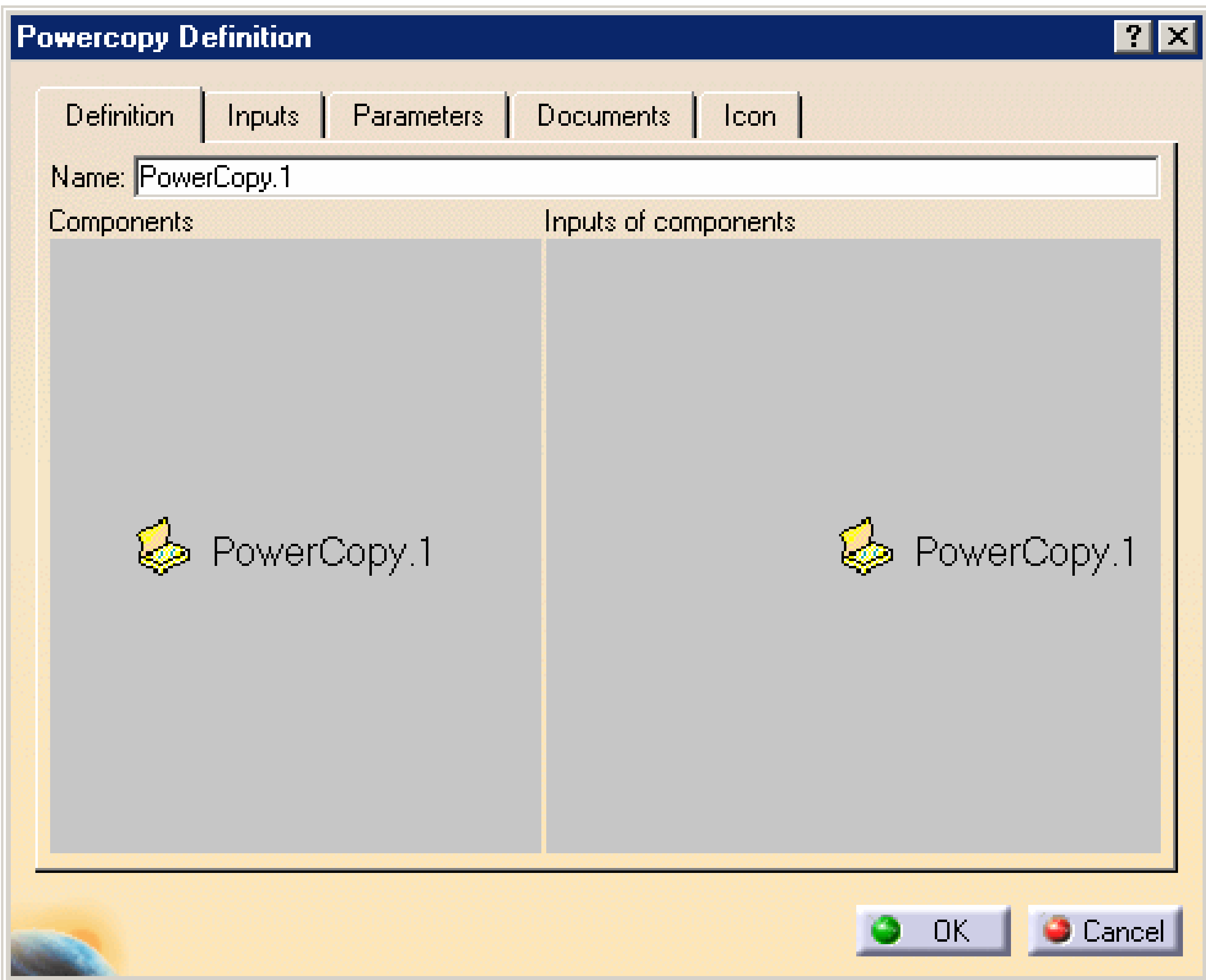
This task shows how to create Power Copy elements, to be reused later.

A Power Copy is a set of features (geometric elements, formulas, constraints, annotations and so forth) that are grouped in order to be used in a different context, and presenting the ability to be completely redefined when pasted.
This Power Copy captures the design intent and know-how of the designer thus enabling greater reusability and efficiency.

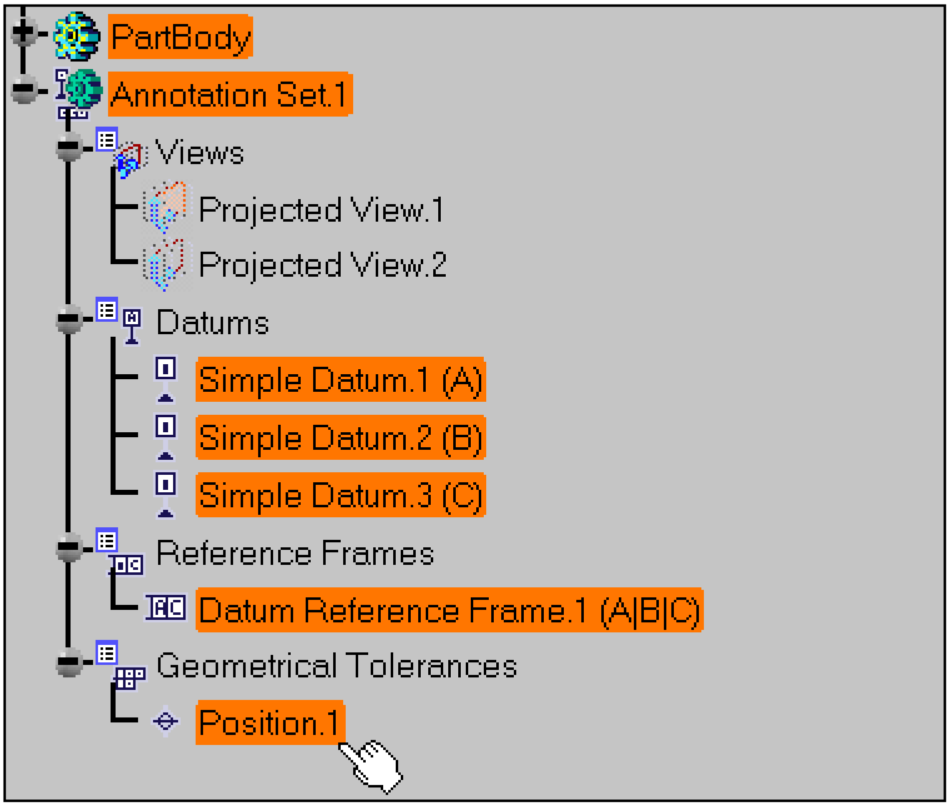
Open the PowerCopy1.CATPart document.

1. Select the **Insert ->Advanced Replication Tools -> Powercopy Creation** command.

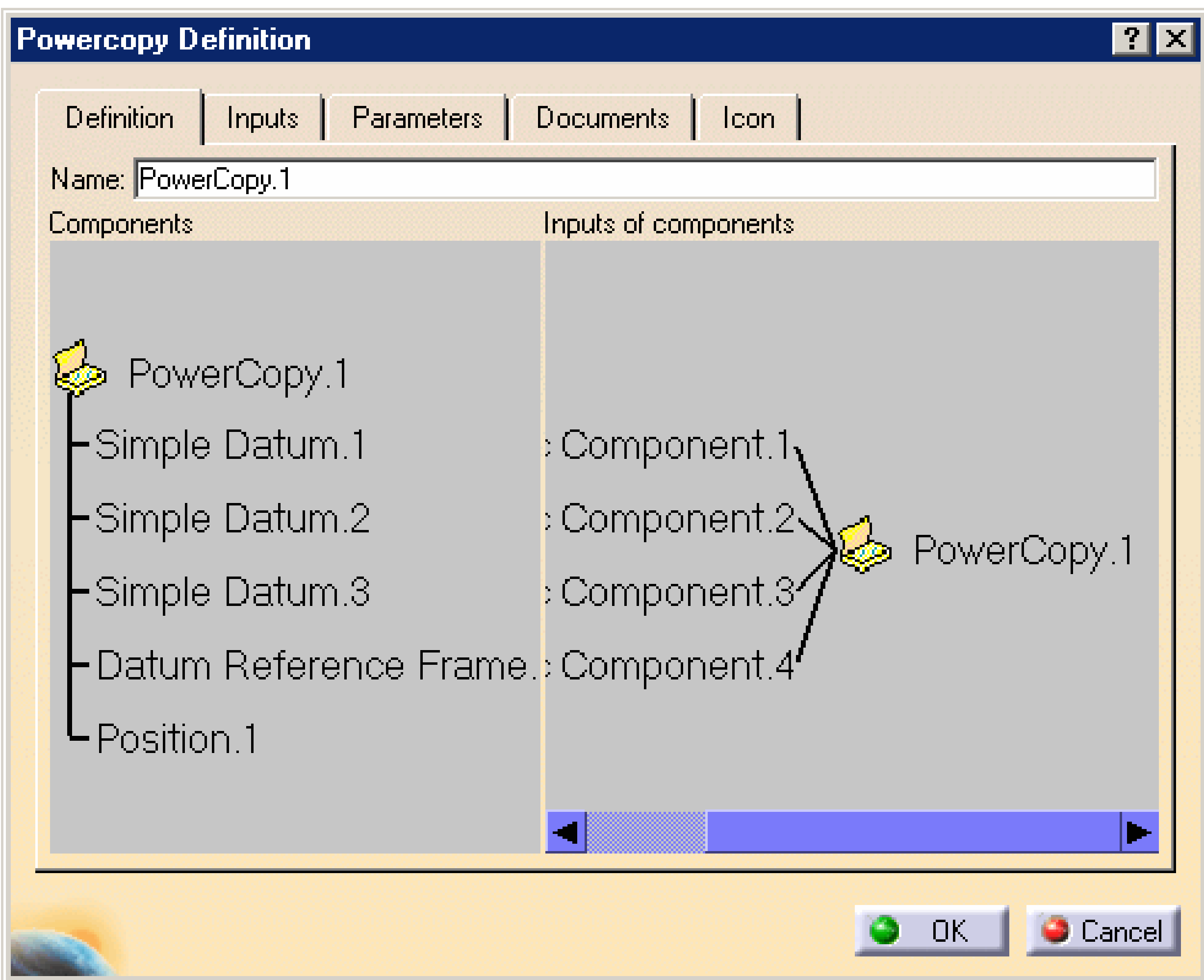
   The **Powercopy Definition** dialog box appears.



2. Select the following elements making up the PowerCopy from the specification tree.

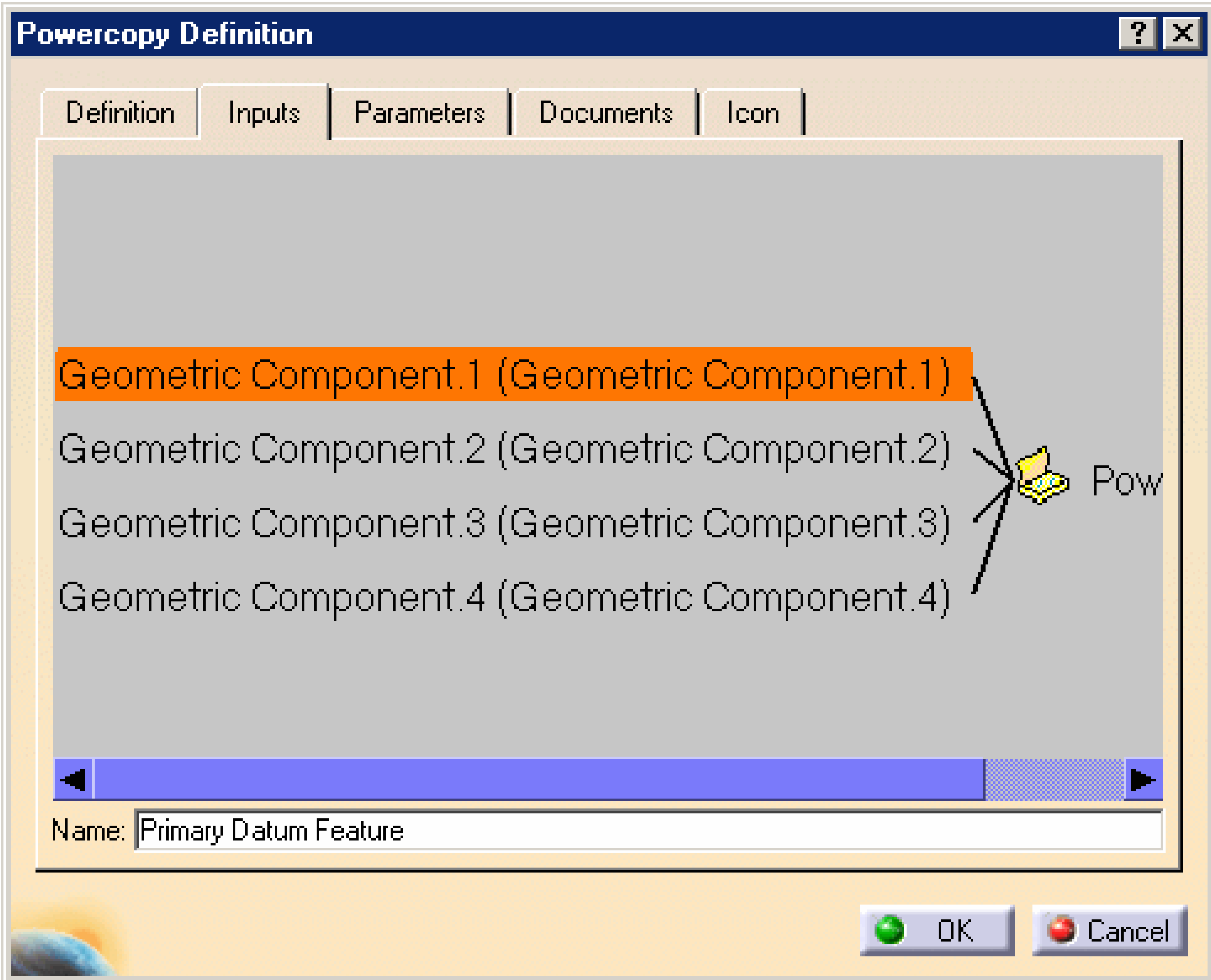


The **Powercopy Definition** dialog box is updated.

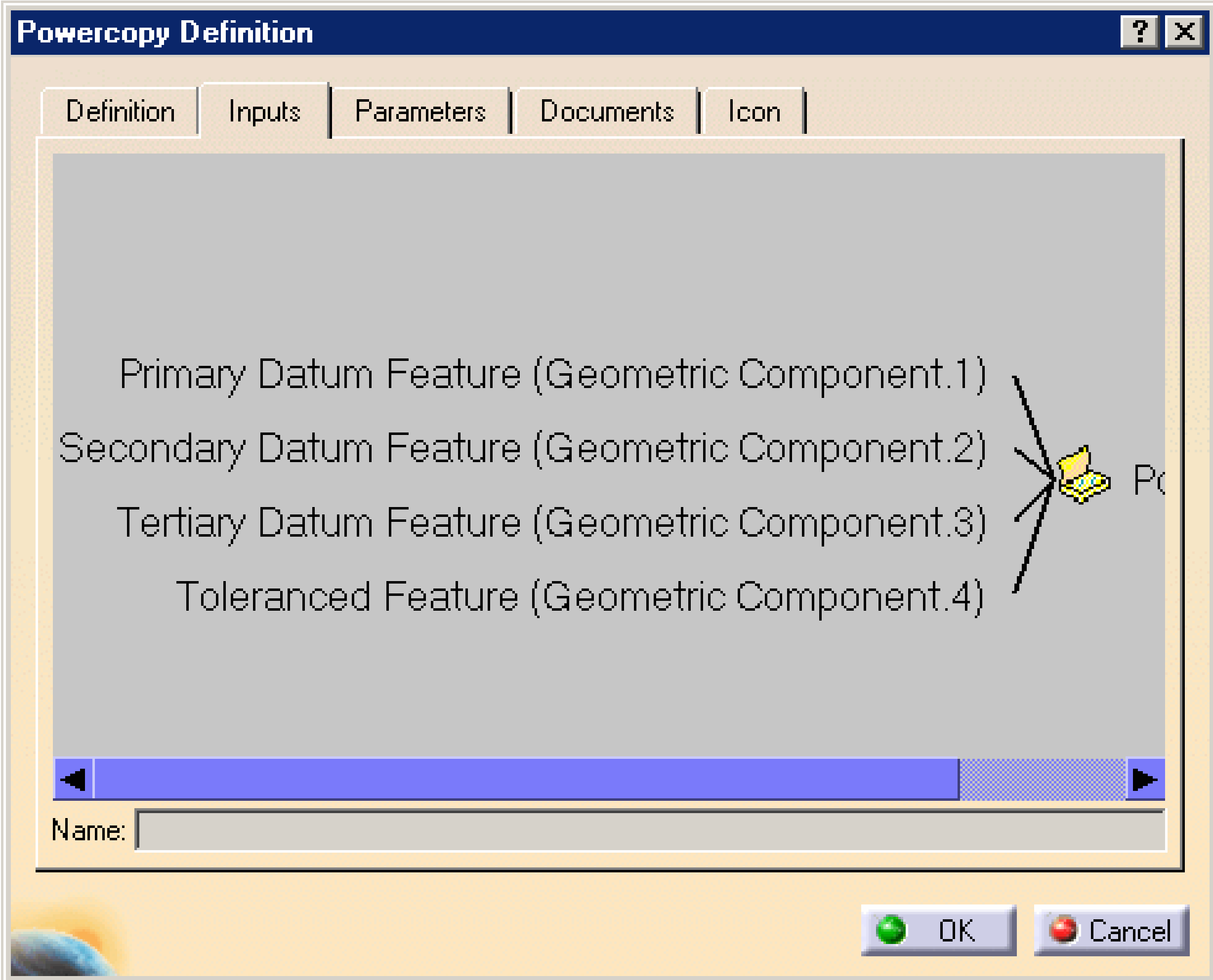


3. Select the **Inputs** tab.

4. Select the **Geometric Component.1** input and rename it into Primary Datum Feature.

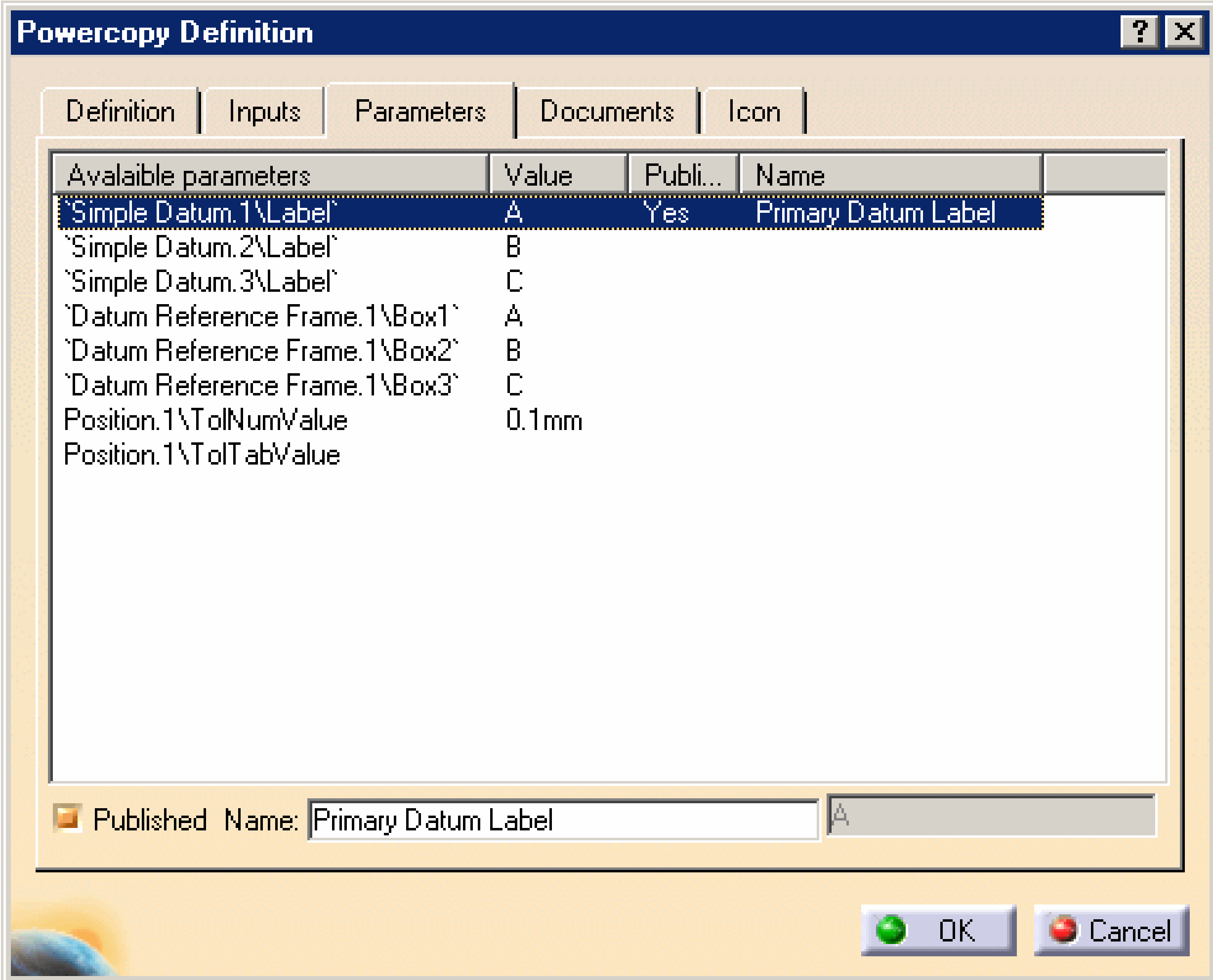


5. Repeat the operation with others inputs like this.

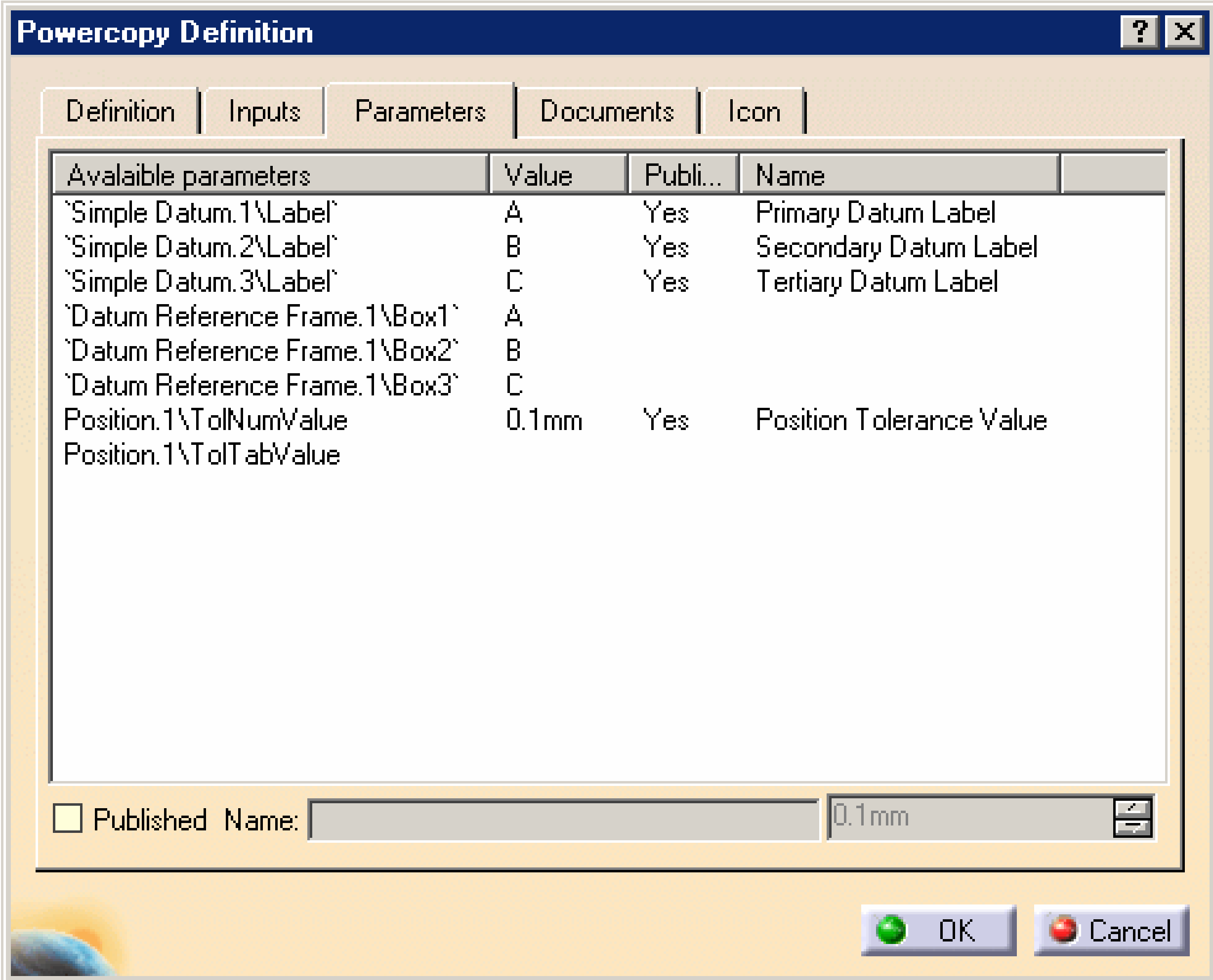


6. Select the **Parameters** tab.

7. Select the **Geometric Component.1** parameters and check **Publish Name** to rename it into Primary Datum Label.

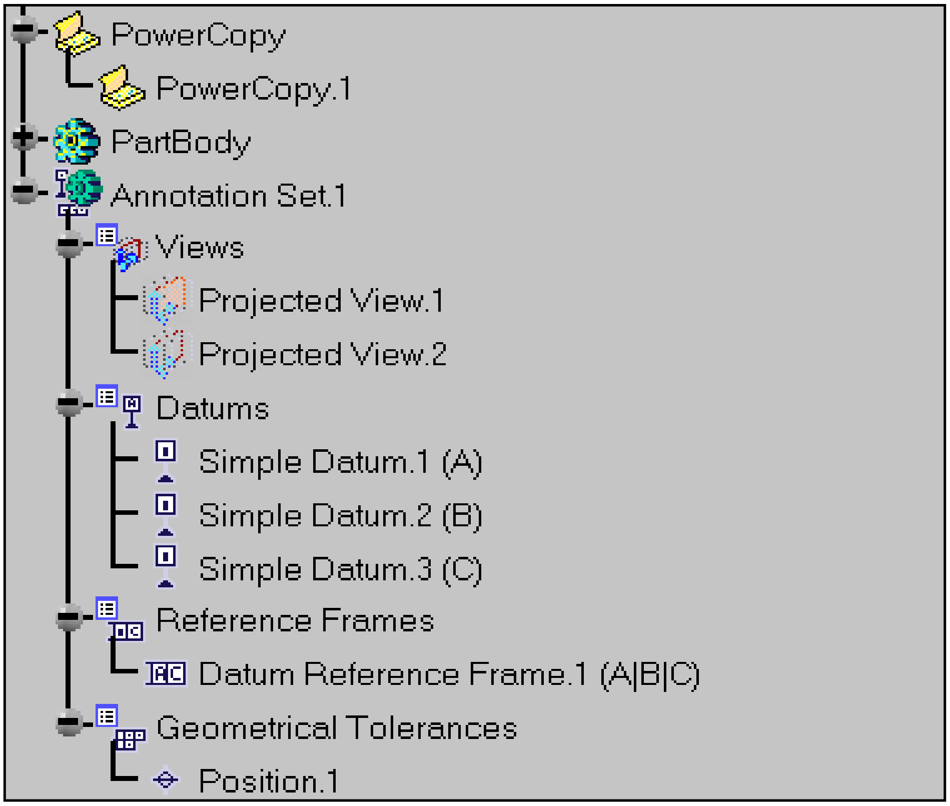


8. Repeat the operation with the following parameters like this.



9. Click **OK** in the **Powercopy Definition** dialog box.

The **Powercopy.1** is created.


PowerCopy
PowerCopy.1
PartBody
Annotation Set.1
Views
Projected View.1
Projected View.2
Datums
Simple Datum.1 (A)
Simple Datum.2 (B)
Simple Datum.3 (C)
Reference Frames
Datum Reference Frame.1 (A|B|C)
Geometrical Tolerances
Position.1

Instantiating Power Copies

This task shows how to instantiate Power Copies once they have been created.
See Creating Power Copy.

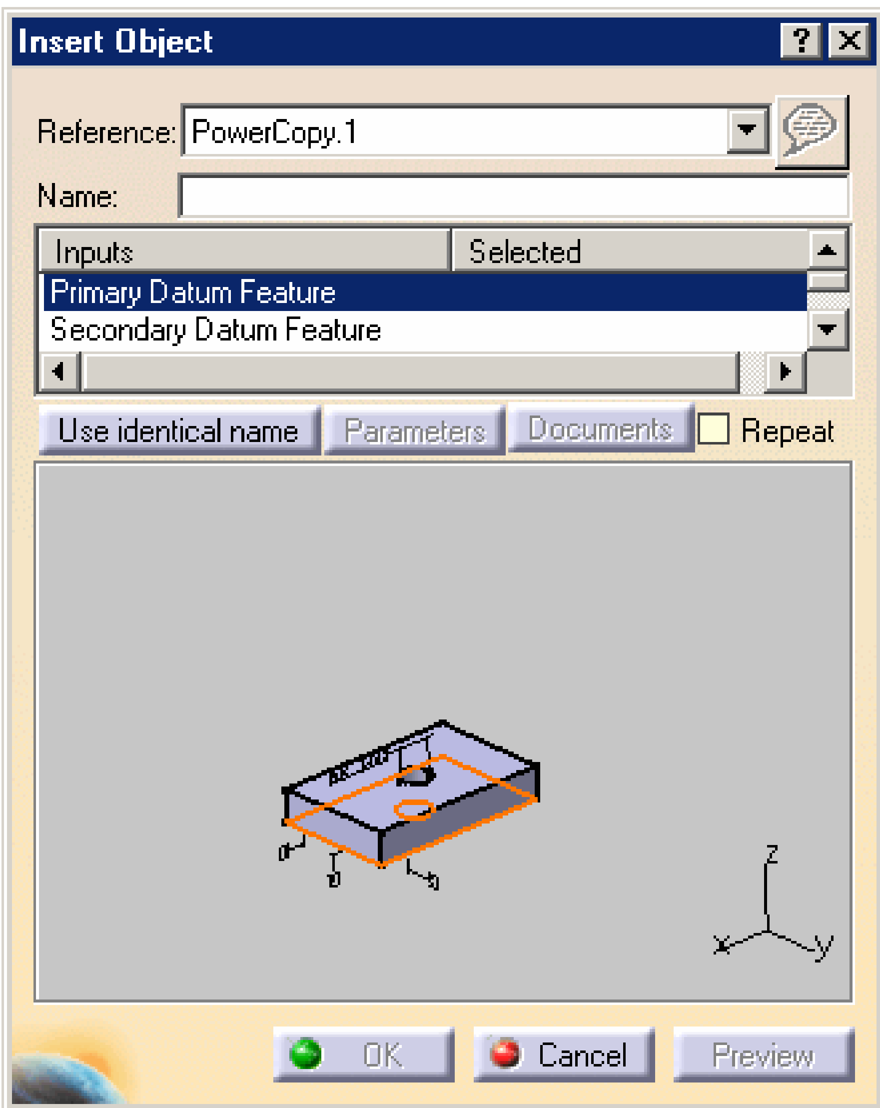
Open the Tolerancing_Annotations_01 CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

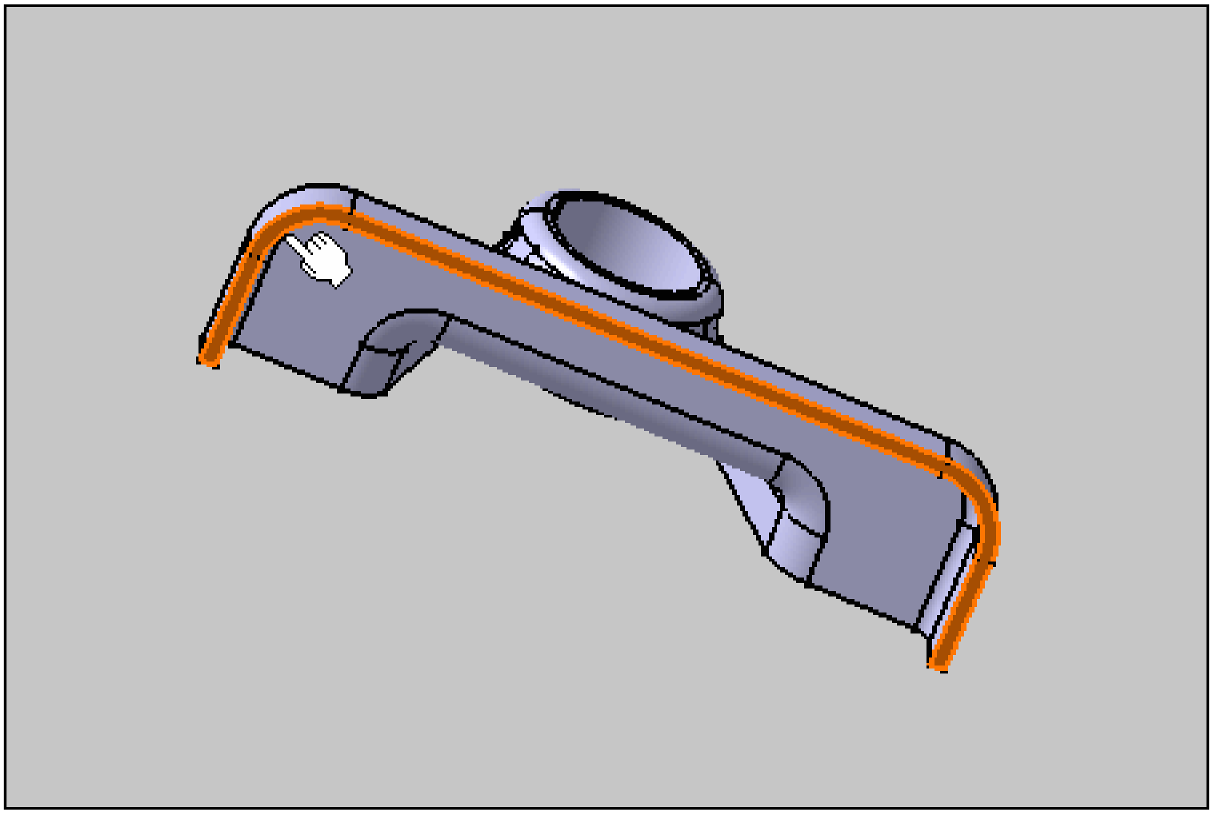
1. Select the **Insert -> Instantiate From Document...** command.

2. Select select PowerCopy2.CATPart from file selection dialog box.

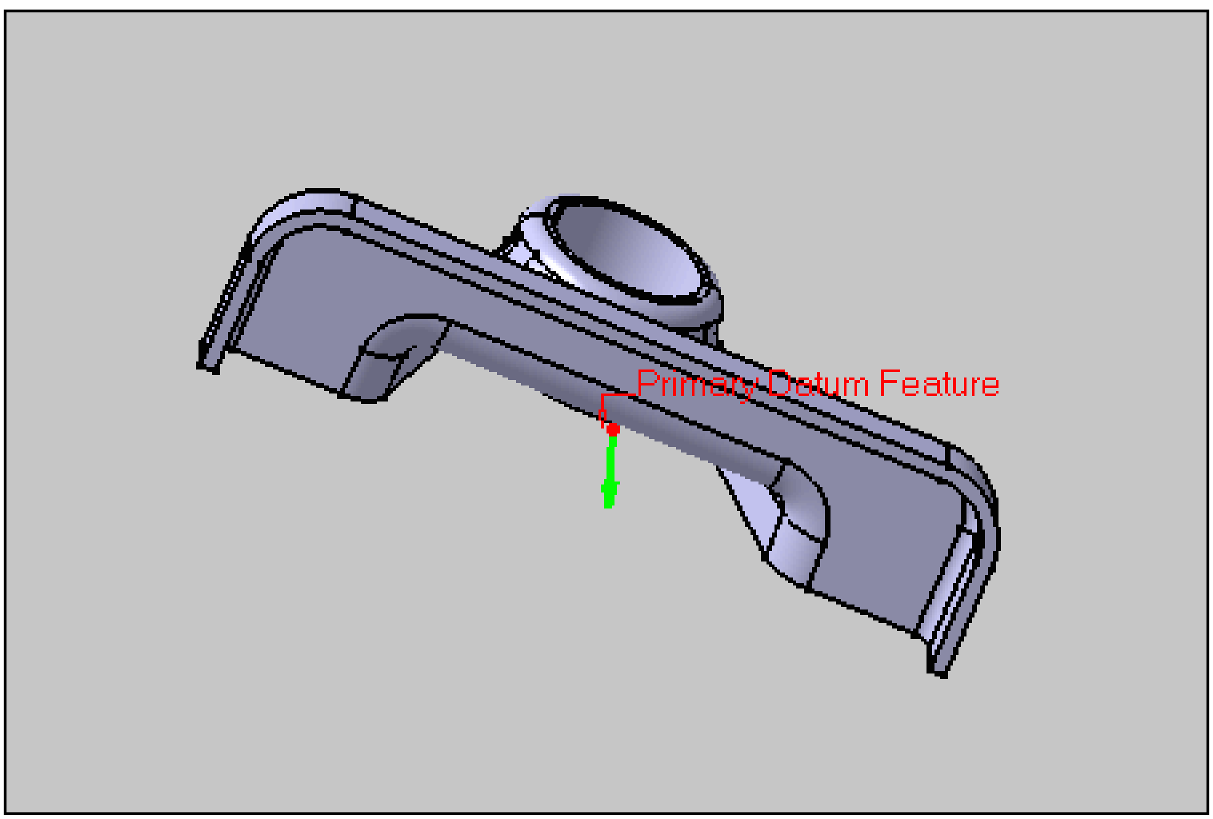
The **Insert Object** dialog box appears. **Primary Datum Feature** is pre-selected.



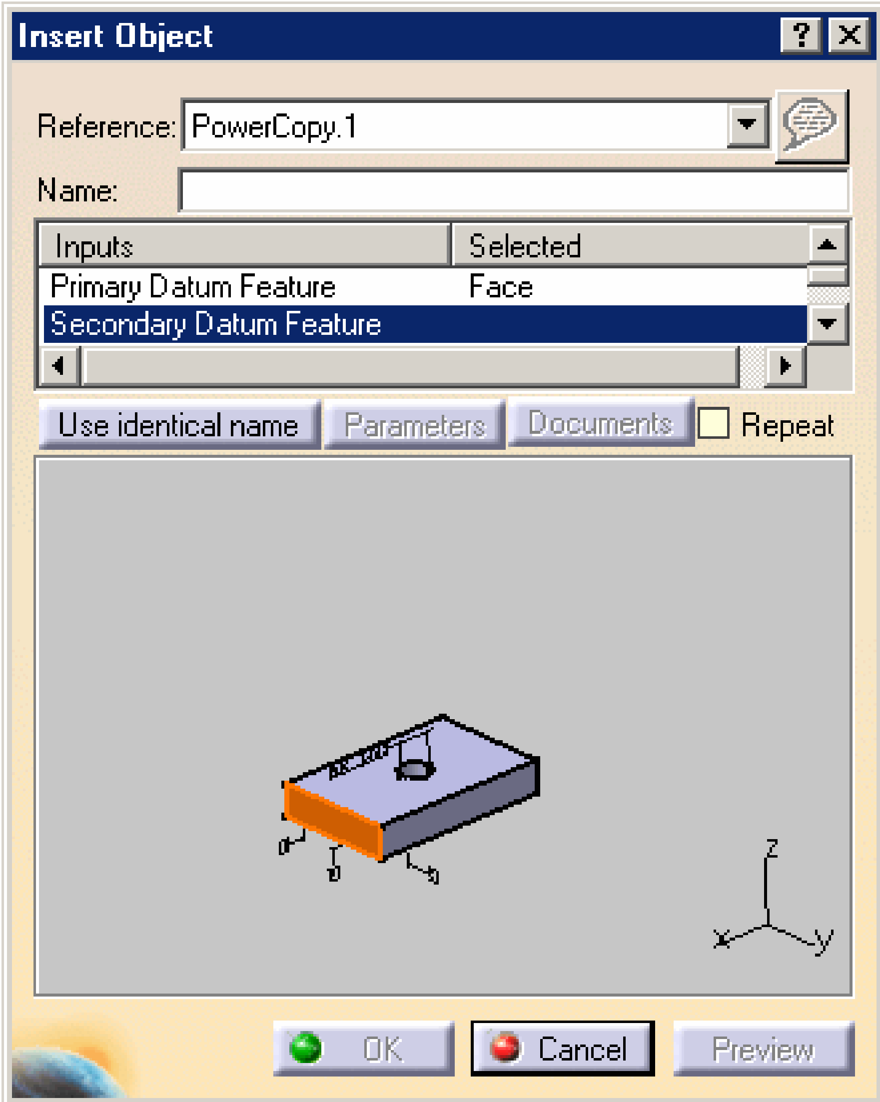
3. Select the surface as shown.



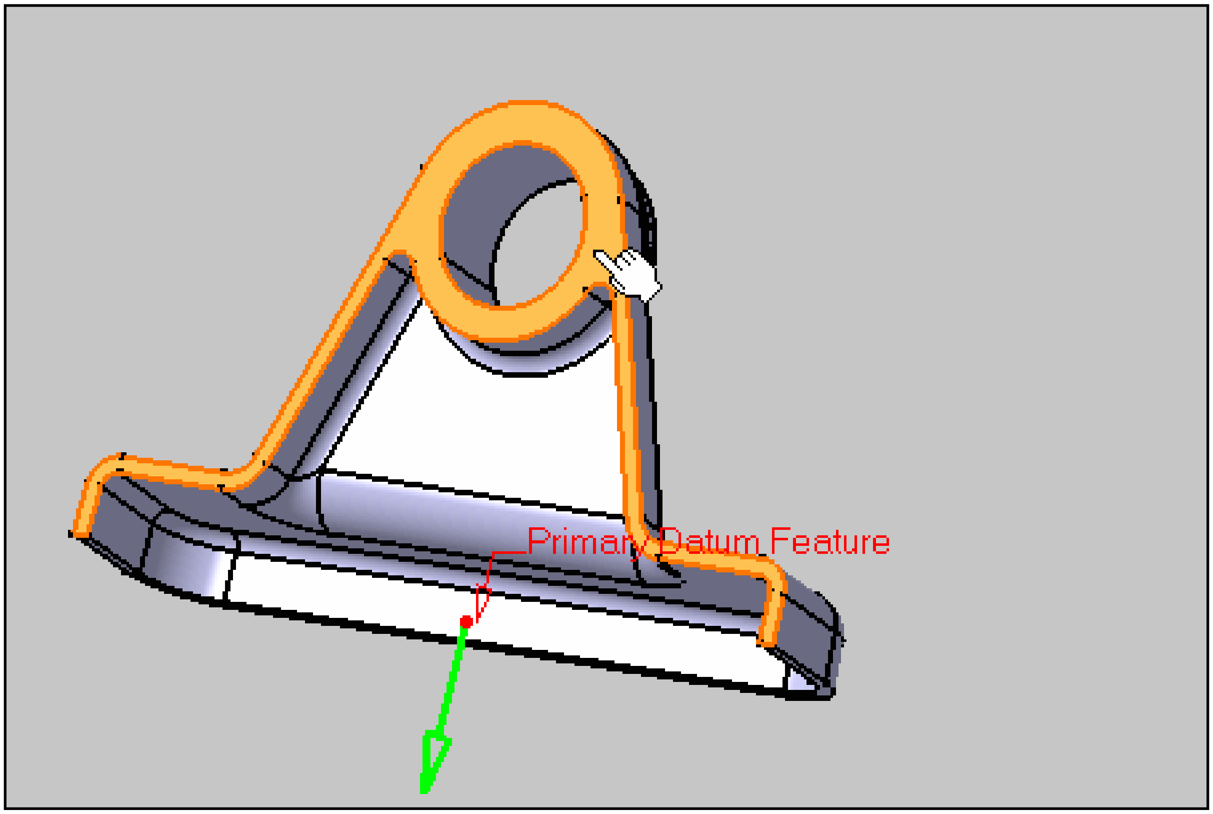
The **Primary Datum Feature** is associated with this surface.



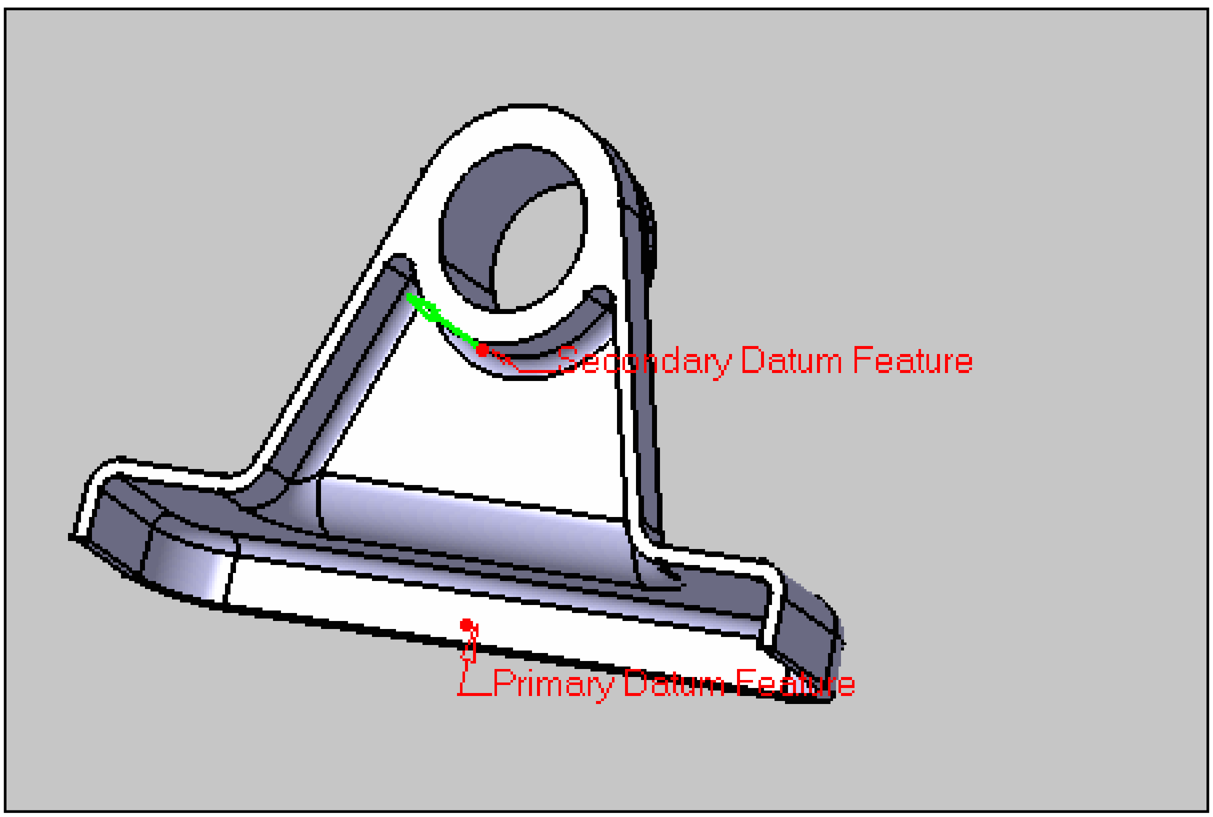
The next input **Secondary Datum Feature** is pre-selected in the **Insert Object** dialog box.



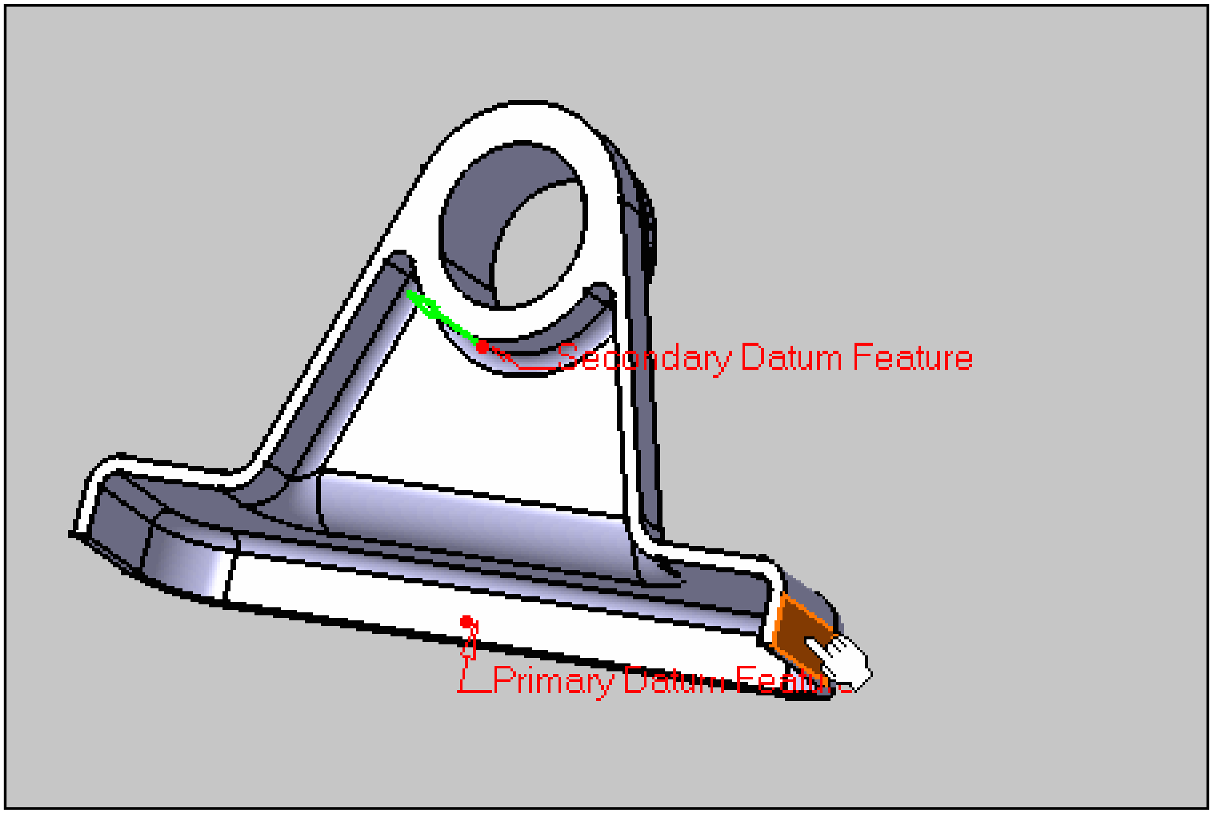
4. Select the surface as shown.



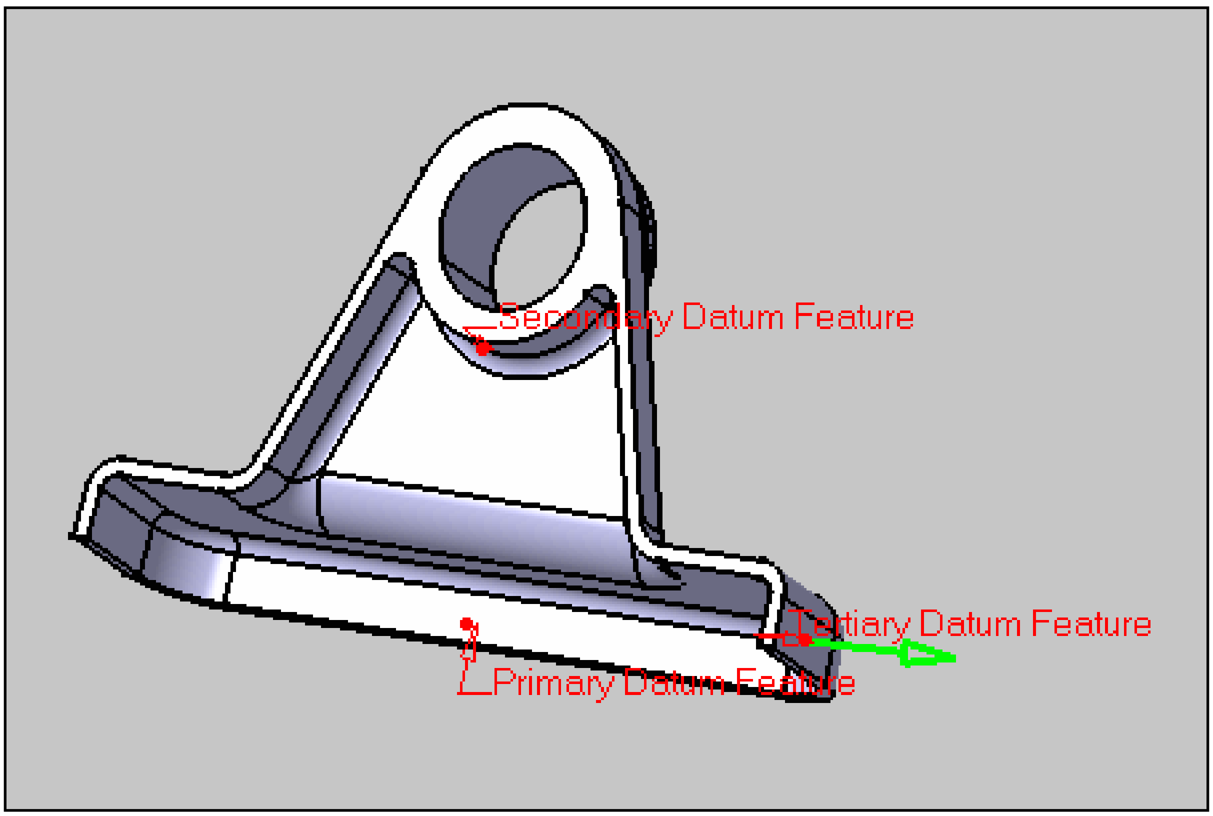
The **Secondary Datum Feature** is associated with this surface.
The next input **Tertiary Datum Feature** is pre-selected in the **Insert Object** dialog box.



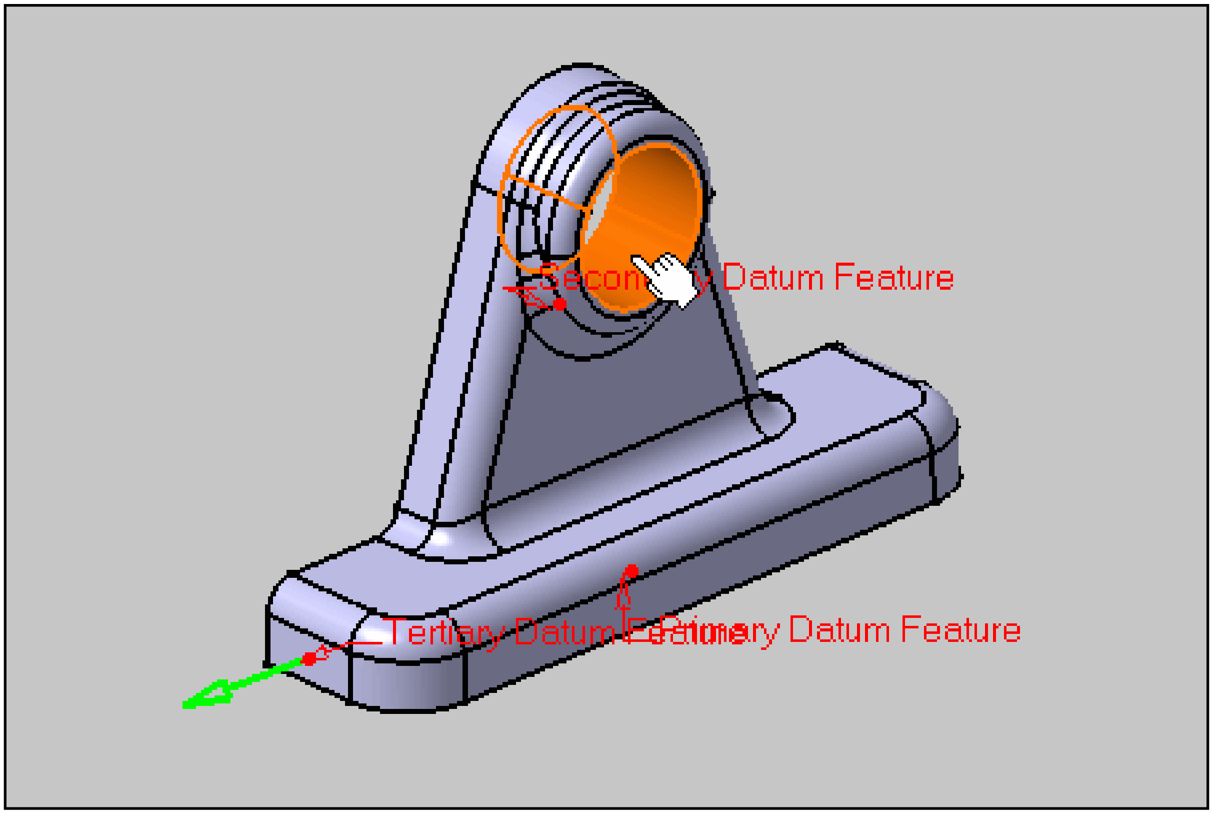
5. Select the surface as shown.



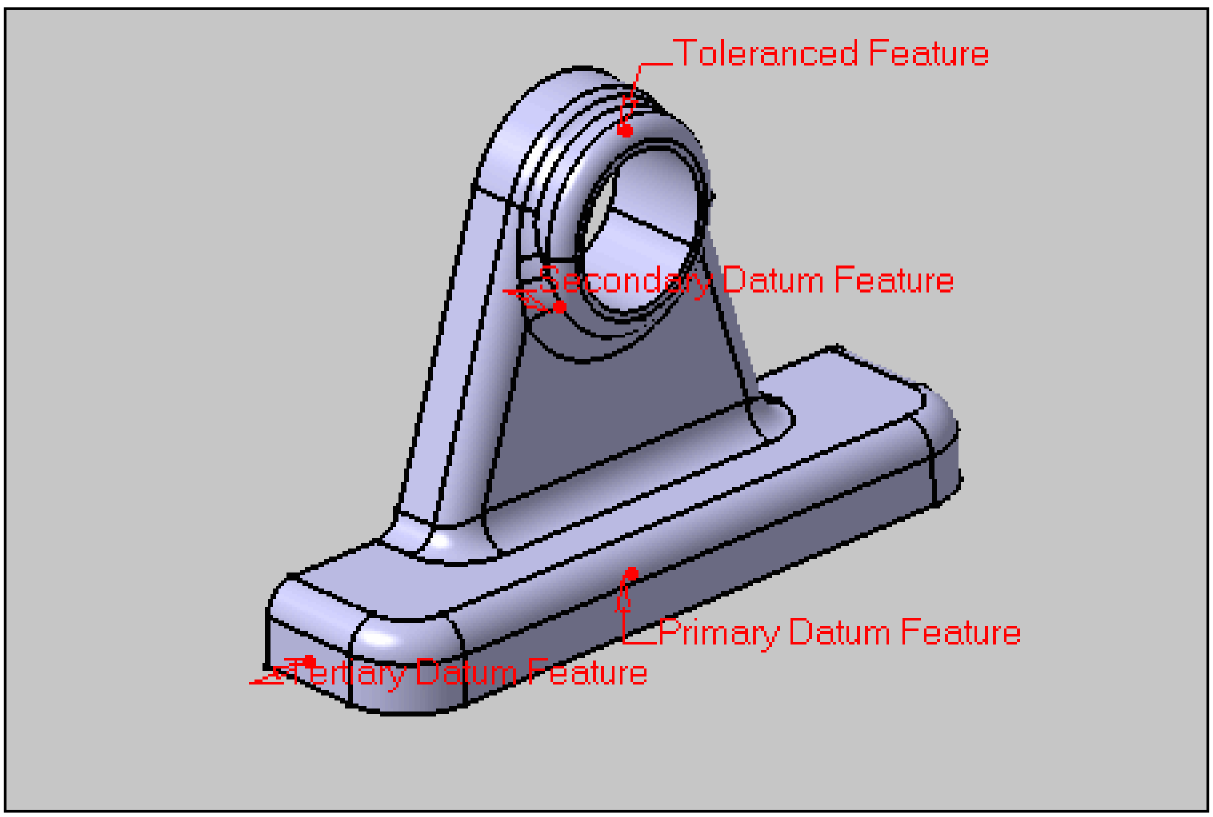
The **Tertiary Datum Feature** is associated with this surface.
The next input **Toleranced Feature** is pre-selected in the **Insert Object** dialog box.



6. Select the surface as shown.

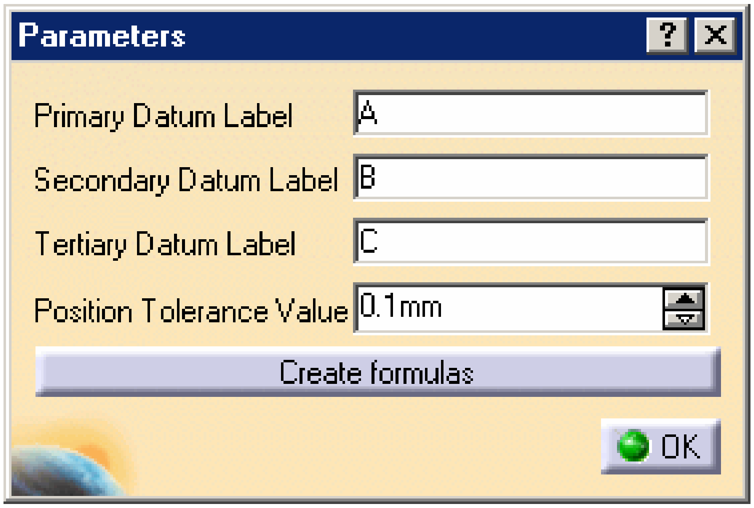


The **Toleranced Feature** is associated with this surface.

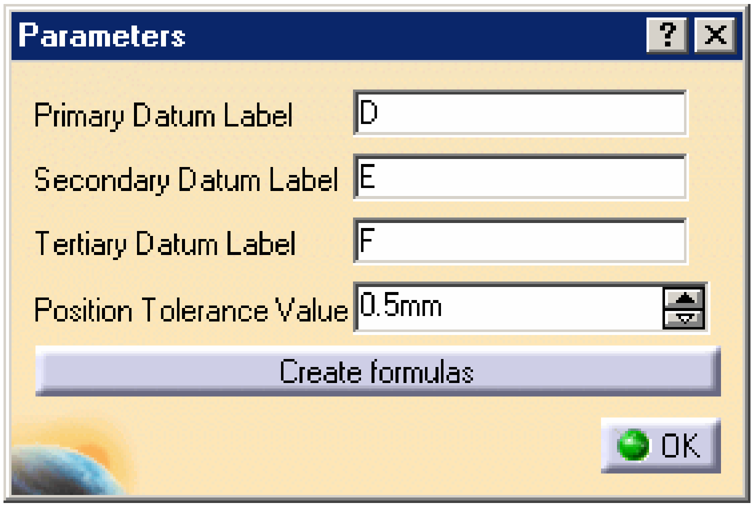


7. Click the **Parameters** in the **Insert Object** dialog box.

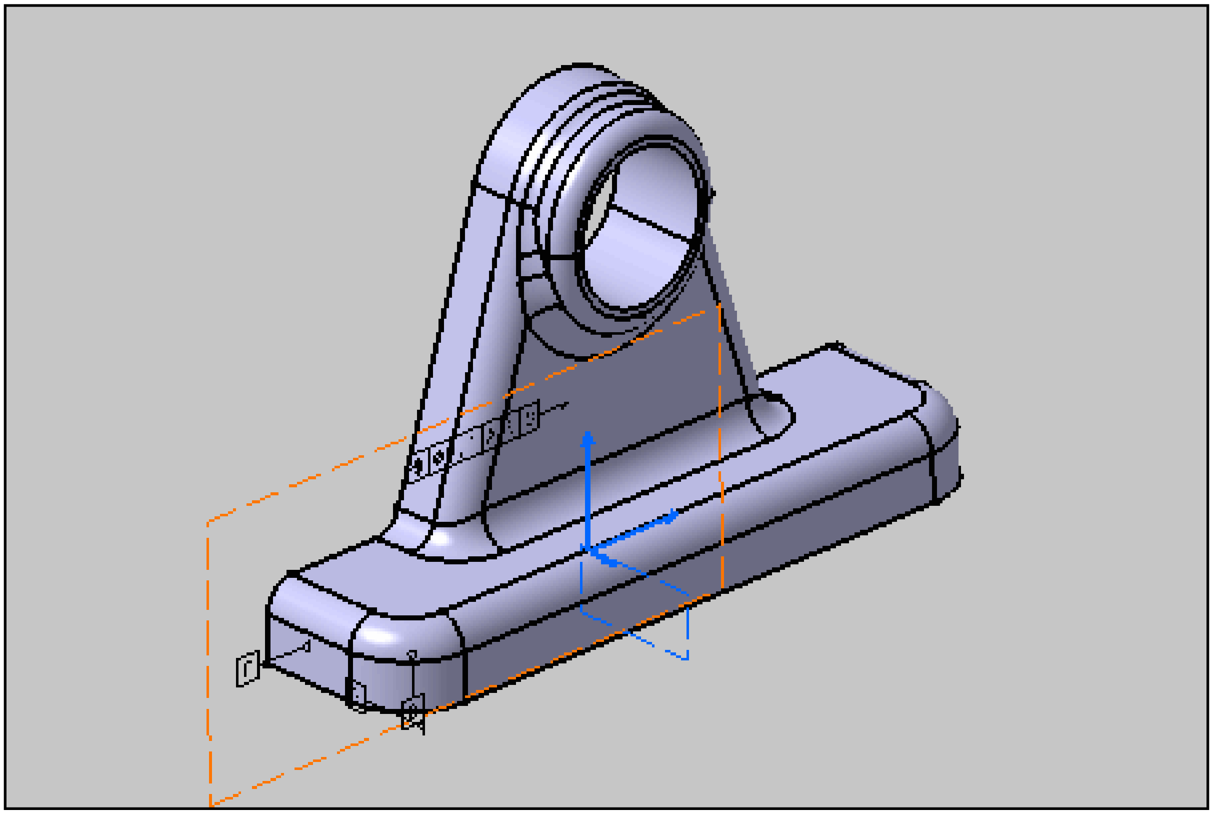
The **Parameters** dialog box appears.



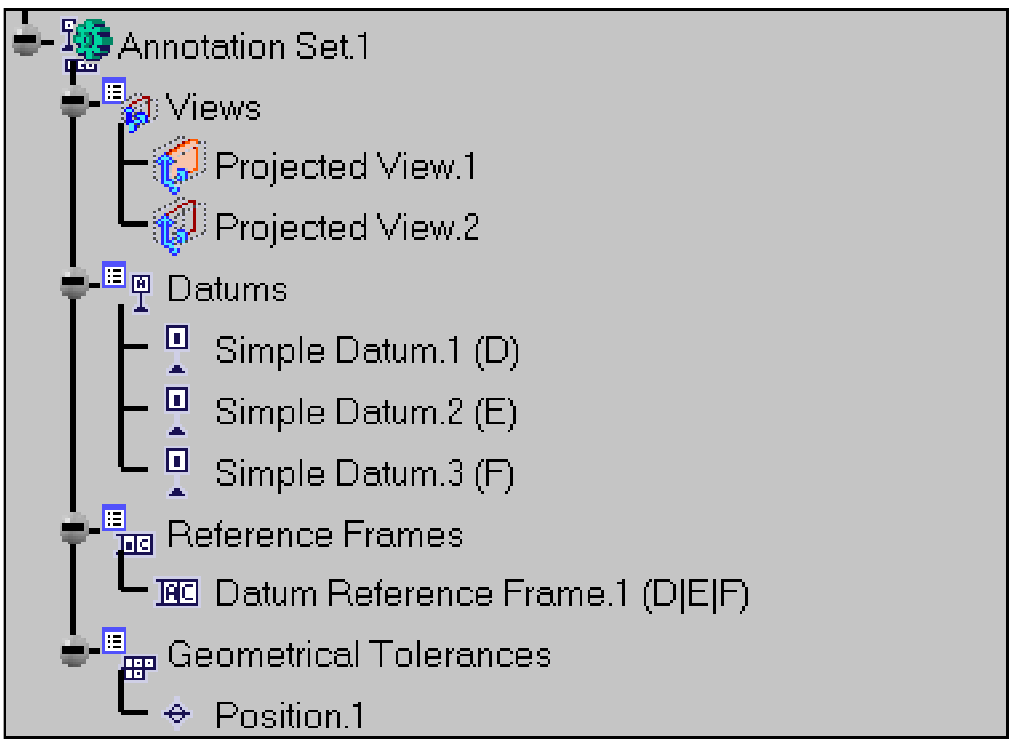
8. Change parameters as shown.



9. Click **OK** in the **Parameters** and **Insert Object** dialog boxes.



Annotations are created.

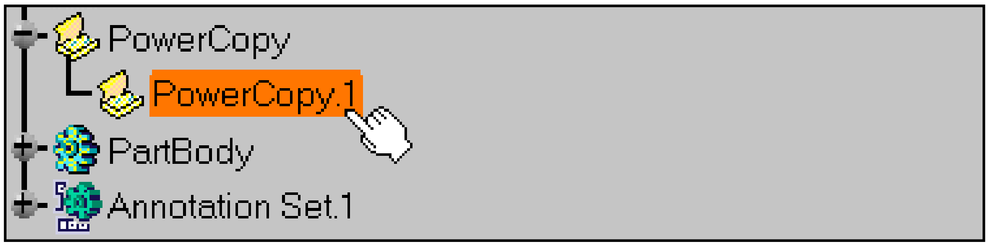


Saving Power Copies into a Catalog

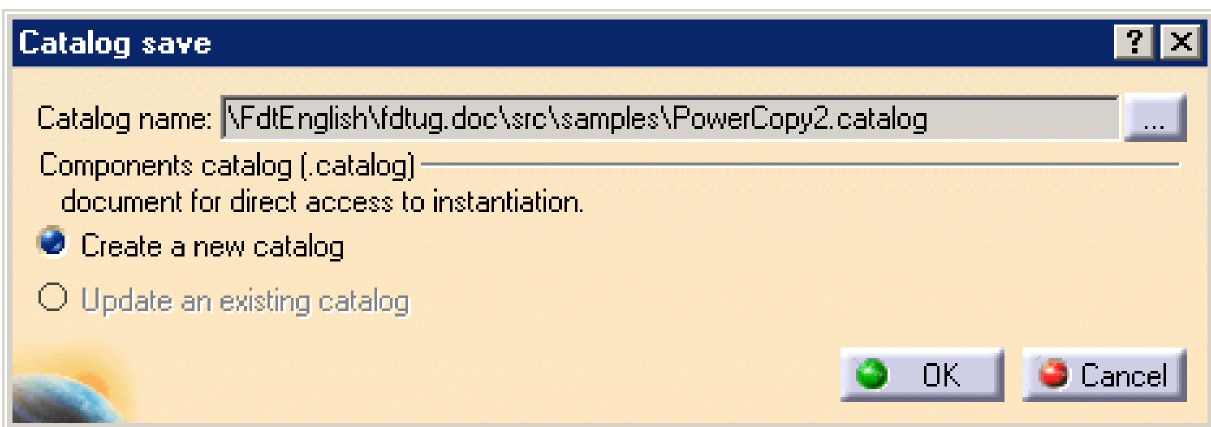
This task shows how to store Power Copy elements into a catalog.

Open the PowerCopy2.CATPart document.

1. Select the **Power Copy.1**.



2. Select the **Insert -> Advanced Replication Tools -> Save In Catalog...** command.



The **Catalog save** dialog box appears.

3. Click OK in the **Catalog save** dialog box.

When creating a catalog for the first time, click the ... button to display the **Open** dialog box, and navigate to the location where you wish to create a catalog.

Then simply key in the catalog name and click Open.

If you wish to add a PowerCopy to an existing catalog, simply activate the **Update an existing catalog** option in the **Catalog save** dialog box.

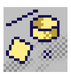
By default, the **Catalog save** dialog box recalls the catalog accessed last.

Providing Constructed Geometry for 3D Annotations

These functionalities provide the capability to query and manage the constructed geometry that is associated to a tolerancing feature (User surface and Group of surfaces features).

Constructed geometry (such as center point, axis, median plane, gage plane, etc) is very often used in order to define the theoretical dimensions of parts or products (framed dimensioning).
These constructed elements do represent the tolerancing feature (User surface and Group of surfaces features) and are used to define the tolerance zone position of geometrical tolerances, the related position of the datums of a datum reference frame, the size and position of a partial surface or a datum target.
The capability that is described here allows either managing constructed geometry that has been manually created by the user.

The existing geometry is the represented geometry, the constructed geometry is the representing geometry.



Create an Automatic Constructed Geometry: click this icon, select the context.

Manage Constructed Geometry: click this icon, select the represented geometry then the representing geometry.

Creating an Automatic Constructed Geometry

This task shows you how to create an automatic constructed geometry.

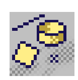
- Creating an automatic constructed geometry allows you to create associative wireframe geometry according to the context or represented geometry.
- The lifecycle of an automatic constructed geometry is managed by the application.
- To customize the constructed geometry options, see Constructed Geometry.

The following constructed geometry elements are available:

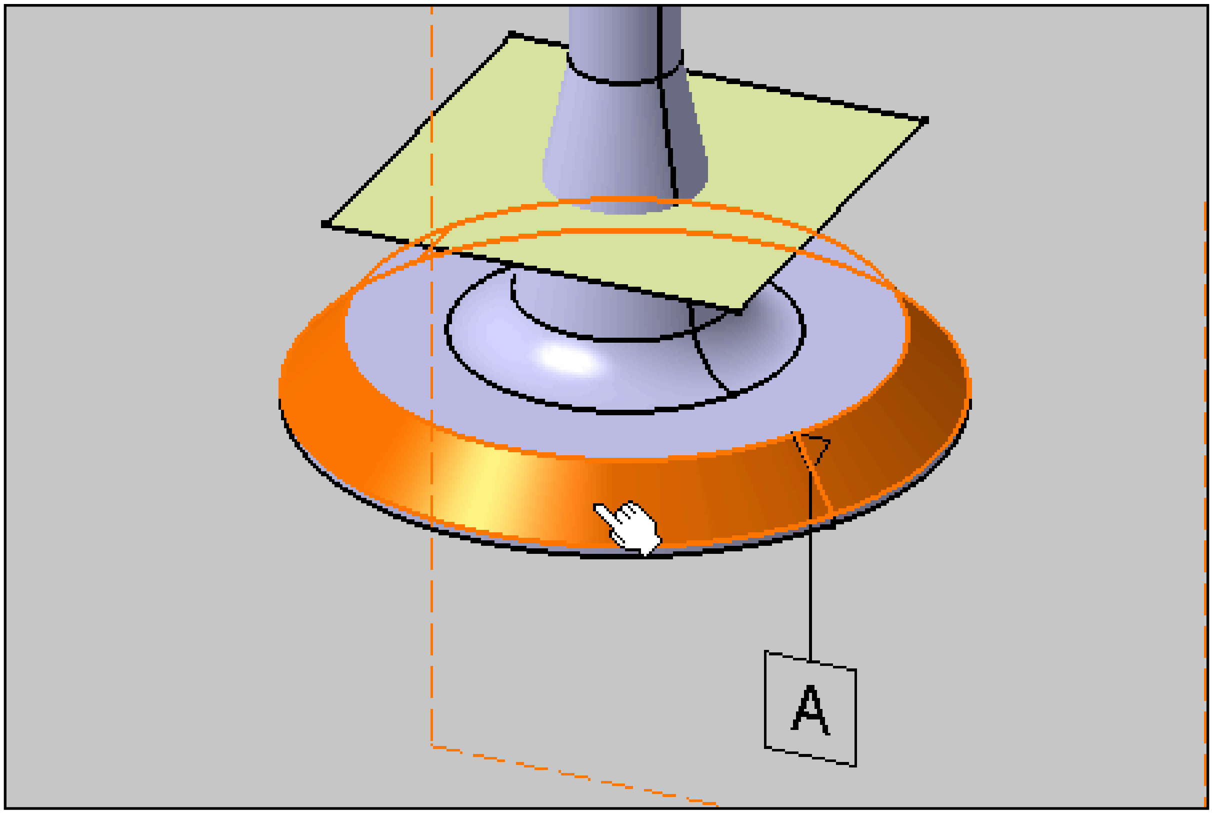
- Point
- Axis
  - Axis cylinder
  - Common axis of coaxial cylinders
- Plane
- Cylinder
  - Circling cylinder: defined passing through three cylinders at least.
  - Reference cylinder: defined by its axis and passing through one or two cylinders.
- Circle
- Thread

Open the Tolerancing_Annotations_08.CATPart document:

- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

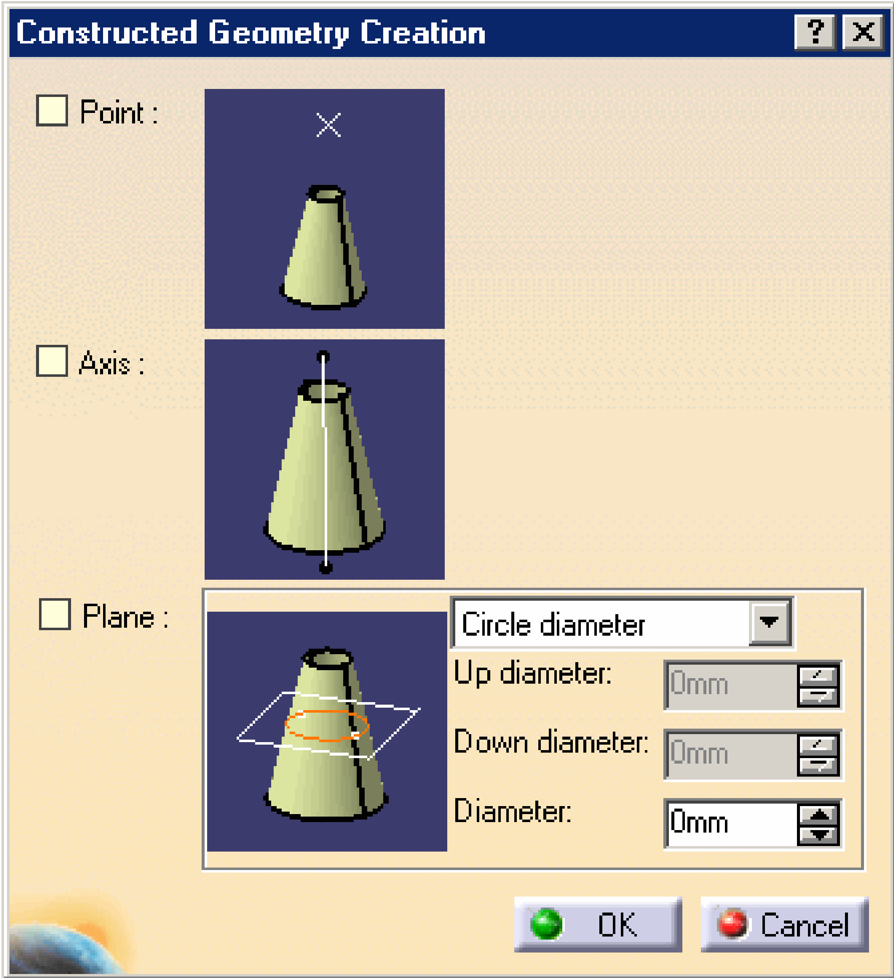
1. Click the **Constructed Geometry Creation** icon: 

2. Select the surface as shown.



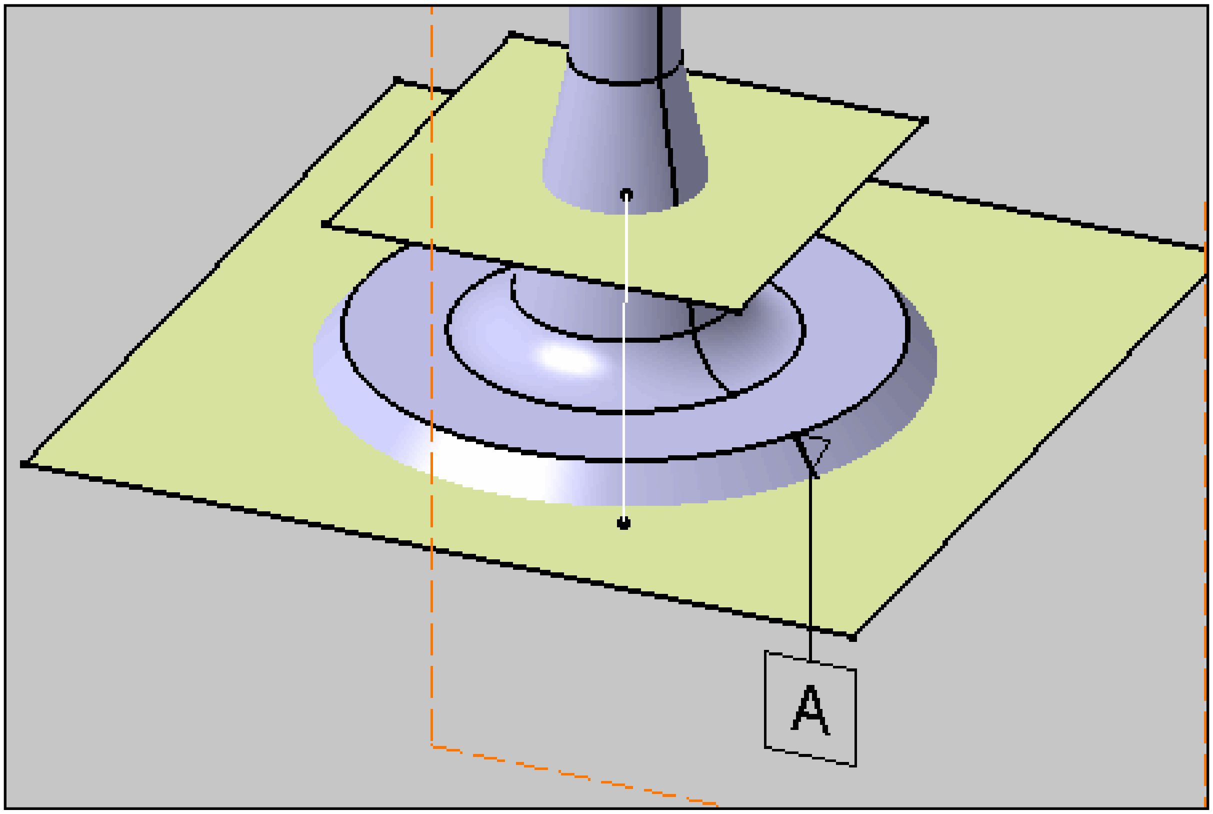
The **Constructed Geometry Creation** dialog box appears.

Double-clicking any automatically constructed geometry displays this dialog box.

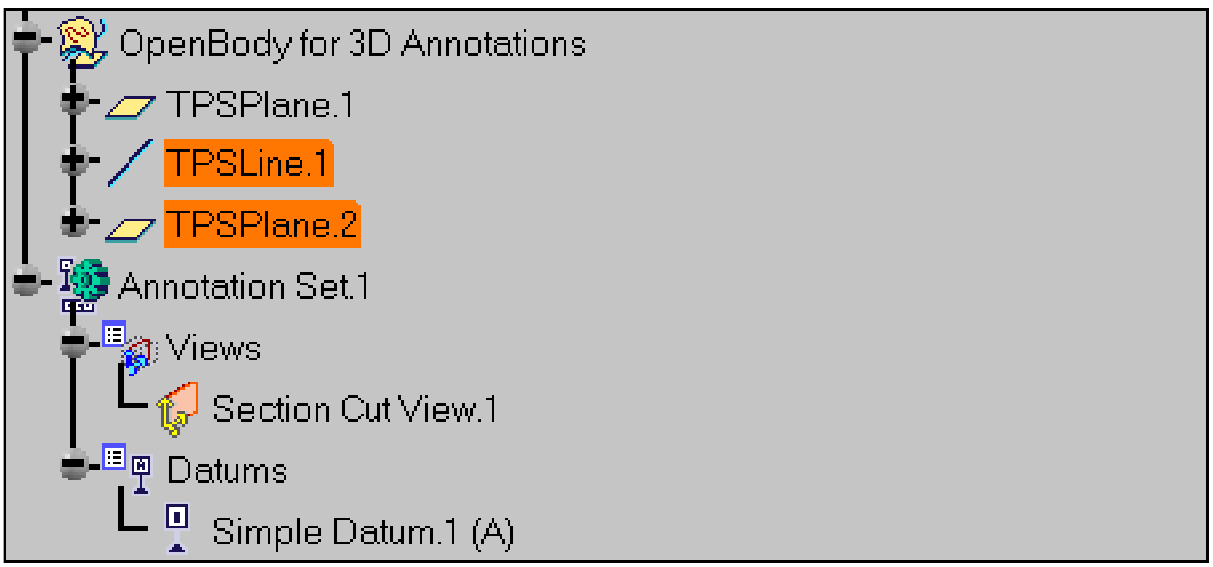


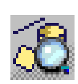
3. Check **Axis** and **Plane** as geometries to be generated and click **OK** in the dialog box.

The axis and the plane are created in the geometry. See also Constructed Geometry options for graphic properties and limits.

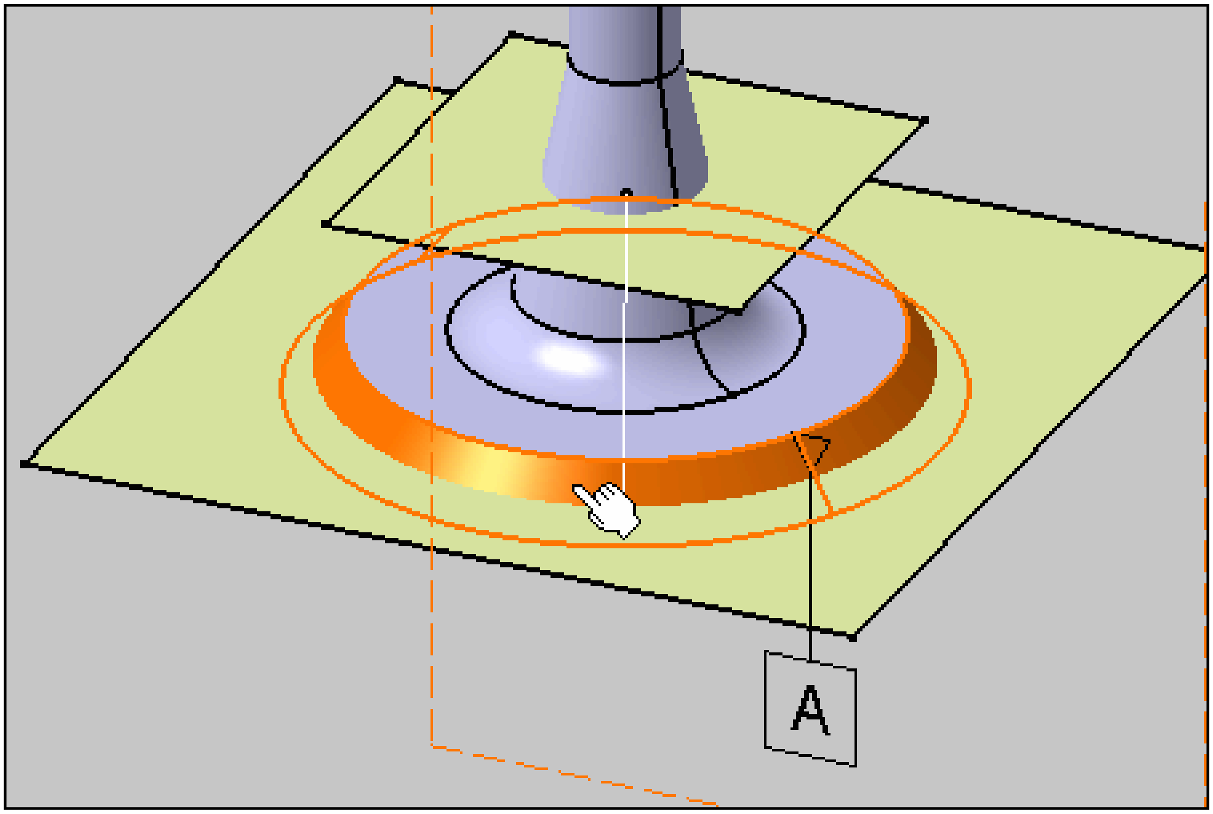


In the specification tree, an open body dedicated to the constructed geometry is added.

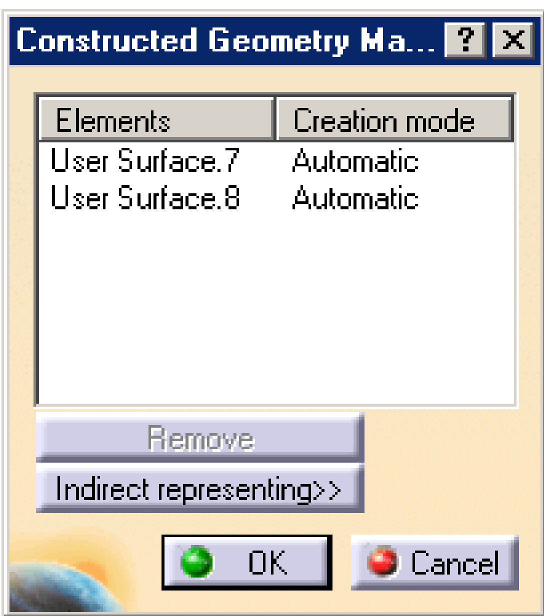


4. Click the **Constructed Geometry Management** icon: 

5. Select the surface as shown.



The **Constructed Geometry Management** dialog box appears and displays the two created geometries and their creation mode.



6. Click **OK**.

Managing Constructed Geometry

This task shows you how to manage constructed geometry.
This functionality allows you to:

- Create a manual constructed geometry. See also Creating an Automatic Constructed Geometry.
- Associate/disassociate an existing geometry as constructed geometry (or representing geometry) of a tolerancing feature (User surface and Group of surfaces) with another existing geometry as represented geometry.
- You can associate an existing constructed geometry (or representing geometry) with another existing geometry. In this case, the constructed geometry previous association is removed, then associated with the new selection.
- An existing geometry cannot be its constructed geometry (or representing geometry) and represented geometry at the same time.
- Query the direct or inherited constructed geometry of a given tolerancing feature.
- Remove existing constructed geometry, but not its indirect representing geometries.
- One or several representing geometries may be referenced by a represented geometry, but a representing geometry element can be referenced by only one represented geometry.

Creating a manual constructed geometry allows you to associate an existing geometry as constructed geometry of a tolerancing feature (User surface and Group of surfaces). The lifecycle of manually constructed geometry is not managed by the application.

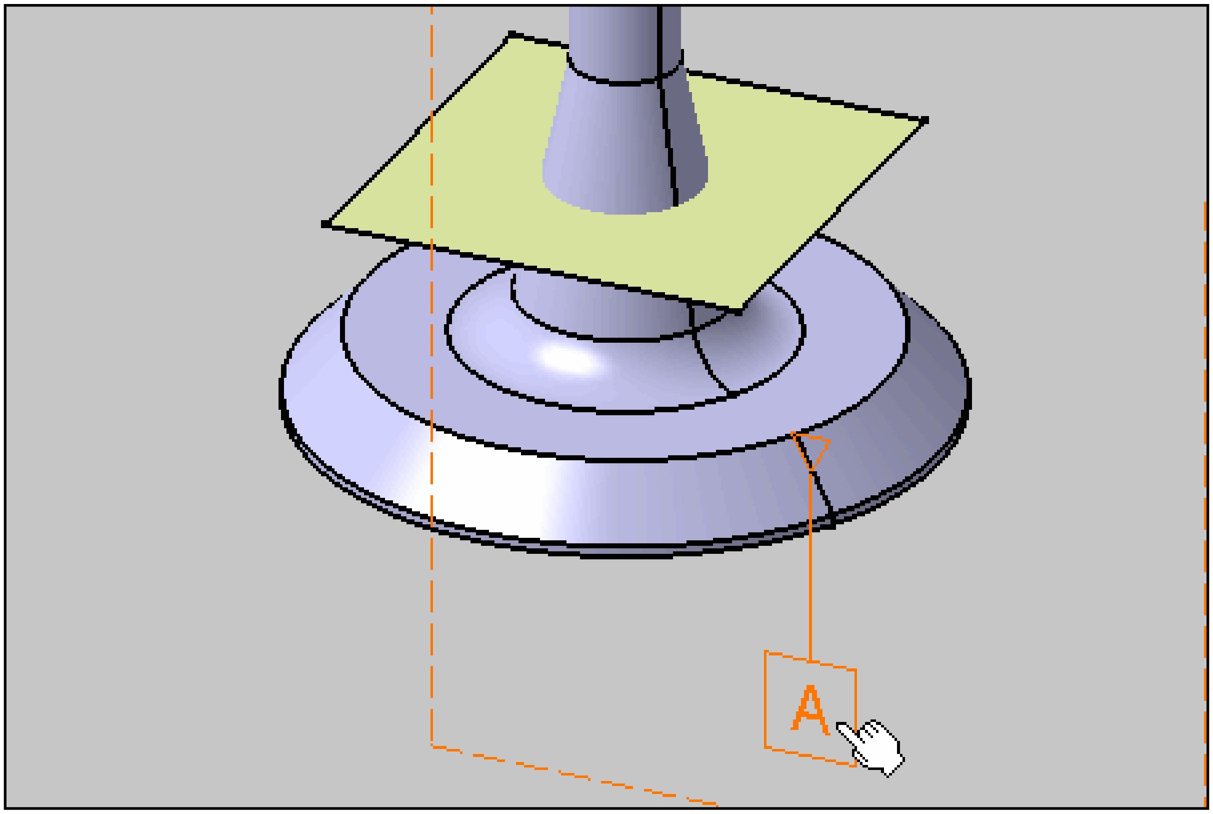
For this, you can select:

- A 3D annotation: the corresponding tolerancing feature is retrieved.
- A geometrical element: the corresponding tolerancing feature is retrieved or created.

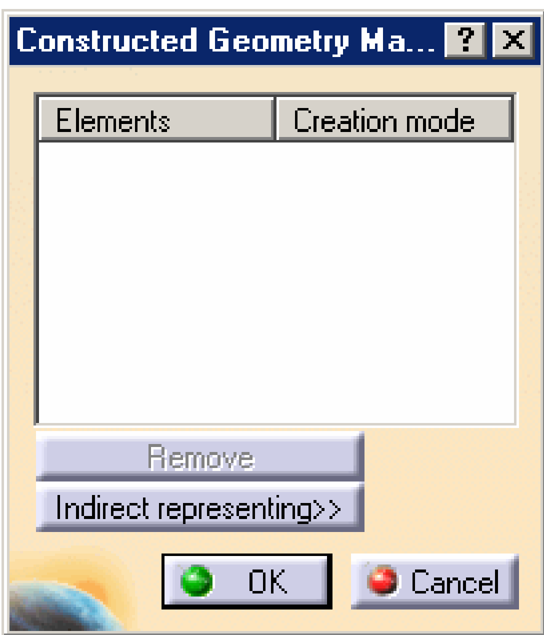
Open the Tolerancing_Annotations_08.CATPart CATPart document.

1. Click the **Constructed Geometry Management** icon: 

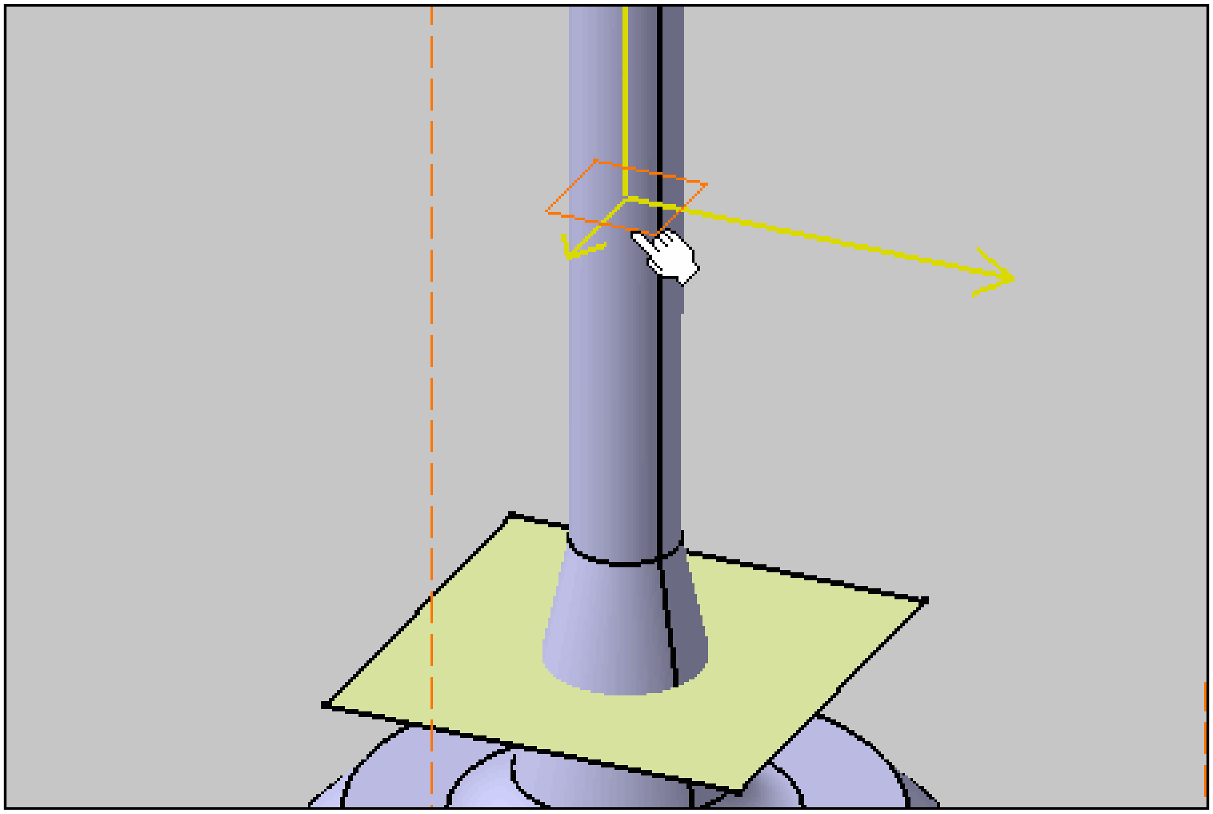
2. Select the datum.



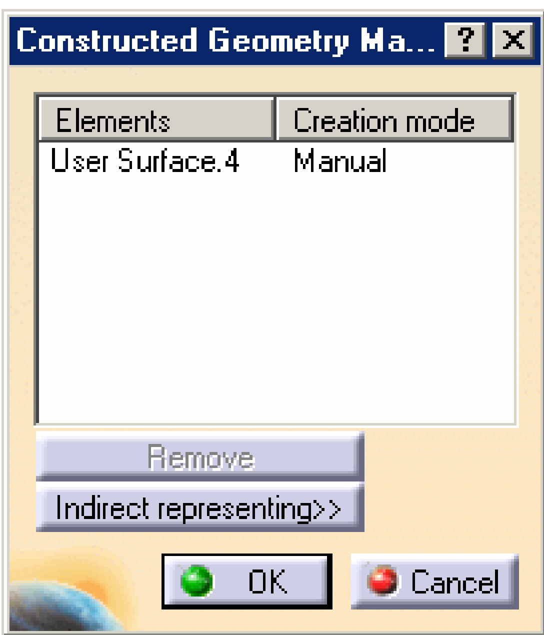
The **Constructed Geometry Management** dialog box appears.



3. Select the reference plane as shown to define the gage plane of the related surface of the annotation.



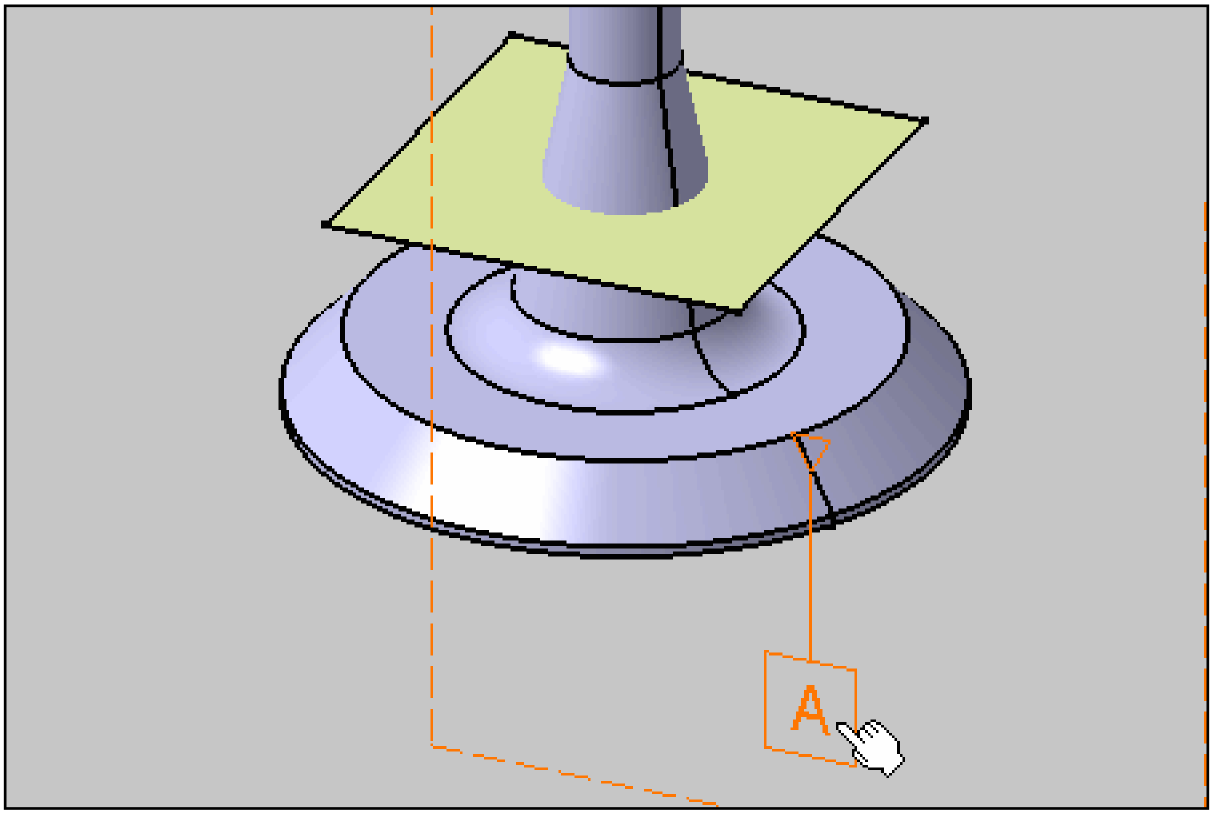
The **Constructed Geometry Management** dialog box is updated, the constructed geometry is created in manual mode: the associativity is managed by the user.



4. Click **OK** in the dialog box.

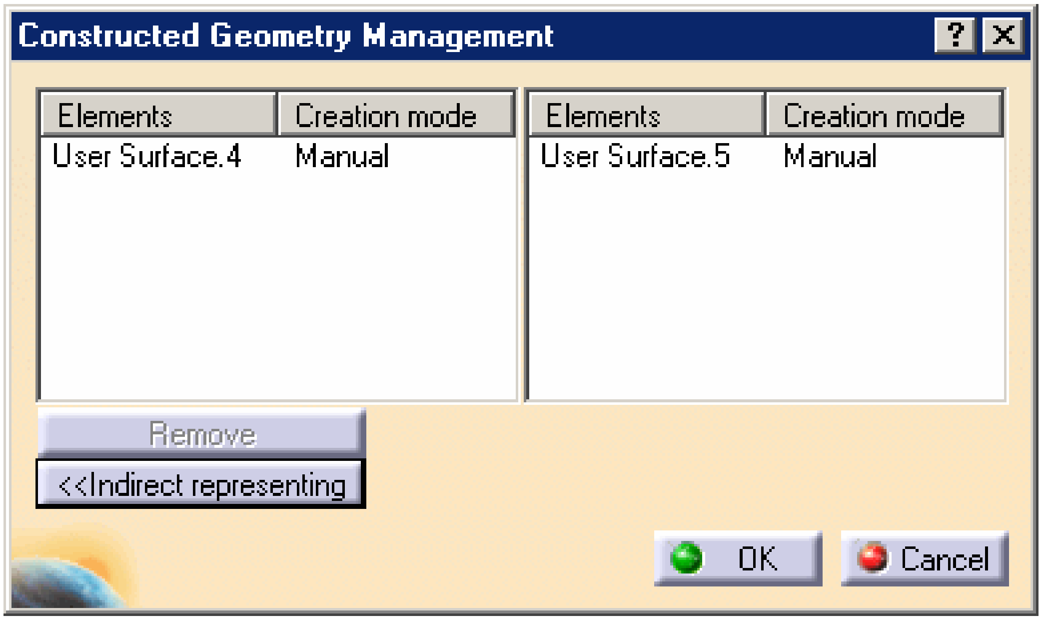
5. Click the **Constructed Geometry Management** icon: 

6. Re-select the previous datum.



7. Click **Indirect representing** in the dialog box.

The **Constructed Geometry Management** dialog box displays the indirect representing of the previously created constructed geometry (**Plane.1** is a representing geometry element, named **User Surface.5**, of the selected reference plane in step 3).



8. Click **OK** in the dialog box.

Interoperability

This section contains interoperability information about the Functional Tolerancing & Annotation workbench.

Optimal CATIA PLM Usability for Functional Tolerancing & Annotation
../../../XomEnglish/xomugfdt.doc/src/xomugfdtxm0200.htm

Optimal CATIA PLM Usability for Functional Tolerancing & Annotation

When working with **ENOVIA V5**, the safe save mode ensures that you only create data in **CATIA V5** that can be correctly saved in **ENOVIA V5**.

ENOVIA V5 offers two different storage modes: Workpackage (Document kept - Publications Exposed) and Explode (Document not kept). Functional Tolerancing & Annotation has been configured to work in the Workpackage mode only.

Functional Tolerancing & Annotation Commands in Enovia V5

All the Functional Tolerancing & Annotation commands are available with the Workpackage mode in Enovia V5.

Technological Package for Functional Tolerancing & Annotation

This functionality allows you to store Functional Tolerancing & Annotations data in a technological package to be saved in ENOVIA VPM context.

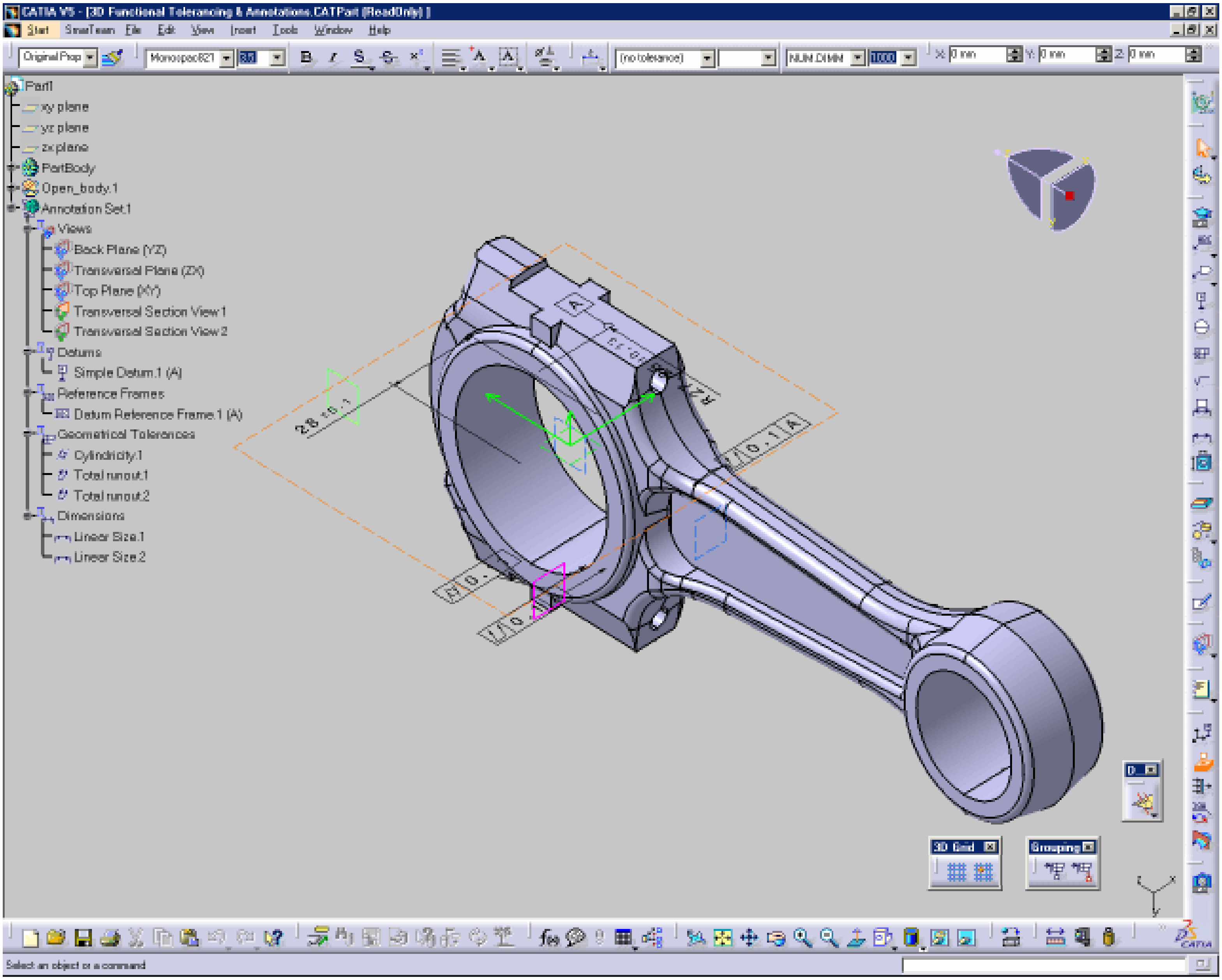
Functional Tolerancing & Annotation Data

All the Functional Tolerancing & Annotation data are storable in a **Technological Package**:

- In case of non existence of a **Technological Package**, a new one will be automatically created (without prompt) when creating new data for the first time. The category of this new **Technological Package** is driven by the applicative command.
- There is no relation between **Technological Package** category and its contents, that is to say that, for instance, a Tolerancing Technological Package may contain or not Functional Tolerancing & Annotations data, and may content other data (DMU Review for instance).

Workbench Description

The **3D Functional Tolerancing & Annotations** workbench looks like this (move the mouse over the various toolbars; the enlarged image and corresponding description will pop-up):



Menu Bar

Annotations Toolbar

Dimension Properties Toolbar

Reporting Toolbar

Style Toolbar

Text Properties Toolbar

Position and Orientation Toolbar

Views/Annotation Planes Toolbar

Visualization Toolbar

Note Object Attribute Toolbar

3D Grid Toolbar

Grouping Toolbar

Menu Bar

This section presents the main menu bar available when you run the application and before creating or opening a document:

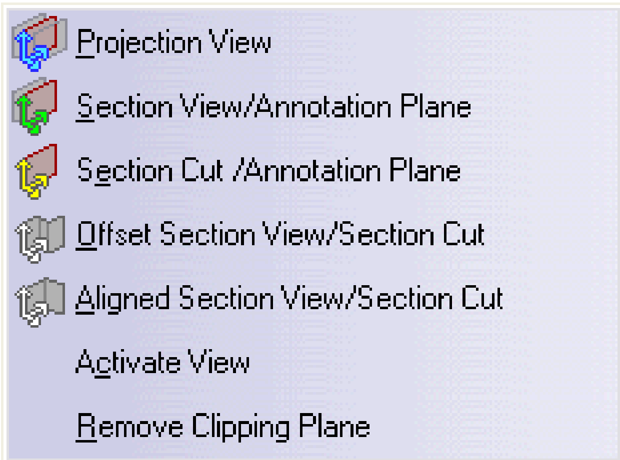
Start	File	Edit	View	Insert	Tools	Windows	Help

Insert

For...	See...
Views/Annotation Planes	Insert -> Views/Annotation Planes Menu
Annotations	Insert -> Annotations Menu
Advanced Replication Tools	Insert -> Advanced Replication Tools Menu
Instantiate From Document...	Instantiating Power Copy

Insert -> Views/Annotation Planes Menu

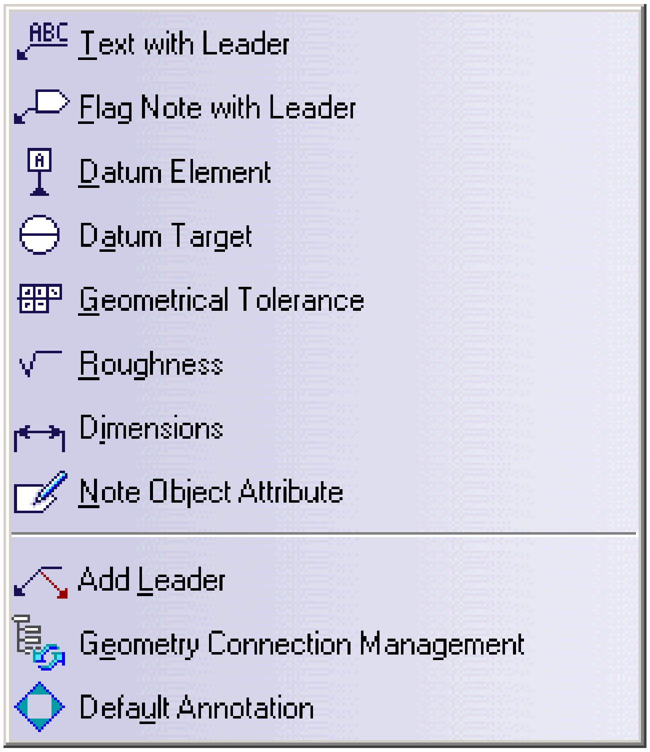
This section presents the Insert -> Views/Annotation Planes menu:



For...	See...
Projection View	Creating a Projection View/Annotation Plane
Section View	Creating a Section View/Annotation Plane
Section Cut	Creating a Section Cut View/Annotation Plane
Offset Section View/Section Cut	Creating an Offset Section View/Section Cut
Aligned Section View/Section Cut	Creating an Aligned Section View/Section Cut

Insert -> Annotations Menu

This section presents the Insert -> Annotations menu:



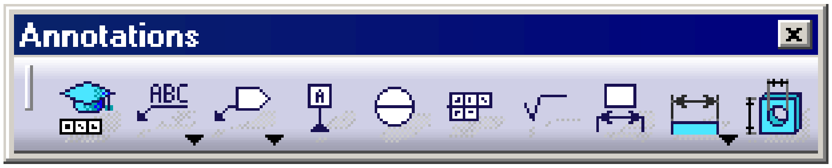
For...	See...
Text with Leader	Creating Texts
Flag Note with Leader	Creating a Flag Note with Leader
Datum	Specifying Datum
Datum Target	Specifying Datum Targets
Geometrical Tolerance	Specifying Geometrical Tolerances
Roughness	Creating Roughness Symbol
Dimensions	Creating Dimensions
Note Object Attribute	Creating Note Object Attribute
Add Leader	Adding Leaders and Using Breakpoint
Geometry Connection Management	Managing Annotation Connection
Default Annotation	Creating an Automatic Default Annotation

Insert -> Advanced Replication Tools Menu

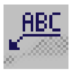
This section presents the Insert -> Advanced Replication Tools menu:

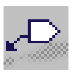
For...	See...
PowerCopy Creation...	Creating Power Copy
Save in Catalog...	Saving Power Copy into a Catalog

Annotations Toolbar

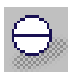


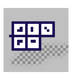
 See Introducing the Tolerancing Advisor

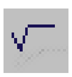
 Jump to Texts Sub-Toolbar

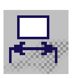
 Jump to Flag Notes Sub-Toolbar

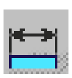
 See Specifying Datum

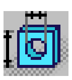
 See Specifying Datum Targets

 See Specifying Geometrical Tolerances

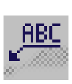
 See Creating Roughness Symbol

 See Creating Basic Dimensions

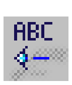
 Jump to Dimensions Sub-Toolbar

 See Generating Dimensions

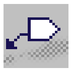
Texts Sub-Toolbar

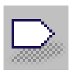
 See Creating Texts

 See Creating Texts

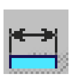
 See Creating Texts

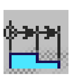
Flag Notes Sub-Toolbar

 See Creating Flag Notes

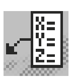
 See Creating Flag Notes

Dimensions Sub-Toolbar

 See Creating Dimensions

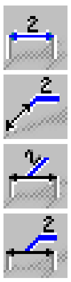
 See Creating Cumulated Dimensions

 See Creating Stacked Dimensions

 See Creating Coordinate Dimensions

 See Creating Curvilinear Dimensions

Dimension Properties Toolbar



See Creating Dimensions

See Dimension Tolerance Display

See Tolerance Display

See Dimension Numerical Display , see Dimension Units reference.

See Precision Display

See Setting Dimension Representations, displayed during dimension creation process only.

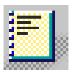
See Setting Dimension Representations, displayed during dimension creation process only.

See Setting Dimension Representations, displayed during dimension creation process only.

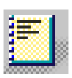
See Setting Dimension Representations, displayed during dimension creation process only.

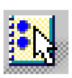
See Setting Dimension Representations, displayed during dimension creation process only.

Reporting Toolbar

 Jump to Reports Sub-Toolbar

Reports Sub-Toolbar

 See Generating a Check Report

 See Customizing the Reporting

Style Toolbar

See Defining Default Properties

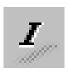

See Copying Graphic Properties

Text Properties Toolbar

See Setting Graphic Properties

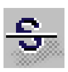
See Setting Graphic Properties

See Setting Graphic Properties



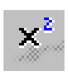
See Setting Graphic Properties

See Setting Graphic Properties

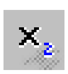


See Setting Graphic Properties

See Setting Graphic Properties



See Setting Graphic Properties

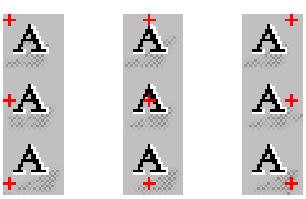


See Setting Graphic Properties

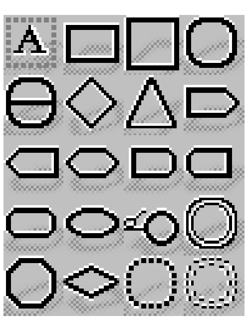
See Setting Graphic Properties

See Setting Graphic Properties

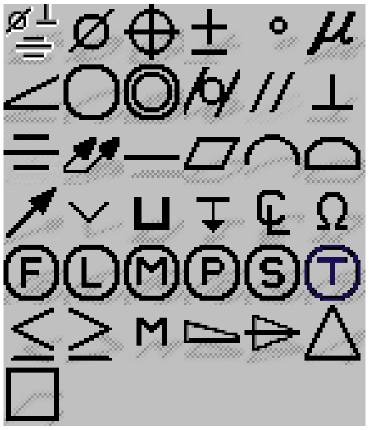
See Setting Graphic Properties



See Setting Graphic Properties

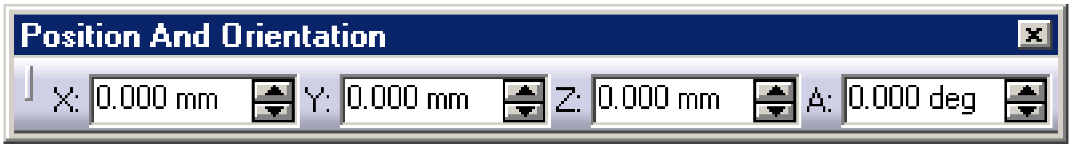


See Setting Graphic Properties



See Setting Graphic Properties

Position and Orientation Toolbar



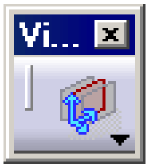
See Moving Annotations

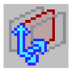
See Moving Annotations

See Moving Annotations

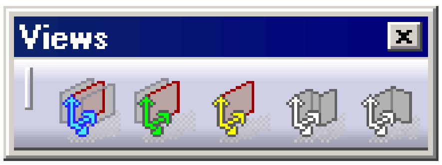
See Moving Annotations

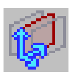
Views/Annotation Planes Toolbar

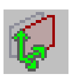


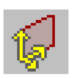
 Jump to Views Sub-Toolbar

Views Sub-Toolbar

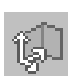


 See Creating a Projection View/Annotation Plane

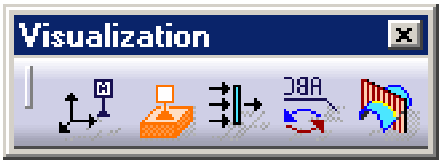
 See Creating a Section View/Annotation Plane

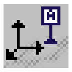
 See Creating a Section Cut View/Annotation Plane

 See Creating an Offset Section View/Section Cut

 See See Creating an Aligned Section View/Section Cut

Visualization Toolbar

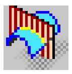


 See Disabling/Enabling Annotations

 See Querying 3D Annotations

 See Filtering Annotations

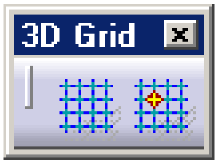
 See Mirroring Annotations

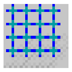
 See Clipping Annotations Plane

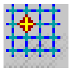
Note Object Attribute Toolbar

 See Creating Note Object Attribute

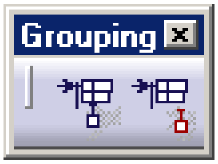
3D Grid Toolbar

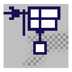


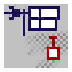
 See Using 3D Grid

 See Using 3D Grid

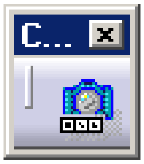
Grouping Toolbar



 See Grouping Annotations Automatically

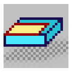
 See Grouping and Ordering Annotations

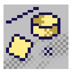
Capture Toolbar

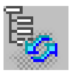


See Creating an Annotation Capture:
This command open the Tolerancing & Annotation Captures workshop.

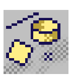
Geometry for 3D Annotations Toolbar

 See Creating a Partial Surface

 Jump to Geometry for 3D Annotations Sub-Toolbar

 See Managing Annotation Connection

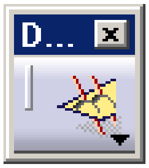
Geometry for 3D Annotations Sub-Toolbar

 See Creating an Automatic Constructed Geometry

 See Managing Constructed Geometry

 See Dimensioning and Tolerancing Threads using the Tolerancing Advisor

Deviations Toolbar (Compact)

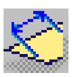


 Jump to Deviations Sub-Toolbar (Extended)

Deviations Sub-Toolbar (Extended)

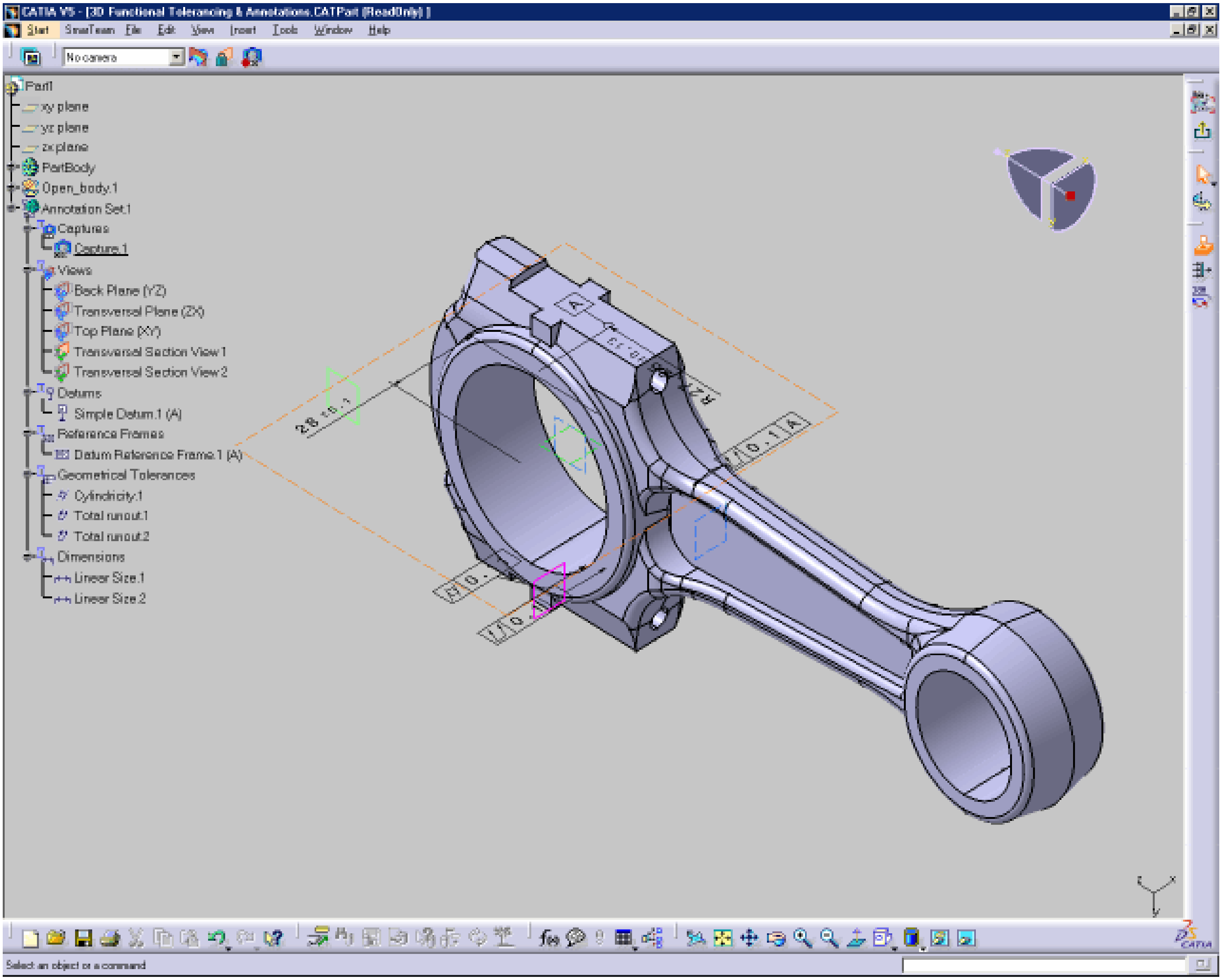
 See Creating a Deviation

 See Creating a Correlated Deviation

 See Creating a Distance Between Two Points

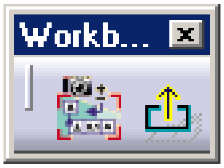
Workshop Description

The **Tolerancing & Annotation Captures** workshop looks like this (move the mouse over image's links and have the enlarged image and corresponding description pop up):



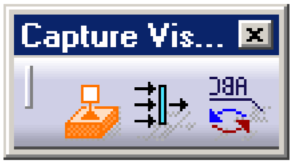
Workbench Toolbar
Capture Visualization Toolbar
Capture Options Toolbar
Camera Toolbar

Workbench Toolbar



 See Creating an Annotation Capture

Capture Visualization Toolbar



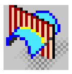
 See Managing Associativity Between Elements

 See Filtering Annotations

 See Mirroring Annotations

Capture Options Toolbar

See Managing Capture Options

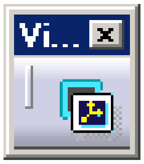


See Managing Capture Options

See Managing Capture Options

See Managing Capture Options

Camera Toolbar



 See Creating a Camera

Customizing

Before you start your first working session, you can customize the way you work to suit your habits.
This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.

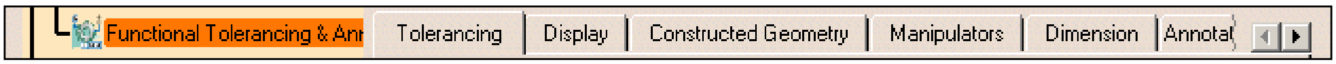
1. Select the **Tools -> Options** command.

   The **Options** dialog box opens.

2. Select the **Mechanical Design** category in the left-hand box.

3. Select the **Functional Tolerancing & Annotation** sub-category.

   Eight tabs are displayed:



- The Tolerancing tab lets you set the tolerancing options.
- The Display tab lets you define the annotation display options.
- The Constructed Geometry
- The Manipulators tab lets you set the manipulator options.
- The Dimension
- The Annotation tab lets you the annotation display options.
- The Tolerances
- The View/Annotation Plane tab lets you define the view/annotation plane options.

4. Select the **Infrastructure** category in the left-hand box.

5. Select the **Product Structure** sub-category.

   Two tabs also interfere with Functional Tolerancing & Annotation.

- Cache Management
- Cgr Management

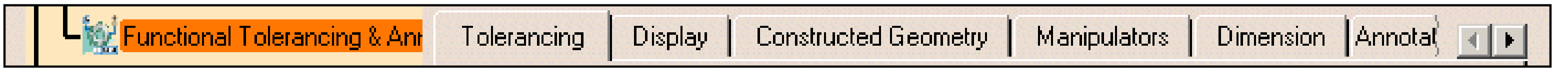
6. Select the **General** category in the left-hand box.

7. Select the **Display** sub-category.

   One tab also interfere with Functional Tolerancing & Annotation.

- Navigation

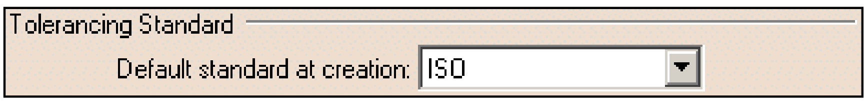
Tolerancing



This page deals with the options concerning:

- The Tolerancing Standard.
- The Semantic Control.
- The Leader associativity to the geometry.
- The Rotation.
- The Note Object Attribute.

Tolerancing Standard



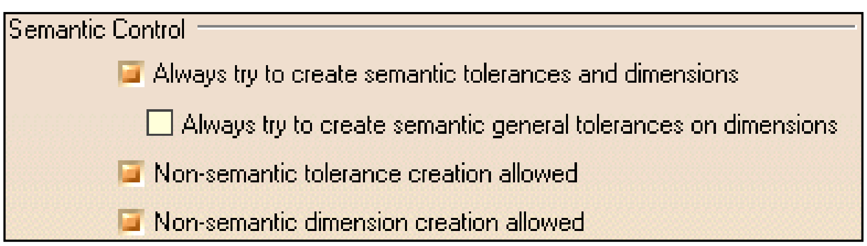
Default standard at creation

Five conventional standards are available:

- **ASME**: American Society for Mechanical Engineers
- **ASME 3D**: American Society for Mechanical Engineers
- **ANSI**: American National Standards Institute
- **JIS**: Japanese Industrial Standard
- **ISO**: International Organization for Standardization

By default, the **ISO** standard is selected.

Semantic Control



Defines the semantic control options:

Always try to create semantic tolerances and dimensions

Tries to set as semantic an annotation during its creation. Applicable on annotation which can be semantic.

By default, this option is selected.

Always try to create semantic general tolerances on dimensions

Tries to set as semantic a general tolerances dimension during its creation. Applicable on dimension which can be semantic, available only if Always try to create semantic tolerances and dimensions is selected.

By default, this option is not selected.

Non-semantic tolerance creation allowed

Defines whether non-semantic tolerances creation is allowed.

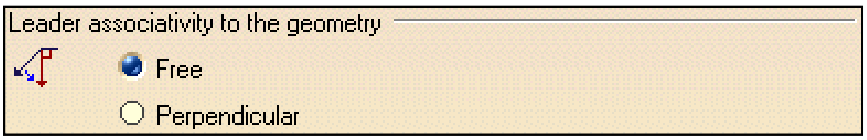
By default, this option is selected.

Non-semantic dimension creation allowed

Defines whether non-semantic dimensions creation is allowed.

By default, this option is selected.

Leader associativity to the geometry

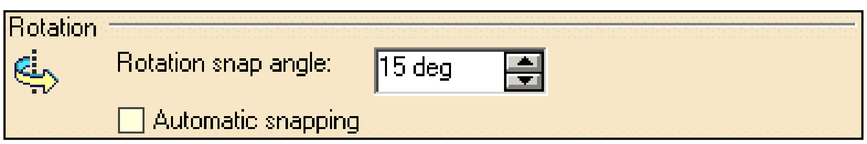


Defines the leader associativity options:

- **Free**: specifies that leader annotations are freely positioned relative to their geometrical elements.
- **Perpendicular**: specifies that leader annotations are positioned perpendicular to their geometrical elements.

By default, the **Free** option is selected.

Rotation



Defines the rotation options:

Rotation snap angle

Defines an angle value for rotating elements. This option is used to rotate text elements (text, frame, or leader).

By default, the snap angle value is 15deg.

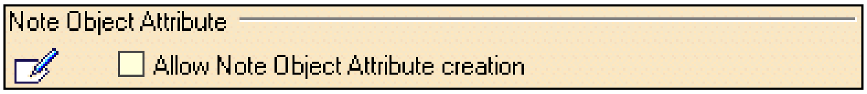
Automatic snapping

Defines whether the rotation will be snapped to the angle value.

By default, this option is not selected.

Note Object Attribute

Defines the Note Object Attribute options:

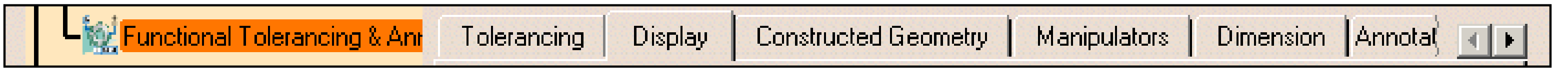


Allow Note Object Attribute creation

Defines whether note object attribute may be created by user. Enable or disable the icon and menu item commands.

By default, this option is not selected.

Display



This page deals with the options concerning:

- The Non-Semantic.
- The Grid.
- The Annotations in Specification Tree.
- The Partial Surface.
- The Annotation Parameters.
- The Surface Normal.
- The 3D Annotation Query.

Non-Semantic

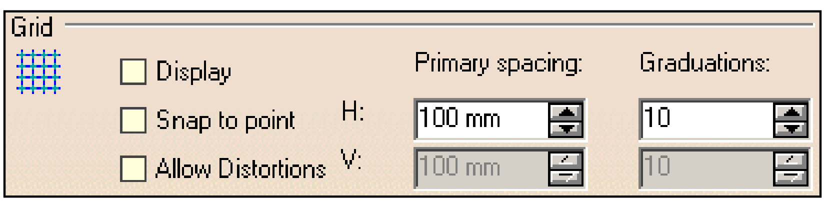
Defines the non-semantic options:

Mark non-semantic annotations

Defines whether non-semantic annotations (datum elements, datum targets, geometrical tolerances, linear and angular dimensions) are marked with a wavy red line in the specification tree and in the geometry.

By default, this option is not selected.

Grid



Defines the grid options:

Display

Defines whether the grid is displayed.

By default, this option is not selected.

Snap to point

Defines whether annotations are snapped to the grid's point.

By default, this option is not selected.

Allow Distortions

Defines whether grid spacing and graduations are the same horizontally and vertically.

By default, this option is not selected.

H Primary spacing

Defines the grid's horizontal spacing.

By default, the value is 100mm.

H Graduations

Defines the grid's horizontal graduations.

By default, the number of graduation is 10.

V Primary spacing

Defines the grid's vertical spacing, available only if Allow Distortions is selected.

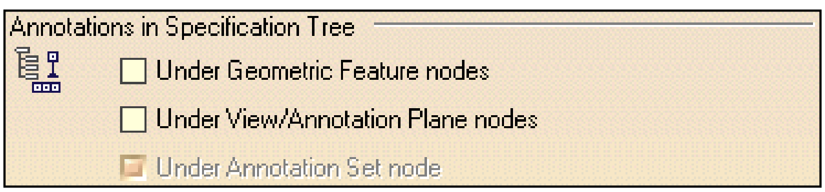
By default, the value is 100mm.

V Graduations

Defines the grid's vertical graduations, available only if Allow Distortions is selected.

By default, the number of graduation is 10.

Annotations in Specification Tree



Defines the annotations in specification tree options:

Under Geometric Feature nodes

Defines that 3D annotations should be displayed under the geometric feature nodes in the specification tree. This lets you view 3D annotations under the Part Design or Generative Shape Design feature nodes to which they are applied.

By default, this option is not selected.

Under View/Annotation Plane nodes

Defines that 3D annotations should be displayed under the view/annotation plane nodes in the specification tree. This lets you view 3D annotations under the view node to which they are linked.

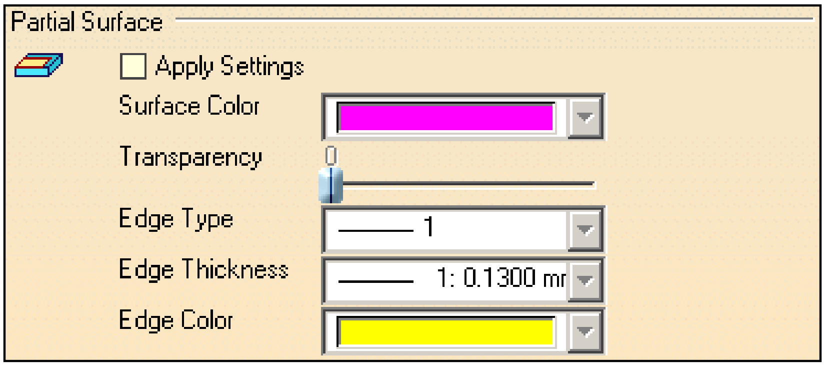
By default, this option is not selected.

Under Annotations Set node

Defines that 3D annotations should be displayed under the annotation set node in the specification tree, available only if Under View/Annotation Plane nodes is selected.

By default, this option is selected.

Partial Surface



Defines the partial surface options:

Apply Settings

Defines whether the following settings are applied while creating a partial surface feature.

By default, this option is not selected.

Surface Color

Defines the surface color of the partial surface.

By default, the color is magenta. See the screen capture.

Transparency

Defines the surface color transparency of the partial surface.

By default, the value is 0.

Edge Type

Defines the edge type of the partial surface's border.

By default, the edge type is 1.

Edge Thickness

Defines the edge thickness of the partial surface's border.

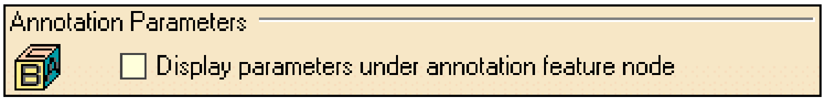
By default, the edge thickness is 1.

Edge Color

Defines the edge color of the partial surface's border.

By default, the color is yellow. See the screen capture.

Annotation Parameters



Defines the annotation parameter options:

Display parameters under annotation feature node

Defines that knowledge parameters (such as tolerance values, datum label, etc.) of annotations should be displayed under the annotation feature node in the specification tree; also defines that feature parameters of dimensions (accessible through the **Edit Generative Parameter** command) should be displayed under the dimension feature node in the specification tree.
Note that in order to have the value of the parameters displayed in the specification tree, you need to select the **With value** knowledge setting in **Tools -> Options -> General -> Parameters and Measure -> Knowledge** tab.

By default, this option is not selected.

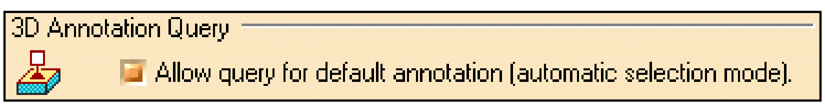
Surface Normal

Display for shifted profile tolerance

Defines whether the normal of all the selected surfaces are displayed, or not, when a shifted profile tolerance is specified or queried.

By default, this option is not selected.

3D Annotation Query

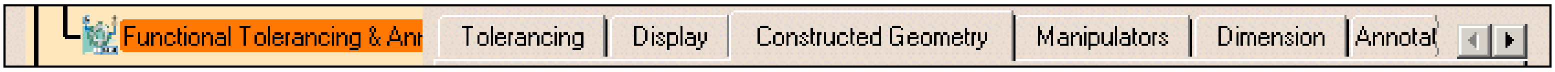


Allow query for default annotation (automatic selection mode)

Defines whether the query for default annotation is allowed, or not. This option allows you to highlight the related annotations or geometrical elements with the selected annotation or the related annotations with the selected geometrical element.

By default, this option is selected.

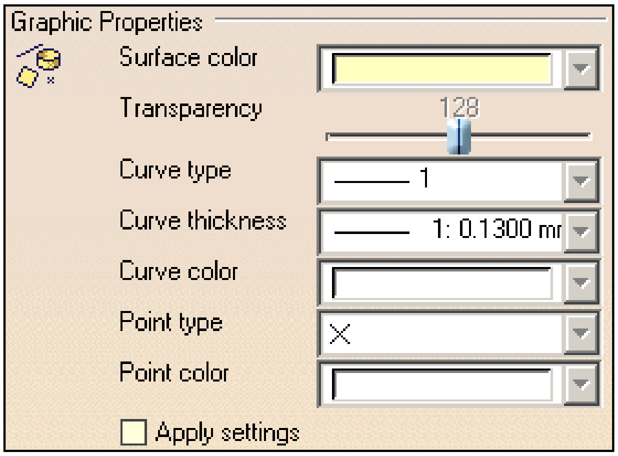
Constructed Geometry



This page deals with the options concerning:

- The Graphic Properties.
- The Limits.
- The Automatic Creation.

Graphic Properties



Defines the graphic properties options:

Surface color

Defines the constructed geometry surface color for plane and cylinder.

By default, the color is light yellow. See the screen capture.

Transparency

Defines the surface color transparency of the constructed geometry.

By default, the value is 128.

Curve type

Defines the constructed geometry curve type.

By default, the curve type is 0.

Curve thickness

Defines the constructed geometry curve thickness.

By default, the curve thickness is 1.

Curve color

Defines the constructed geometry curve color.

By default, the color is white. See the screen capture.

Point type

Defines the constructed geometry point type.

By default, the point type is a cross symbol. See the screen capture.

Point color

Defines the constructed geometry point color.

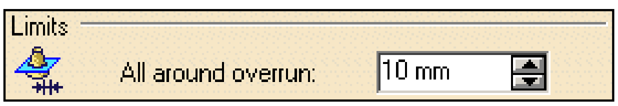
By default, the color is white. See the screen capture.

Apply settings

Defines whether all the graphic properties settings are applied or not.

By default, this option is not selected.

Limits



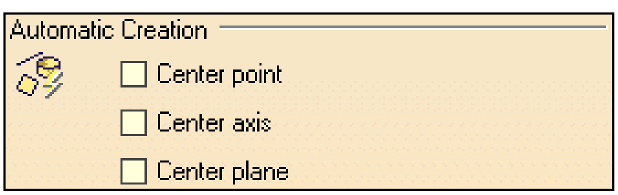
Defines the limit options:

All around overrun

Defines the minimal limit between the constructed geometry and its related geometry.

By default, the value is 10mm.

Automatic Creation



Defines the automatic creation options:

Center point

Defines whether all the center point's constructed geometry is automatically created or not, for circle center, sphere center.

By default, this option is not selected.

Center axis

Defines whether all the center axis's constructed geometry is automatically created or not, for cylinder, cone.

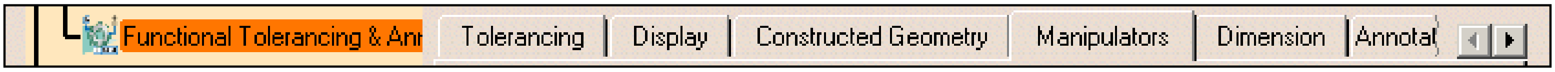
By default, this option is not selected.

Center plane

Defines whether all the center plane's constructed geometry is automatically created or not, for slot.

By default, this option is not selected.

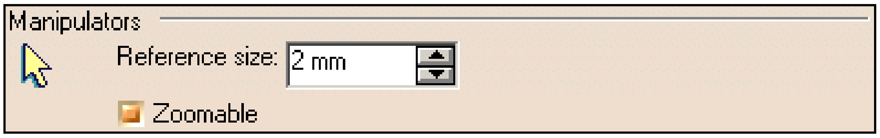
Manipulators



This page deals with the options concerning:

- The Manipulators.
- The Dimension Manipulators.

Manipulators



Defines the manipulator options:

Reference size

Defines the annotation manipulator's size.

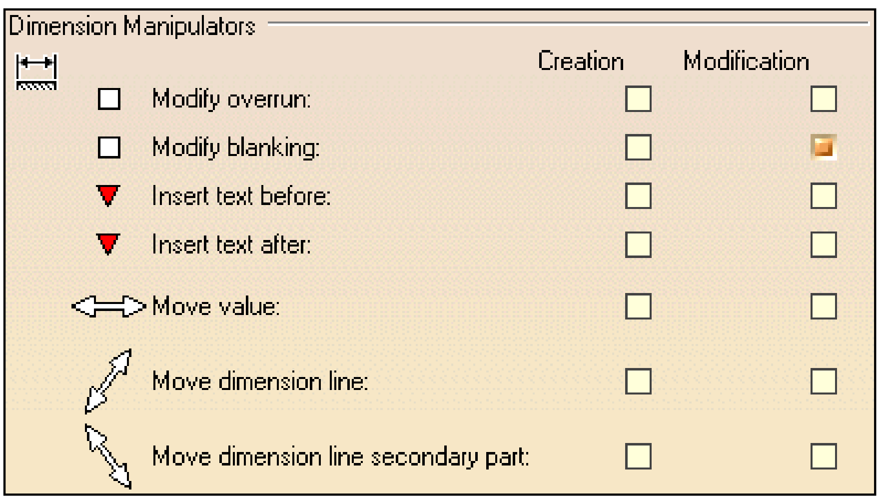
By default, the reference size is 2mm.

Zoomable

Defines whether the annotation manipulator is zoomable or not.

By default, this option is selected.

Dimension Manipulators



Defines the dimension manipulator options:

Modify overrun

Defines whether overrun extension lines can be modified during creation or modification.

By default, the creation and modification options are not selected.

Modify blanking

Defines whether blanking can be modified during creation or modification.

By default, the creation option is not selected and the modification option is selected.

Insert text before

Defines whether a text before can be inserted during creation or modification.

By default, the creation and modification options are not selected.

Insert text after

Defines whether a text after can be inserted during creation or modification.

By default, the creation and modification options are not selected.

Move value

Defines whether only the value can be moved during creation or modification.

By default, the creation and modification options are not selected.

Move dimension line

Defines whether only the dimension line can be moved during creation or modification.

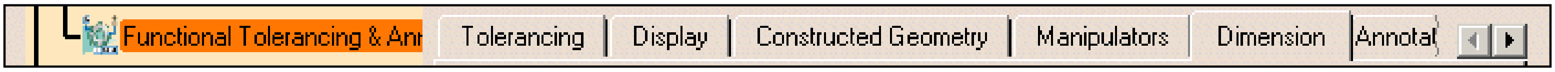
By default, the creation and modification options are not selected.

Move dimension line secondary part

Defines whether only the dimension line secondary part can be moved during creation or modification.

By default, the creation and modification options are not selected.

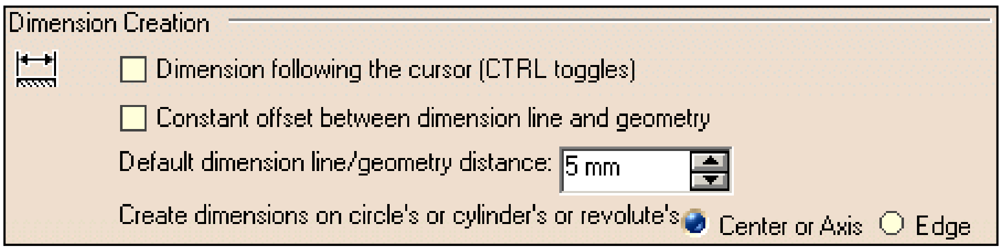
Dimension



This page deals with the options concerning:

- The Dimension Creation.
- The Move.
- The Line-Up.
- The Dimension related to an origin.

Dimension Creation



Defines the dimension creation options:

Dimension following the cursor (CTRL toggles)

Defines whether the dimension line is positioned according to the cursor, following it dynamically during the creation process or not.

By default, this option is not selected.

Constant offset between dimension line and geometry

Defines whether the distance between the created dimension and the geometry remains the same when you move the geometry or not.

By default, this option is not selected.

Default dimension line/geometry distance

Defines the value at which the dimension is created from the geometry.
If you create associativity between the dimension and the geometry, you can define the value at which the dimension will remain positioned.

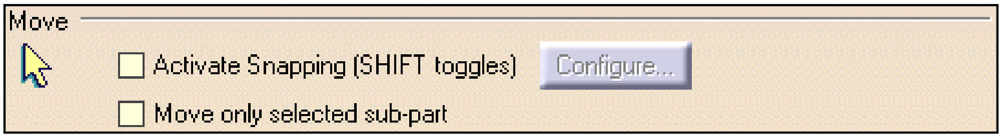
By default, the distance is 5mm.

By default, dimension circles on their

Defines the dimension you will create between a circle and another element will be either on the circle center (or axis cylinder) or on the circle edge.

By default, the circle center (or axis cylinder) option is selected.

Move



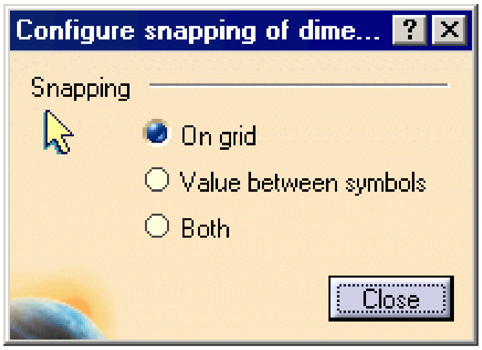
Defines the annotation move options:

Activate Snapping (SHIFT toggles)

Defines whether the dimension will be snapped .

By default, this option is not selected.

Additionally click the **Configure...** button. In the **Configure snapping of dimension** dialog box, specify whether the dimension should be snapped on the grid, or whether the dimension value should be located at its default position between symbols (it will work only if the cursor is between the symbols), or both.



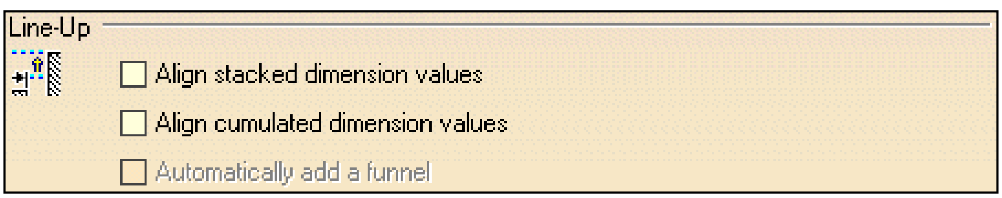
By default, this **On grid** option is selected.

Move only selected sub-part

Defines whether only a dimension sub-part (text, line, etc...) will be moved.

By default, this option is not selected.

Line-Up



Defines the line-up options:

Align stacked dimension values

Defines whether the values of a group of stacked dimensions are aligned on the value of the smallest dimension of the group.

By default, this option is not selected.

Align cumulated dimension values

Defines whether the values of a group of cumulated dimensions are aligned on the value of the smallest dimension of the group.

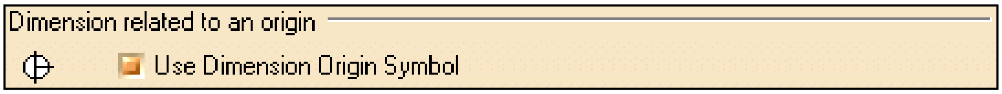
By default, this option is not selected.

Automatically add a funnel

Defines whether the value of a cumulated dimension requires a funnel added automatically to be displayed correctly, available only if Align cumulated dimension values is selected.

By default, this option is not selected.

Dimension related to an origin



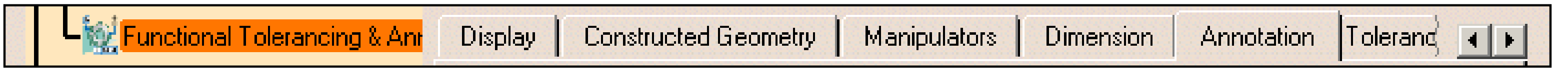
Defines the dimension origin options:

Use Dimension Origin Symbol

Defines whether the dimension origin symbol is used.

By default, this option is selected.

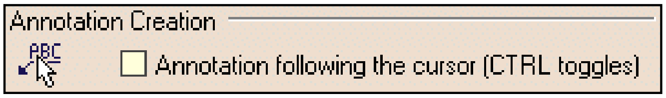
Annotation



This page deals with the options concerning:

- The Annotation Creation.
- The Geometrical Tolerance.

Annotation Creation



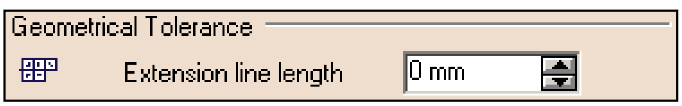
Defines the annotation creation options:

Annotation following the cursor (CTRL toggles)

Defines whether the annotation is positioned according to the cursor, following it dynamically during the creation process or not.

By default, this option is not selected.

Geometrical Tolerance



Extension line length

Defines the extension line length between the geometrical frame and the its leader.

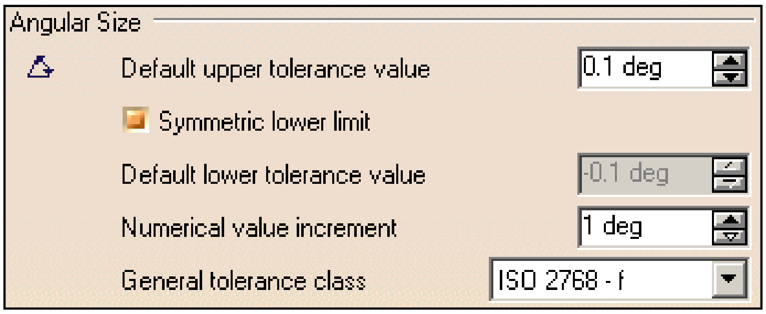
By default, the length is 0mm..

Tolerances

This page deals with the options concerning:

- The Angular Size.
- The Linear Size.
- The Geometrical Tolerance.

Angular Size



Defines the angular size options:

Default upper tolerance value

Defines the default upper tolerance value for angular size.

By default, the upper tolerance is 0.1deg.

Symmetric lower limit

Defines whether the default lower tolerance value is symmetric in relation to the default upper tolerance value.

By default, this option is selected.

Default lower tolerance value

Defines the default lower tolerance value for angular size, enable when Symmetric lower limit is not selected.

By default, the lower tolerance is -0.1deg.

Numerical value increment

Defines the increment for angular size numerical value.

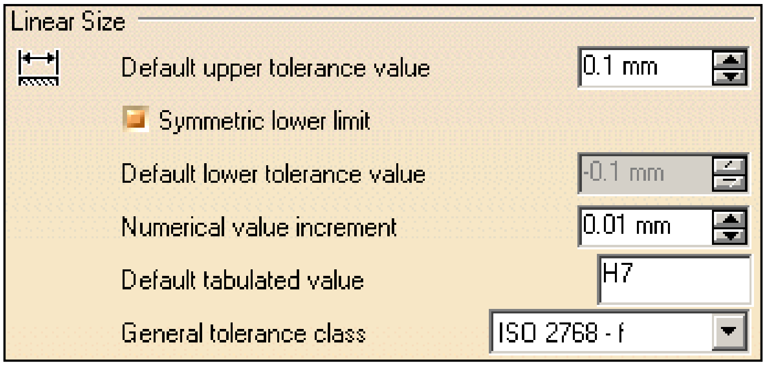
By default, the increment is 1deg.

General tolerance class

Defines the general tolerance class for angular size, according to the selected standard in Tolerancing Standard.

By default, the general tolerance class is ISO 2768 - f.

Linear Size



Defines the linear size options:

Default upper tolerance value

Defines the default upper tolerance value for linear size.

By default, the upper tolerance is 0.1mm.

Symmetric lower limit

Defines whether the default lower tolerance value is symmetric in relation to the default upper tolerance value.

By default, this option is selected.

Default lower tolerance value

Defines the default lower tolerance value for linear size, enable when Symmetric lower limit is not selected.

By default, the lower tolerance is -0.1mm.

Numerical value increment

Defines the increment for linear size numerical value.

By default, the increment is 0.01mm.

Default tabulated value

Defines the default tabulated for linear size.

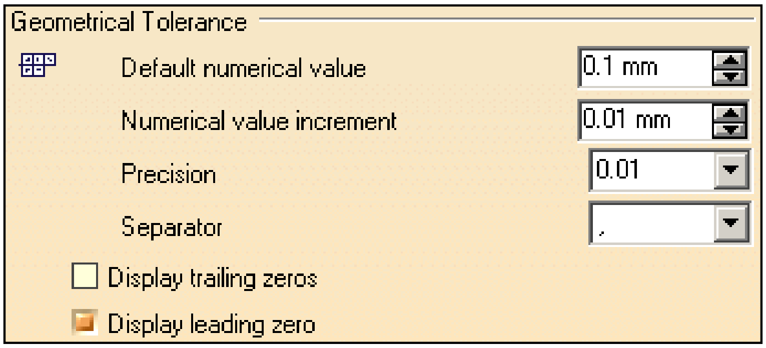
By default, the tabulated value is H7.

General tolerance class

Defines the general tolerance class for angular size, according to the selected standard in Tolerancing Standard.

By default, the general tolerance class is ISO 2768 - f.

Geometrical Tolerance



Defines the geometrical tolerance options:

Default numerical value

Defines the default numerical for geometrical tolerance.

By default, the numerical value is 0.1mm.

Numerical value increment

Defines the increment for the to geometrical tolerance numerical value.

By default, the numerical increment value is 0.01mm.

Precision

Defines the geometrical tolerance precision after the numerical separator.

By default, the precision is 0.01.

Separator

Defines the geometrical tolerance symbol used as numerical separator.

By default, the separator is a comma ",".

Display trailing zeros

Defines whether "0"s complete the number of digit displayed after the separator, according to the precision.

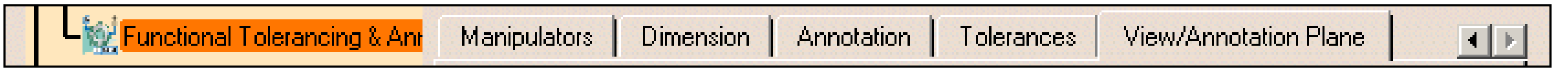
By default, this option is not selected.

Display leading zeros

Defines whether the "0" before the numerical separator, when value is less than 1, is displayed.

By default, this option is selected.

View/Annotation Plane



This page deals with the options concerning:

- The View/Annotation Plane Associativity.
- The View/Annotation Plane Display.

View/Annotation Plane Associativity

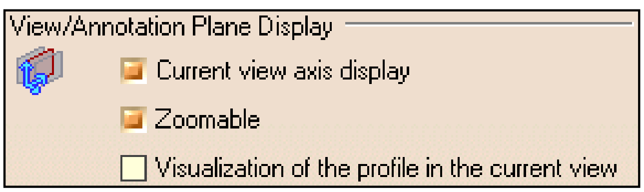
Defines the View/Annotation Plane associativity options:

Create views associative to geometry

Creates views associative to the geometry, so that views and their annotations are automatically updated when the geometry is modified.

By default, this option is selected.

View/Annotation Plane Display



Defines the View/Annotation Plane display options:

Current view axis display

Defines whether the active annotation plane axis system is displayed.

By default, this option is selected.

Zoomable

Defines whether the annotation plane axis is zoomable.

By default, this option is selected.

Visualization of the profile in the current view

Defines whether the view/annotation plane profile on the part/product is displayed.

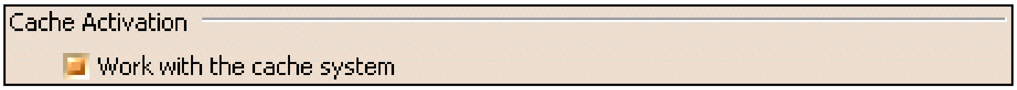
By default, this option is not selected.

Cache Management for CATProduct and CATProcess Document

This page deals with the options concerning:

- The Cache Activation.

Cache Activation



Please refer to *Infrastructure user's guide* to know more about the Product Structure **Cache Management** options.

Work with the cache system

You need to select this option to work with the cache system. When selected, this option allows you work in **Visualization** mode, otherwise you are working in **Design** mode.

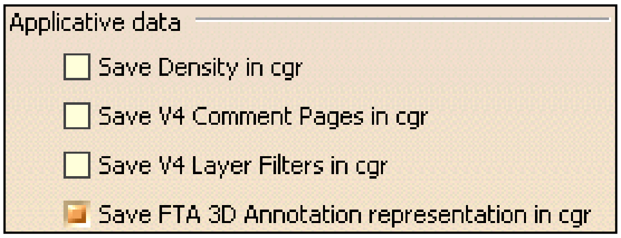
By default, this option is not selected.

Cgr Management for 3D Annotation

This page deals with the options concerning:

- The Applicative data.

Applicative data



Please refer to *Infrastructure user's guide* to know more about the Product Structure **Cgr Management** options.

Save FTA 3D Annotation representation in cgr

You need to select this option to add the 3D annotations representation contained in a CATProduct or a CATProcess document to the generated cgr documents. This option is taken into account when the Cache Activation option is selected only.

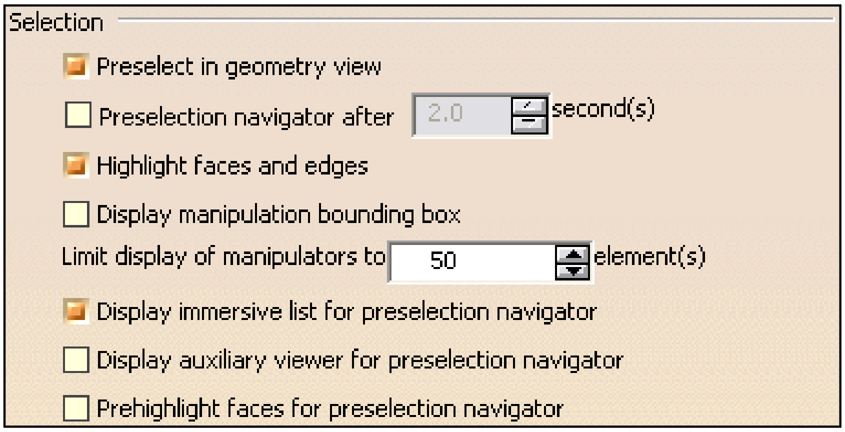
By default, this option is not selected.

Highlighting of the Related Geometry for 3D Annotation

This page deals with the options concerning:

- The Selection.

Selection



Please refer to *Infrastructure user's guide* to know more about the Display **Selection** options.

Highlight faces and edges

You can select this option to improve the highlight of the related geometry when selected an annotation.

By default, this option is not selected.

Reference Information

This section contains reference information about 3D Functional Tolerancing & Annotations workbench.

Standards

Standards (such as ISO, JIS, ANSI, ASME, etc. or company standards) specify the properties and styles of tolerancing and annotation elements such as dimensions, annotations, etc. so that they will be applied to all elements of a given type within a single part or product, as well as in all parts or products which use the same parent standard.

Standards are managed by an administrator. Each standard is defined in an XML file, which makes it possible to customize globally the appearance and behavior of tolerancing and annotation elements.

Standards in the 3D Functional Tolerancing and Annotation workbench are those used and customized in the Drafting workbenches: they are 2D standards, transposed to 3D. As they are defined for the Drafting workbenches, some standard parameters apply only to Drafting applications. However, a great number of parameters apply to both the 3D Functional Tolerancing and Annotation and Drafting workbenches.

The values of the parameters in the file are taken into account when the first Functional Tolerancing and Annotation view is created, based on the current parent standard. Once the first view has been created, modifying the standard file will not affect this CATPart document.

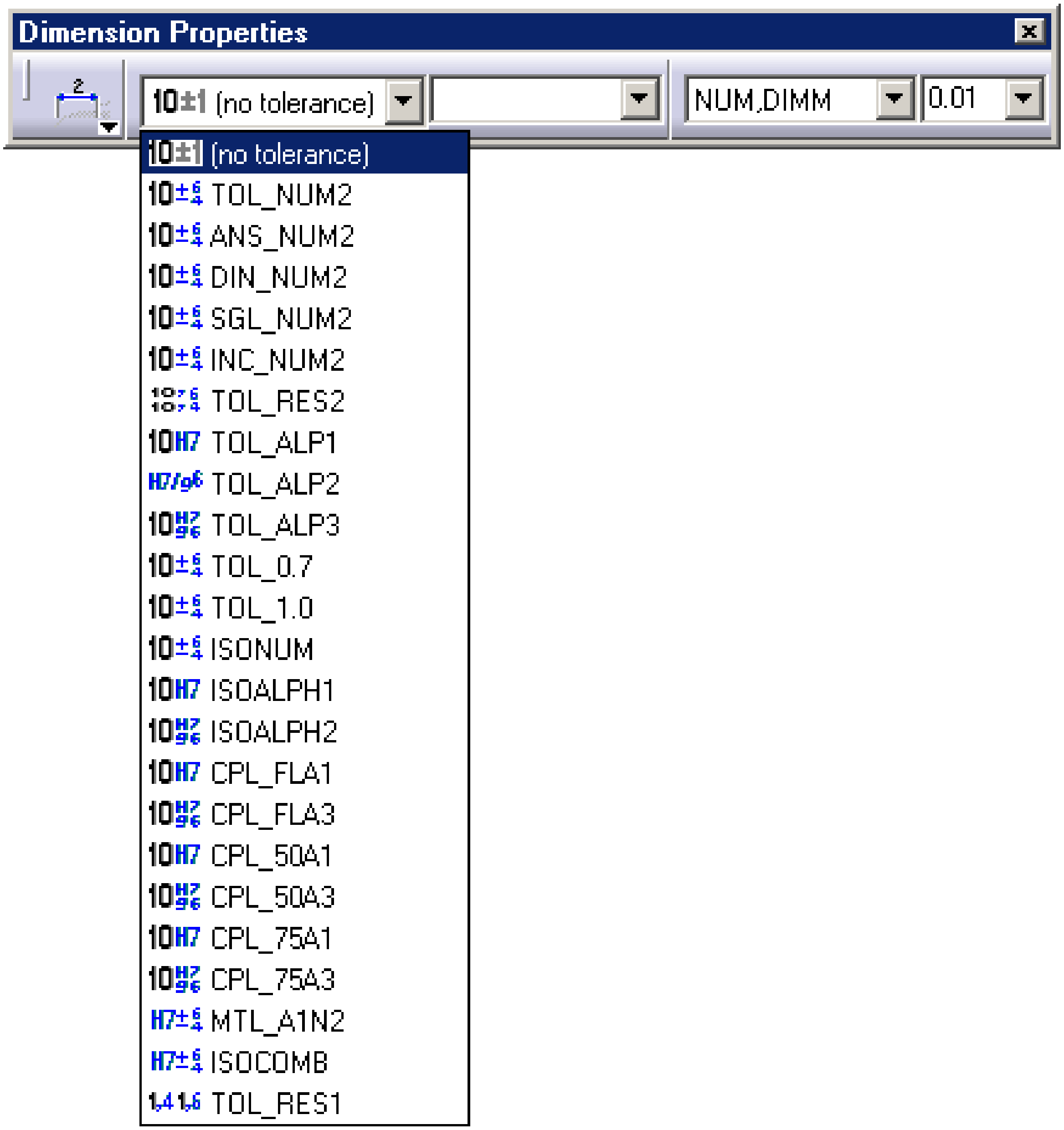
For more information on standards and their administration, refer to the User Tasks > Administration Tasks chapter in the **Interactive Drafting User's Guide**.

Dimension Tolerance Display

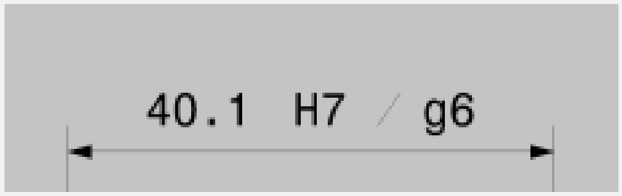
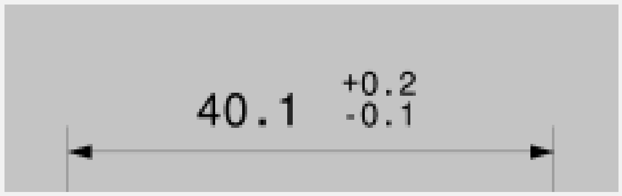
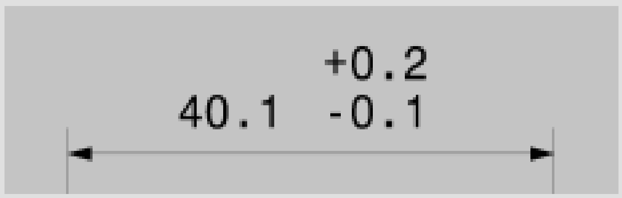
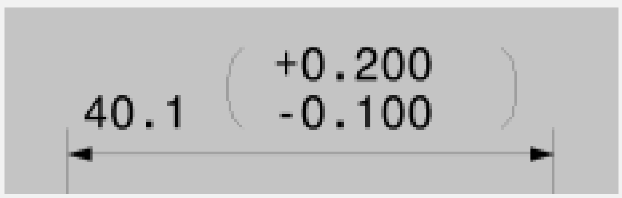
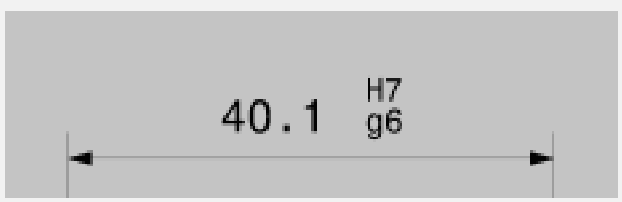
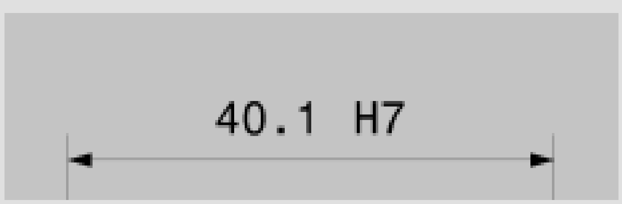
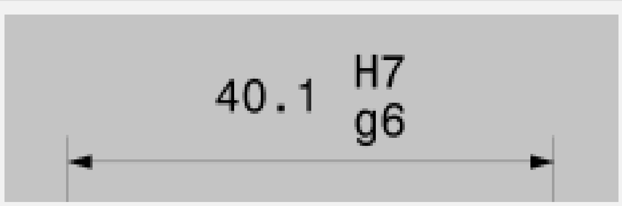
Dimension Numerical Display

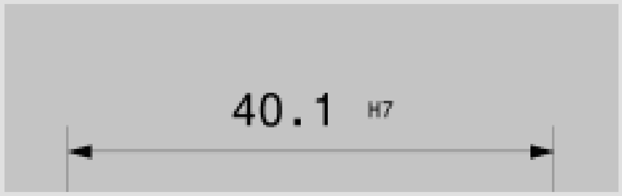
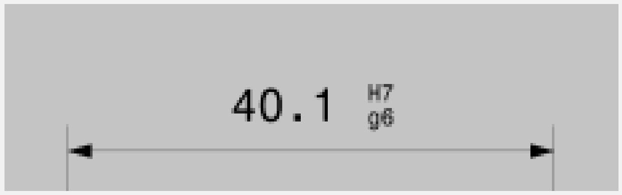
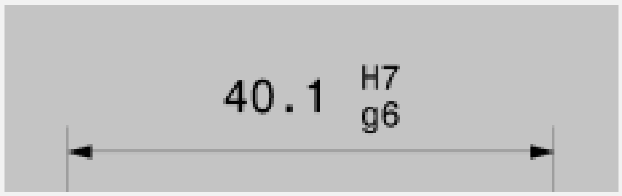
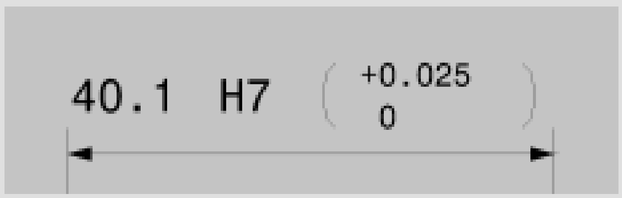
Dimension Tolerance Display

Dimension tolerance display formats are listed below. These formats are available in the **Properties** dialog box, **Tolerance** tab.



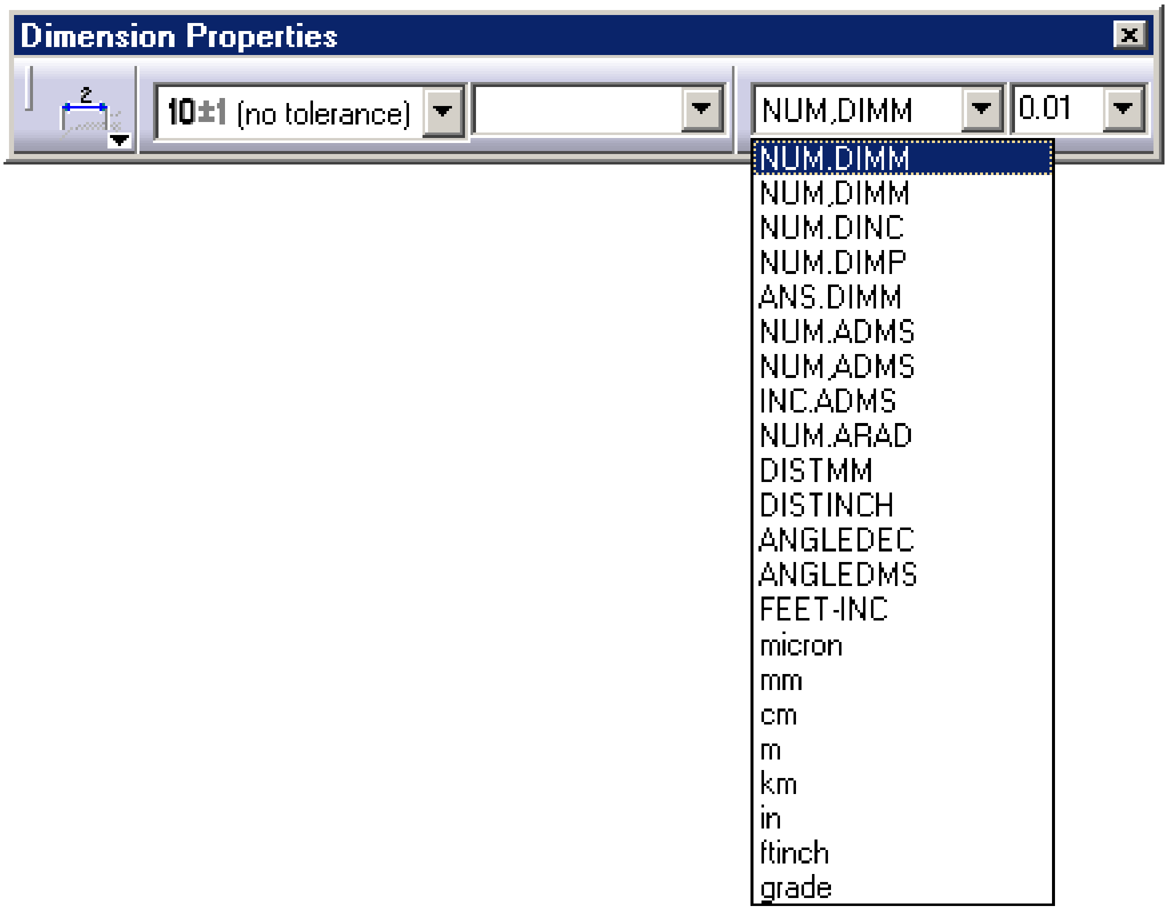
Name	Display	Description
(no tolerance)	40.1	No tolerance displayed.
TOL_NUM2	40.1 +0.2 -0.1	Numerical superimposed (small).
ANS_NUM2	40.1 +0.200 -0.100	Numerical superimposed with trailing zeros (large).
DIN_NUM2	40.1 +0.2 -0.1	Numerical superimposed (small).
SGL_NUM2	40.1 (+0.200 -0.100)	Numerical superimposed with trailing zeros between parentheses (small).
INC_NUM2	40.1 +0.2 -0.1	Numerical superimposed (large).
TOL_RES2	40.3 40	Numerical resolved, superimposed.
TOL_ALP1	40.1 H7	Alphanumerical single value (large).

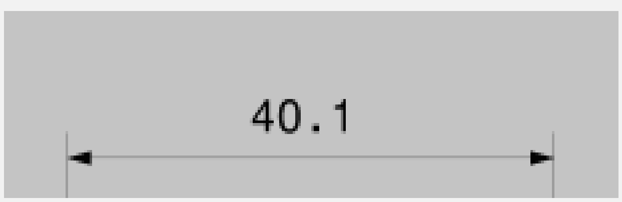
H7/g6 TOL_ALP2		Alphanumerical double value side-by-side (large).
10 H7/g6 TOL_ALP3		Alphanumerical double value superimposed (small).
10 ± TOL_0.7		Numerical superimposed (small).
10 ± TOL_1.0		Numerical superimposed (large).
10 ± ISONUM		Numerical superimposed with trailing zeros between parentheses (large).
10H7 ISOALPH1		Alphanumerical single value (large).
10 H7/g6 ISOALPH2		Alphanumerical double value superimposed (small).
10H7 CPL_FLA1		Alphanumerical single value (large).
10 H7/g6 CPL_FLA3		Alphanumerical single value (large).

10H7 CPL_50A1		Alphanumerical single value (small).
10H7/g6 CPL_50A3		Alphanumerical double value superimposed (small).
10H7 CPL_75A1		Alphanumerical single value (medium).
10H7/g6 CPL_75A3		Alphanumerical double value superimposed (medium).
H7± MTL_A1N2		Alphanumerical single value (large) and numerical superimposed between parentheses (small).
H7± ISOCOMB		Alphanumerical single value (large) and numerical superimposed between parentheses (large).
1,4 1,6 TOL_RES1		Numerical resolved, side-by-side.

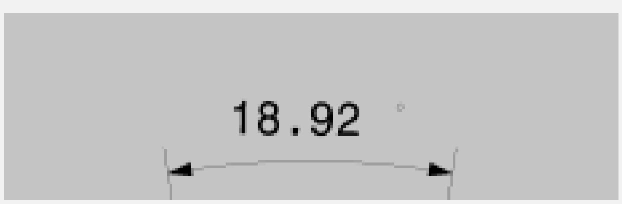
Dimension Numerical Display

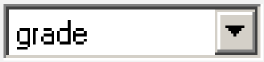
Dimension numerical display formats are listed below. These formats are available in the **Properties** dialog box, **Value** tab.



Name	Display	Description
NUM.DIMM		Millimeters with dot.
NUM,DIMM		Millimeters with comma.
NUM.DINC		Inches with trailing zeros.

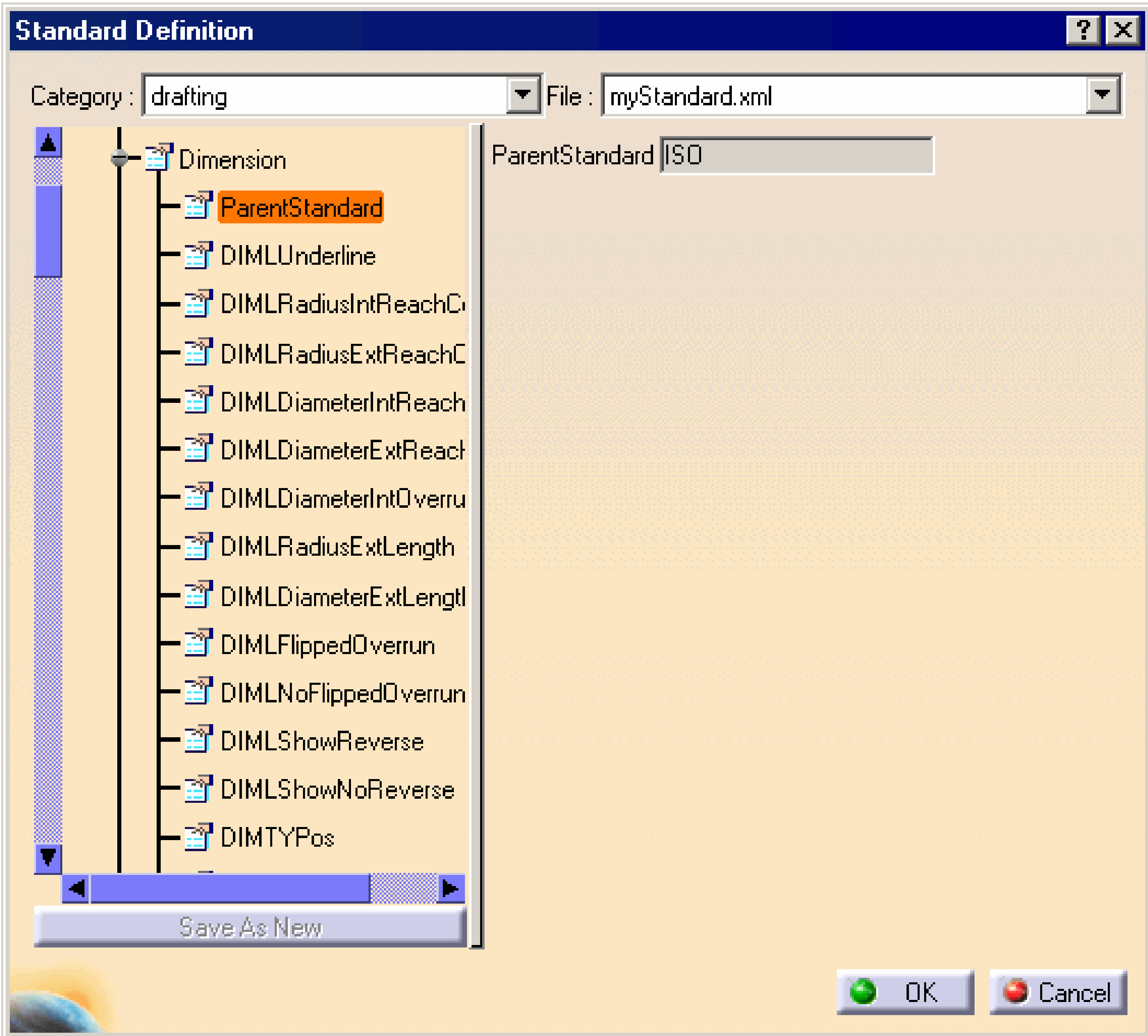
NUM.DIMP	1.579 "	Inches with unit display.
ANS.DIMM	40.100	Millimeters with trailing zeros.
DISTMM	40.1	Millimeters with dot.
DISTINCH	1.579 "	Inches with unit display
FEET-INC	1.579 "	Feet and Inches with unit display
micron	40100	Microns.
mm	40.1	Millimeters.
cm	4.01	Centimeters.
m	.0401	Meters.

km		Kilometers.
in		Inches.
ftinch		Feet and Inches.
NUM.ADMS		Degrees, Minutes, Seconds with dot.
NUM.ADMS		Degrees, Minutes, Seconds with comma.
INC.ADMS		Degrees, Minutes, Seconds with dot and
NUM.ARAD		Radians.
ANGLEDEC		Degrees with decimal format.
ANGLEDMS		Degrees, Minutes, Seconds with dot.

		Grades.

Semantic Support

The rules which define the validity of dimensioning and tolerancing specifications are defined by the parent standard parameter of the standard used:



- For ANSI parent standard, American Standards are used.
- For ISO parent standard, International Standards are used.
- For JIS parent standard, International Standards are used.

Extension to Standards

- For ANSI and ISO standards:
  - Oriented Dimension: capability to create semantic toleranced dimensions on non-Features of Size (between planar face or cylindrical face or spherical face or axis or point or plane or circle, and planar face or cylindrical face or spherical face or axis or point or plane or circle).
  - Capability to create semantic dimensioning and tolerancing on constructed geometry.
- For ISO standards only:
  - Capability to create datum on non-canonical surface (complex surface or prismatic surface).
  - Capability to create shifted profile tolerances.
  - Capability to use tangent plane modifier.

Normative References

The table below provides a list of normative references for:

American Standards

ANSI B4.2-1978	Preferred Metric Limits and Fits
ASME Y14.5M-1994	Dimensioning and tolerancing - Revision of ANSI Y14.5M-1982
ASME	A standard is being developed on DIGITAL MODELING. This project will cover the representation of dimensioning and tolerancing annotations in the 3D space.
ASME Y14.41M-2003	Digital product Definition. Data Practices.

International Standards

ISO	Standards are being developed for the specification of dimensioning and tolerancing annotation in the 3D space.
ISO 406-1987	Technical drawings -- Tolerancing of linear and angular dimensions
ISO 286	ISO system of limits and fits
ISO 1101-1983	Geometrical tolerancing. Tolerancing of form, orientation, location and run-out
ISO/FDIS 1101-1998	Geometrical product specification (GPS). Geometrical Tolerancing - Generalities, definitions, symbols, indication on drawings. Final Draft International Standard (FDIS). Revision of ISO 1101-1983. Note: The new ISO 1101 standard will be published in october 2000 and will replace ISO 1101-1983.
ISO 1660-1987	Technical drawings. Dimensioning and tolerancing of profiles
ISO 2692-1988	Technical drawings. Geometrical tolerancing. Maximum material principle
ISO 2692 Amd 1-1992	Technical drawings. Geometrical tolerancing. Maximum material principle- Amendment 1: Least material requirement
ISO 5458-1998	Geometrical Product Specifications (GPS). Geometrical tolerancing - Positional tolerancing

ISO 5459-1981	Technical drawings. Geometrical tolerancing. Datums and datum systems for geometrical tolerances Under revision
ISO/DIS 5459-1	Geometrical product specification (GPS). Datums for geometrical tolerancing. Part 1: General terms and definitions, 1998-10-15, Committee Draft. (Revision of ISO 5459-1981)Draft International Standard (DIS)
ISO/DIS 5459-2	Geometrical product specification (GPS). Datums for geometrical tolerancing. Part 2: Datums and datum-systems; explanations and indication 1998-10-15, Committee Draft. (Revision of ISO 5459-1981) Draft International Standard (DIS)
ISO 8015-1985	Technical drawings. Fundamental tolerancing principle
ISO 10578-1992	Technical drawings. Tolerancing of orientation and location. Projected tolerance zone
ISO 10579-1993	Technical drawings. Dimensioning and Tolerancing. Non-rigid parts
ISO 14660-1	Geometrical product specification (GPS). Geometric features. Part 1: General terms and definitions
ISO 14660-2	Geometrical product specification (GPS). Geometric features. Part 2: Extracted median line of a cylinder and a cone; extracted median surface; local size of an extracted feature
ISO/TS 17450-1999	Geometrical product specification (GPS). Model for geometric specification and verification.

Principles and Fundamental Rules for Geometrical Tolerancing

All the dimension, form, orientation and position specifications (either on a geometric feature or on a geometric feature group of a part) are independent (see ISO 8015).

The dimensions of the features and their geometry are independent, regarding the form, the orientation and the position (see ISO 8015).

Each dimension shall have a tolerance, except for the dimensions specially identified as reference, maximum, minimum, or stock (commercial stack size). The tolerance may be applied directly to the dimension (or indirectly in case of basic dimensions), indicated by a general note, or located in a supplementary block of the drawing format, see ANSI Y14.1, ASME Y14.5M-1994).

Dimensioning and tolerancing shall be complete so there is full understanding of the characteristics of each feature. Neither scaling (measuring the size of a feature directly from an engineering drawing) nor assumption of a distance or size is permitted, except as follows: non-dimensioned drawings, such as loft, printed wiring, templates, and master layouts prepared an stable material, are excluded provided the necessary control dimensions are specified, (ASME Y14.5M-1994).

Geometrical Tolerancing

Geometrical Tolerance is the general term applied to the category of tolerances used to control form, profile, orientation, location and runout, (ASME Y14.5M-1994).

True Geometrical Counterpart represents the theoretically perfect boundary (virtual condition or actual mating envelope) or best-fit (tangent) plane of a specified datum feature, (ASME Y14.5M-1994).

The geometrical tolerancing is divided into four types (by both ISO and ASME/ANSI):

- Form tolerances
- Orientation tolerances
- Location or position tolerances
- Runout tolerances

Geometrical tolerance objective is the boundary of spaces in which the toleranced feature has to be located with regards to the specified datum or datum system, to meet the tolerance specification. These particular tolerances allow to limit either actual feature defects or fitted features, with respect to nominal characteristics, and without considering the features' dimensions.

Geometrical tolerancing is based on three feature types:

> Tolerance features: a toleranced feature is an actual feature (point, line, surface, except for projected tolerance), or a fitted or a constructed feature. If the toleranced feature corresponds to a group, then each component of the group has the same nature, and the toleranced feature is a toleranced feature group.
>
> Tolerance zone: a tolerance zone is a space (either surface or volume), bounded by one or several nominal features. That space defines the toleranced feature location in order to satisfy the tolerance specification, (see ISO 1101). When the geometrical tolerance applies on a feature group, then one tolerance zone is linked to one feature.
>
> Datum elements or datum systems.

Even if the Geometrical Tolerance creation is accomplished without any semantic links, we recommend you to specify datum elements and then declare your geometrical tolerancing with references.

Symbols for Geometrical Tolerances

The list of available symbols for geometrical tolerances.

Tolerances	Characteristics	Symbol	Datum needed
Form	Straightness	⏤	No
	Flatness	⏥	No
	Roundness	○	No
	Cylindricity	⌭	No
	Profile any line	⌒	No
	Profile any surface	⌓	No
Orientation	Parallelism	//	Yes
	Perpendicularity	⊥	Yes
	Angularity	∠	Yes
	Profile any line	⌒	Yes
	Profile any surface	⌓	Yes
Location	Position	⌖	Yes or no
	Concentricity (for center points)	◎	Yes
	Coaxiality (for axis)	◎	Yes
	Symmetry	⌯	Yes
	Profile any line	⌒	Yes
	Profile any surface	⌓	Yes
Run-out	Circular run-out	↗	Yes
	Total run-out	⌰	Yes

Symbols for Modifiers

The list of available symbols for modifiers.

Least Material Condition	Symbols
Maximum Material Condition (MMC)	M
Least Material Condition (LMC)	L
Tangent plane	T
Regardless of Feature of Size (RFS)	S
Free State	F
Projected Tolerance zone	P

Datum Principles

Datum Elements and Datum Systems

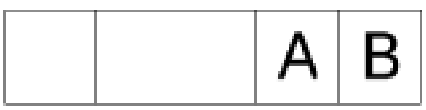
Datum elements and datum systems are only specified in case of geometrical tolerancing (not on dimensional tolerancing and except on form tolerancing). You can specify simple datum elements, common datum elements, datum targets or specified datum systems.

The toleranced feature is positioned relatively with the tolerance zone, and this set is positioned relatively with the datum or the datum system. These related positions will be specified using basic dimensions and they are displayed or not.

For more information about datum constitution and specification, see ISO 5459, ASME Y15.5M and ISO 1101.

Datum System Composition

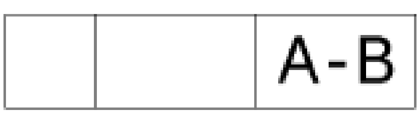
When the identifiers are specified separately in each frame of the tolerance frame, the datum elements represent a datum system. A hierarchy is established between the datum elements. Reference A is the primary datum and reference B is the secondary datum. The datum system fitting would be performed first on datum A, then on datum B, with respect to A. By the way, a tertiary datum can also be specified.



The datum elements have to be fitted successively (following their specification order) with respect to the hierarchy for the measurement computations.

When only one identifier is specified in the tolerance frame, the datum is a single datum.

When two identifiers separated by a dash are specified in the tolerance frame, the datum is a common datum. The two datum elements are to be considered simultaneously. The datum system fitting would be performed in the meantime on datum A and on datum B (both datum elements have to be fitted simultaneously for measurement computations).



These symbols represent two specifications in the meantime: A | B and B | A.

Datum writing rules

A capital letter is used to identify the datum element in the tolerance frame. The datum triangle may be filled or non-filled.

When the datum triangle is placed on the outline of the element or on its extension line, the datum element represents the surface itself or the 2D representation of the surface, which is a line.

When the datum triangle is placed in the alignment of the dimension line, the datum element represents the median element (usually an axis or a median plane).

When the datum triangle points directly on a median element, the datum element represents either the median element itself (usually an axis or a median plane) or the resulting median element of the collection of the considered elements.

The tolerance frame may also be related to the datum element using a leader line.

When the tolerance frame is related to one datum element through a leader line, the datum identifier may be omitted in the tolerance frame.

Concepts

3D Annotations and Annotation Planes

3D annotations are dimensions, tolerances, notes, text or symbols displayed in 3D according to the same type of 2D annotations defined by standards (ISO, ASME, ANSI, JIS, DIN...).

A 3D annotation is always linked to a user surface or a group of surfaces.

A 3D annotation is displayed in 3D following the orientation of a particular plane called Annotation Plane.
Two types of 3D annotation are available: Semantic 3D Annotation and Non-semantic 3D Annotation.

Non-semantic 3D Annotations

3D annotations that are not defined in ISO or ASME/ANSI standards:

- Text
- Flagnote
- Note Object Attribute (NOA)

3D annotations where only their graphical attributes are take into account.
There is no control on attribute values.
There is no control of consistency regarding the geometry on which it is applied and the other annotations.
There is no control of the syntax regarding the standard in use.

- Roughness
- Datum
- Datum target
- Geometrical tolerance
- Dimension

Semantic 3D Annotations

3D annotations on which attribute values, consistency with geometry and syntax are controlled.
The annotation validity is warranted along the life cycle application to be re-used and well understood by other applications like tolerance analysis, inspection, manufacturing, assembly process...

- Datum
- Datum target
- Datum reference frame
- Geometrical tolerance
- Dimension

User Surface

Depending the way the parts or products have been designed the surface the user wants to consider can be made of one or several geometrical elements.

Group of Surfaces

A group of surfaces is a set of user surface or group of surfaces. It is used to modelized Tab/Slot, Profile and Pattern features as defined in standards.

Annotation Planes

Annotation planes are the 3D equivalent of 2D views. They contain the 3D annotation or define their orientation.
Three types of Annotation planes are available: Projection View, Section View and Section Cut.

- In Projection View Annotation Plane, 3D annotations may be:
  - Located in planes parallel to this annotation plane and in the background and foreground spaces bounded by this annotation plane.
  - Related to the geometry finding an intersection with this annotation plane.
  - Lying on/belonging to this annotation plane.
- In Section View Annotation Plane, 3D annotations may be:
  - Located in planes parallel to this annotation plane and in the background space bounded by this annotation plane.
  - Related to the geometry finding an intersection with this annotation plane.
  - Lying on/belonging to this annotation plane.
- In Section Cut Annotation Plane, 3D annotations may be:
  - Only related to the geometry finding an intersection with this annotation plane.
  - Only lying on/belonging to this annotation plane.

Non-semantic and Semantic Usage

The goal of the Functional Tolerancing & Annotation product is to fully cover the ISO or ASME/ANSI standards semantic and syntax definitions. Non-semantic 3D annotation can be used in case of Functional Tolerancing & Annotation lacking capabilities or company usage of symbols and syntax that are not covered by standards.

The following Semantic 3D Annotations are available:

- Datum
- Datum target
- Toleranced dimensions applied to Features of Size:
  - Circle
  - Sphere
  - Cylinder
  - Tab/Slot
  - Angular Tab/Slot
  - Elongated Hole
  - Rectangular Holes
- Toleranced dimensions applied to non-Features of Size:
  - Oriented dimension.

Note Object Attribute

Purpose

The Note Object Attribute purpose feature is to provide a way to define user types of 3D annotations and to define their 3D display.

Usage

The recommended usage for such Note Object Attribute feature is the following:

- The administrator defines all the company types of Note Object Attribute and stores them in a dedicated catalog:
  - Finish Data (paint, special processes)
  - Surface treatment types
  - Grain Direction
  - Key Characteristics
  - Electrical Bonding Location and Specification
  - Part Marking Location and Specification
  - Process Data (Specification for processes)
  - ...
- The administrator turns off the Note Object Attribute creation setting (see Tolerancing setting) to lock it in order to forbid the creation of new Note Object Attribute types by users.
- Users instantiate from the catalog the types of Note Object Attribute they want to specify.

Filtering

You can filter by type of Note Object Attribute the 3D annotations by using the filter command.

Properties

Text Graphical Properties: apply graphical properties to a text.

Text Properties Toolbar: apply graphical properties to a text

Semantic Numerical Display Properties: set semantic numerical display.

Text Graphical Properties

Take into account according to the options set in the **Style** and **Text Properties** toolbars and/or using the **Copy Object Format** command and/or **User Default** command.
See Managing Graphical Properties.

Object	Properties		Application Default Properties	User Default Properties	Only User Default Properties
Font	Font	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Style	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Size	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	UnderLine	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Color	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Ratio	Yes	Application Default	User Default	User Default
	Strikethrough	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Superscript	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Subscript	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Overline	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
Text	Frame	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Color	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Thickness	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Line Type	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Anchor point	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Anchor Line	Yes	Application Default	Application Default	User Default

	Line Spacing	Yes	Application Default	User Default	User Default
	Line Spacing Mode	Yes	Application Default	User Default	User Default
	Justification	Yes	Text Properties Toolbar	User Default	User Default
	Word wrap	Yes	Application Default	User Default	User Default
	Reference	Yes	Application Default	User Default	User Default
	Orientation	Yes	Application Default	User Default	User Default
	Angle	Yes	Application Default	User Default	User Default
	Mirroring	Yes	Application Default	User Default	User Default
	Auto flip	Yes	Application Default	User Default	User Default
Graphic	Color	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Linetype	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Thickness	Yes	Text Properties Toolbar	Text Properties Toolbar	User Default
	Pickable	No	Application Default	Application Default	Application Default

Text Properties Toolbar

Set the text appearance in a text annotation from the options available in the Text Properties toolbar.
Note that the operating mode described here is valid for datum elements, datum targets and geometrical tolerances too.

These properties are:

Font Name: changes the font of text

Font Size: changes the font size of text

Bold: changes the weight of text, toggles between normal and heavy (bold).

Italic: changes the angle of text, toggles between normal and slanted (italic)

Underline: adds a line under the text.

Strikethrough: adds a line through the middle of the text.

Overline: adds a line above the text.

Superscript: raises the text above the normal text line.

Subscript: lowers the text below the normal text line.

Left Justify: aligns multiple lines of text along the left edge of the text frame.

Center Justify: aligns multiple lines of text in the center of the text frame.

Right Justify: aligns multiple lines of text along the right edge of the text frame

Anchor Point: changes the position of the point that connects the text to the drawing or to an element.
There are nine possibilities:

Along the top of the text: left, center, or right.

Along the vertical center of text: left, center or right.

Along the bottom of the text: left, center, or right.

Frame: draws a single-line frame around the text. The frame size can be variable, or fixed.

Insert Symbol: inserts several symbol types including geometrical tolerancing ones especially in the text editor.

Font Name

Select the font.

Font Size

Select the size.

Bold

Select or unselect the icon.

Italic

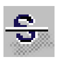
Select or unselect the icon.

Underline

Select the icon.

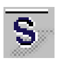
Strikethrough

Select the icon.



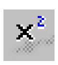
Overline

Select the icon.



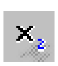
Superscript

Select the icon.



Subscript

Select the icon.



Left Justify

Select the icon.

Center Justify

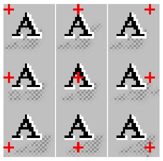
Select the icon.

Right Justify

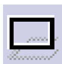
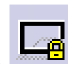
Select the icon.

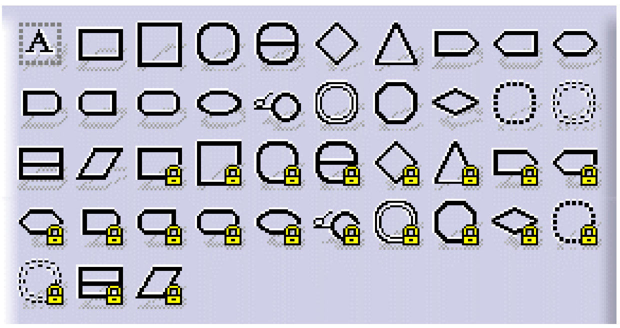
Anchor Point

Select the icon.



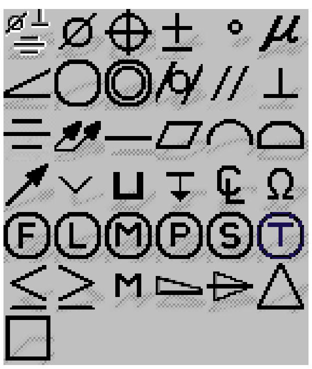
Frame

Select the icon. You can choose to create each frame with either a variable or a fixed size. For a rectangular frame, for example, the icon  represents the variable-size frame, and the icon  (with the padlock) represents the fixed-size frame.



Insert Symbol

Select the symbol.



Semantic Numerical Display Properties

Set the numerical display of any semantic geometrical tolerance containing numerical values.
These properties are available for an annotation set or any semantic geometrical tolerance.
Default settings are defined from the selected standard and the annotation set properties.

They are applied when creating a semantic geometrical tolerance,
right-click the annotation set then **Properties -> Tolerancing & Annotations** tab.

These settings can be individually modified for each semantic geometrical tolerance,
right-click the semantic geometrical tolerance then **Properties -> 3D Annotation** tab.

There are four properties:

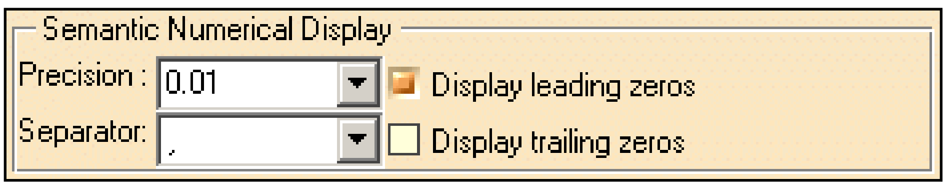
Precision: defines the number of digit displayed after the separator.

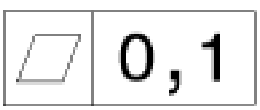
Separator: defines the symbol used as numerical separator.

Display leading zeros: displays or not the "0" before the separator when value is less than 1.

Display trailing zeros: displays or not "0" to complete the number of digit displayed after the separator according to the precision.

Default settings according to the ISO standard

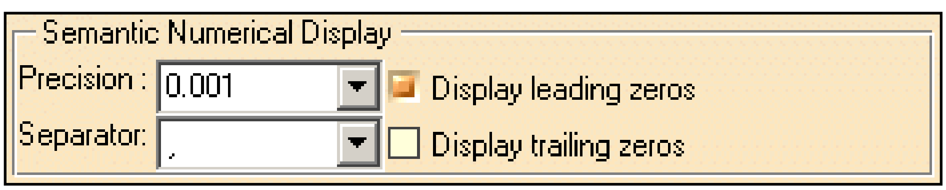




In the following examples we start from the ISO default settings.

Precision

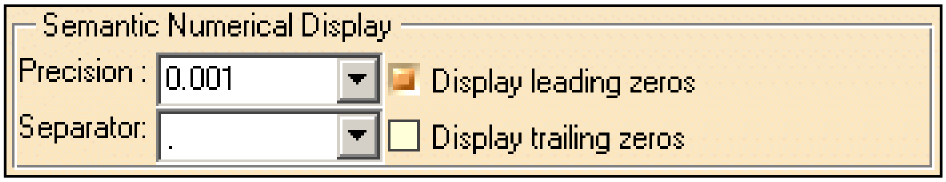
Set the precision to **0.001**

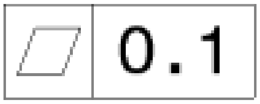


According to the unchecked **Display trailing zeros** setting only the effective digit after the separator are displayed.

Separator

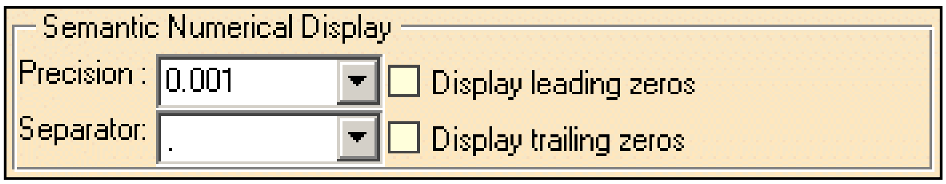
Change the separator symbol for the **.** (dot)

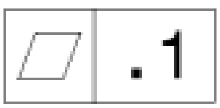




Display leading zeros

Uncheck the **Display leading zeros** setting.

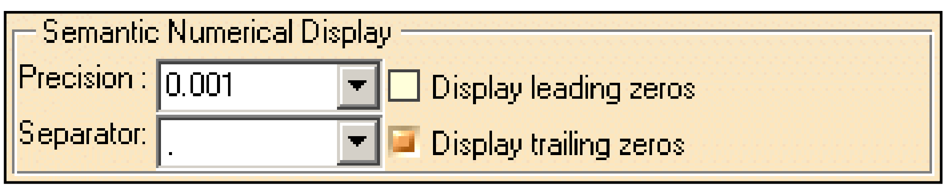


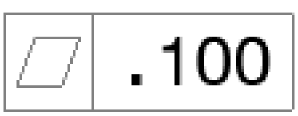


According to the unchecked setting the "0" is removed before the separator.

Display trailing zeros

Check the **Display trailing zeros** setting.

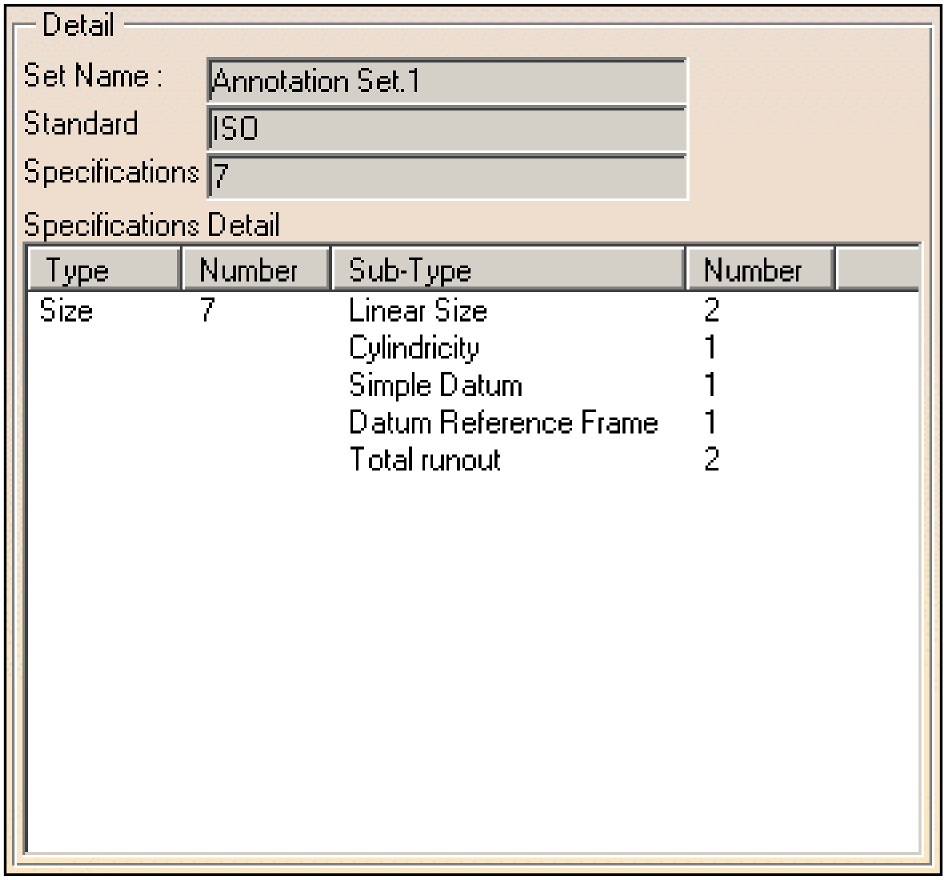




According to the setting "0" are added to complete the precision format.

Annotation Set Detail Properties

Display the annotation set detail properties: select **Properties -> Tolerancing & Annotations** tab



There are four properties:

Set Name: displays the annotation set name.

Standard: displays the annotation set standard.

Specifications: displays the number of annotations contain in the annotation set.

Specifications Detail: displays and quantifies the annotations types and sub-types.

Types	Sub-Types
Datum	Simple Datum Datum System Datum Target Datum Reference Frame
Form	Straightness Axis Straightness Flatness Circularity Cylindricity Profile of any line Profile of any surface Pattern location
Size	Linear Size Angular Size Chamfer Dimension Basic dimension Second Linear Size
Position Orientation and Runout	Parallelism Perpendicularity Angularity Position Concentricity Symmetry Profile of any line with Datum Reference Frame Profile of any surface with Datum Reference Frame Total runout Circular runout
Non Semantic	Text Flag Note Note Object Attribute Datum Geometrical Tolerance Datum Target Weld Dimension
Roughness	

Dimension Units

Default Dimension Unit When Creating an Annotation Set

The default dimension unit is defined by the standard default numerical formats for length and angle, see Tolerancing option for standard selection:

1. Select **Tools -> Standards**...

2. In the **Standards Definition** dialog box, select **Drafting** category and the **Standard_Name.xml** file according to the selected standard.

3. Select **Standard -> Standard_Name -> General -> DefaultNumericalFormatLength** to see the default length format for the selected standard

   and/or

   select **Standard -> Standard_Name -> General -> DefaultNumericalFormatAngle** to see the default angle format for the selected standard.

If no standard has been defined, the dimension unit is set by the **Length** unit in the **Tools -> Options -> General -> Parameters and Measure**, **Units** tab.

Modifying the Default Dimensions Unit

To modify default dimensions unit you can:

- Set the **Numerical Display Description** combo unit in the Dimension Properties Toolbar.
- Set the **Length** (or **Angle**) unit in the **Tools -> Options -> General -> Parameters and Measure**, **Units** tab, in this case the **Numerical Display Description** combo is updated according to the **Length** (or **Angle**) unit.

Statistic Laws

Statistic laws are used to characterize deviation annotations. interpret annotation tolerance in a deviation annotation. These deviation annotations can be directly specified by the user or generated from a annotation's tolerance translation.

> For a statistical analysis, an annotation tolerance is interpreted as random variable or variate.
>
> For a determinist analysis, the mean of the annotation tolerance is used as determinist value.

Normal Law: describe the Normal probability law equations.

Uniform Law: describe the Uniform probability law equations.

Constant Law: describe the Constant probability law equations.

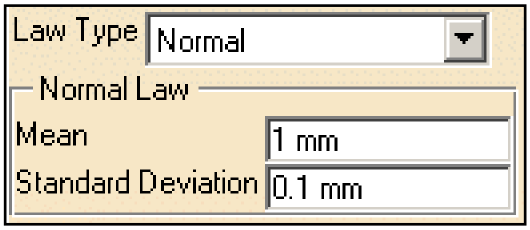
Pearson Law: describe the Pearson probability law equations.

Poisson Law: describe the Poisson probability law equations.

Snedecor Law: describe the Snedecor probability law equations.

Normal Law

The Normal law is parameterized by a mean μ (unit: millimeter) and a standard deviation σ (unit: millimeter). Another name for the Normal law is Gaussian law.



Let's take X a random variable following the Normal law, then:

X is distributed according to the following density of probability:

$$P(x) = \frac{1}{\sigma\sqrt{2\pi}} e^{-(x-\mu)^2/2\sigma^2}$$

With mean:

$$E(X) = \mu$$

With variance:

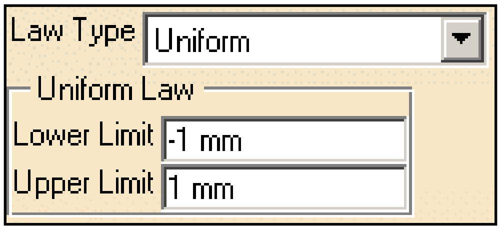
$$Var(X) = \sigma^2$$

With standard deviation:

$$SD(X) = \sigma$$

Uniform Law

The Uniform law is parameterized by a lower limit a (unit: millimeter) and a upper limit b (unit: millimeter). Another name for the Uniform law is Rectangular law.



Let's take X a random variable following the Uniform law, then:

X is distributed according to the following density of probability, where:

$$P(x) = \begin{cases} \frac{1}{b-a} \text{ for } a \le x \le b \\ 0 \quad \text{for } x < a,\, x > b \end{cases}$$

With mean:

$$E(X) = \frac{(b-a)}{2}$$

With variance:

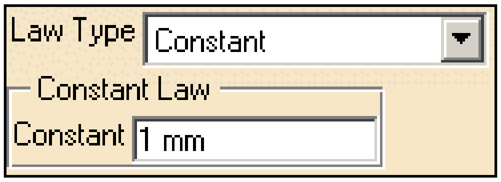
$$Var(X) = \frac{(b-a)^2}{12}$$

With standard deviation:

$$SD(X) = \frac{(b-a)}{2\sqrt{3}}$$

Constant Law

The Constant law is parameterized by a constant c (unit: millimeter).



Let's take X a random variable following the Constant law, then:

X is always equals to the constant:

$$X = c$$

With mean:

$$E(X) = c$$

With variance:

$$Var(X) = 0$$

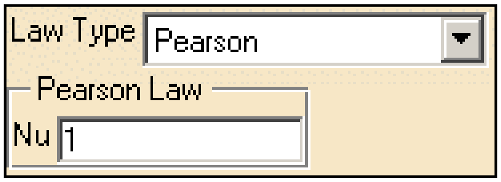
With standard deviation:

$$SD(X) = 0$$

Pearson Law

The Pearson law is parameterized by ν (no unit). Another name for the Pearson law is Chi-squared law.
If ν random variables Y_i $(i = 1,\ldots,\nu)$ follows the Normal law with mean 0 and variance 1, then:

$$\chi^2 \equiv \sum_{i=1}^{\nu} Y_i^2$$



Let's take χ^2 a random variable following the Pearson law, then:

χ^2 is distributed according to the following density of probability, where:

$$P(x) = \frac{x^{\nu/2-1} e^{-\frac{x}{2}}}{\Gamma(\frac{\nu}{2}) 2^{\frac{\nu}{2}}}$$

And where $\Gamma(x)$ is a Gamma function.

With mean:

$$E(\chi^2) = \nu$$

With variance:

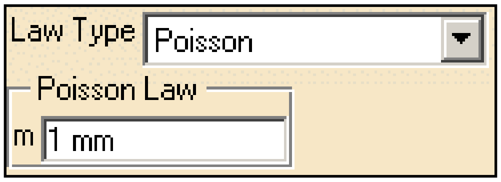
$$Var(\chi^2) = 2\nu$$

With standard deviation:

$$SD(\chi^2) = \sqrt{2\nu}$$

Poisson Law

The Poisson law is parameterized by m (unit: millimeter).



Let's take X a random variable following the Poisson law, then:

X is distributed according to the following density of probability, where:

$$P(x) = \frac{m^x}{x!} e^{-m}$$

With mean:

$$E(X) = m$$

With variance:

$$Var(X) = m$$

With standard deviation:

$$SD(X) = \sqrt{m}$$

Snedecor Law

The Snedecor law is parameterized by two non-dimensional numbers m and n.

Let's take F a random variable following the Snedecor law of parameters m and n, then F can be expressed in terms of two random variables X, Y following Pearson law respectively of parameter m, n as:

$$F \equiv \frac{X/m}{Y/n}$$

F is distributed according to the following density of probability, where:

$$P(x) = \frac{\Gamma\left(\frac{m+n}{2}\right)\left(\frac{m}{n}\right)^{m/2} x^{(m-2)/2}}{\Gamma\left(\frac{m}{2}\right)\Gamma\left(\frac{n}{2}\right)\left(1+\frac{m}{n}x\right)^{(m+n)/2}}$$

With mean:

$$E(F) = \frac{n}{(n-2)}$$

With variance:

$$SD(F) = \sqrt{\frac{2n^2(m+n-2)}{m(n-2)^2(n-4)}}$$

With standard deviation:

$$Var(F) = \frac{2n^2(m+n-2)}{m(n-2)^2(n-4)}$$

Version 4 Functional Dimensioning &Tolerancing Data Migration

There are a few things you need to know before you start migrating Version 4 Functional Dimensioning & Tolerancing data to Version 5:

- Migration is performed using a Copy-Paste operation. You should paste the Version 4 annotation set at the level of the product containing the geometry.
- Version 4 tolerancing data is migrated for parts only, not for assemblies.
- To migrate Version 4 Functional Dimensioning & Tolerancing data to Version 5, a Version 5 Functional Tolerancing & Annotation license is required.
- Not all Version 4 data can be migrated to Version 5, and sometimes, migrated data can be invalid.
- If an annotation set already exists in Version 5, Version 4 data will be pasted to this existing annotation set. In this case, make sure that the parent standard is the same in Version 4 as in Version 5, otherwise the copy will not be performed.
- If there is no existing annotation set in Version 5, a new annotation set will be created. In this case, the Version 4 standard will be used.
- In the case of several Version 4 annotation sets within a Version 4 model, you will either need to merge FD&T data within a single Version 5 annotation set, or split the Version 4 model in several Version 4 models, containing each a single annotation set.
- Version 4 Functional Dimensioning & Tolerancing windows are migrated as Version 5 captures.

Migrated data

The table below summarizes the data which is migrated from Version 4 to Version 5.

Version 4 Data	Version 5 Migration Result
Annotation plane	View/Annotation plane
Tolerancing feature (TTRS), except re-specified features	Group of surfaces, User surface (TTRS)
Re-specified tolerancing feature (TTRS)	Redefined User surface (TTRS)
Surface restriction	Surface restriction
Annotation	Annotation
Geometrical tolerance	Geometrical tolerance
Default tolerance	Default tolerance
Dimensions, except chamfer and thread diameter	Dimensions
TolNOA	NOA
View	Capture

Non-migrated data

The list below summarizes the data which is not migrated from Version 4 to Version 5.

- Basic dimensions based on curvilinear curves
- Dimensions based on re-specified tolerancing feature (TTRS)
- Flag note URLs
- Note Object Attributes
- Thread dimensions
- Chamfer dimensions

Annotations and Cache System

The cgr documents used when you are working with the **Cache System** (in Visualization mode) and the cgr documents you can generate when saving as cgr type. can include the 3D annotations representation (controlled trough a dedicated option).

The 3D annotations that are included in cgr documents can be:

- Displayed in 3D using the **List Annotation Set Switch On/Switch Off** command or the **Annotation Set Switch On/Switch Off** contextual command on an Annotation Set, see Disabling/Enabling Annotations,
- Queried (cross highlight between geometry and annotations) by using the **Select** command (when the **3D-Annotation-Query Switch On/Switch Off** option command is active), see Querying 3D Annotations,
- Filtered by user the **Filter** command , see Filtering Annotations,

as any 3D annotation of CATProduct documents or CATProcess documents or CATPart documents. The graphic properties (color and line thickness) of 3D annotations displayed in visu mode can be temporarily modified, meaning they will be lost when switching to design mode or closing the document. All the other modifications (edit, move, ...) require switching in design mode.

To know more about:

- The Cache System option, see Cache Management.
- The cgr option, see Cgr Management.

Glossary

A C D E F G I N R S T

A

Associated Derived Feature Center point, axis or median plane derived from one or more associated integral features (ISO 14660).

Associated Integral Feature Feature of perfect form associated to the extracted integral feature in accordance with specified conventions (ISO 14660).

C

Common Datum Datum based on two or more surfaces considered simultaneously (ISO/CD 5459-1:1998).

D

Datum A theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established, (ASME Y14.5M-1994).

Situation feature used to define the position and/or orientation of a tolerance zone or to define the position and/or the orientation of the virtual conditions (in the case of the complementary requirements, e.g maximum material requirement), (ISO/CD 5459-1:1998(E)).

Datum System Ordered list of at least two datums and at most three datums, which may be single or common datums, (ISO/CD 5459-1:1998(E)).

Note: A datum-system is used to define the position and/or orientation of a tolerance zone or to define the position and/or the orientation of the virtual condition (in the case of complementary requirements, e.g maximum material requirement).

(Datum) Target

A specified point, line, or area on a part used to establish a datum, (ASME Y14.5M-1994).

Portion of an integral feature (surface) which can be a point, a line or an area and which is used for establishing a datum, (ISO/CD 5459-1:1998(E)).

Datum Feature

An actual feature of a part that is used to establish a datum, (ASME Y14.5M-1994).

Derived Feature

Center point, median line or median surface from one or more integral features (ISO 14660)

Examples:

1- The center of a sphere is derived feature obtained from the sphere, which is an integral feature.

2 -The median line of a cylinder is a derived feature obtained from the cylindrical surface, which is an integral feature.

E

Extracted Derived Feature

Center point, median line or median surface derived from one or more extracted integral features (ISO 14660).

Note: For convenience:

- the derived median line of an extracted cylindrical surface is called an extracted median line (see ISO 14660-2).

- the derived median surface of 2 opposite extracted planar surfaces is called an extracted median surface (see ISO 14660-2).

Extracted Integral Feature

Approximated representation of the real (integral) feature, obtained by extracting a finite number of points from the real (integral) feature performed in accordance with specified conventions. (ISO 14660)

Note: This representation is defined according to the required function of the feature. Several such representations may exist for each real (integral) feature.

F

Feature

The general term applied to a physical portion of a part, such as a surface, pin, tab, hole or slot, (ASME Y14.5M-1994).

Geometric feature, point, line or surface, (ISO 14660-1).

Feature of Size

Geometric shape defined by a linear or angular dimension which is a size(ISO 14660)

Notes

1- The features of size can be a cylinder, a sphere, two parallel opposite surfaces, a cone or a wedge.

2- In standards such as ISO 286-1 and ISO/R 1938 the terms plain workpiece and single features are used with a meaning close to "feature size".

G

Geometric Feature

Point, line or surface (ISO 14660).

I

Integral Feature

Surface or line on a surface (ISO 14660)

Note: An integral feature is intrinsically defined.

N

Nominal integral feature

Theoretically exact integral feature as defined by a technical drawing or by other means (ISO 14660).

Nominal Derived Feature

Center point, axis or median plane derived from one or more nominal integral features (ISO 14660)

Note: On the technical drawing nominal derived features are mostly represented by chain lines (dash-dotted lines).

R

Real (integral) Feature	Integral feature part of a real surface of a workpiece limited by the adjacent real (integral) features (ISO 14660). Note: No real derived feature exist.
Real Surface of a Workpiece	Set of features which physically exist and separate the entire workpiece from the surrounding medium (ISO 14660).
Restricted Surface	Portion of an integral feature (surface) which is an area and which is used for establishing a datum (ISO/CD 5459-1:1998) Note: the area of a restricted surface is relatively large compared with the complete surface.

S

Single Datum	Datum based on one surface considered alone (ISO/CD 5459-1:1998).
Situation Feature	Ideal feature which is a point, a straight line, a plane or a helix, from which the position and/or the orientation of a feature can be defined (ISO/CD 5459-1:1998) Note: Situation features are defined by and are characteristics of the class of surface involved.
Surface class	Group of surfaces having the same degree of freedom for which the surfaces are unvarying, (ISO/CDIS 5459-1:1998(E)).

T

Tolerance zone	Space limited by one or several geometrically perfect lines or surfaces, and characterized by a linear dimension, called a tolerance (ISO/FDIS 1101:2000(E))

Index

B

C

D

E

F

G

H

I

L

M

N

O

P

T

V

annotation option

option

Y

yellow reference axis