Version 5 Release 12

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CATIA Documentation



Infrastructure



Mechanical Design



Shape Design
& Styling



Analysis



Equipment &
Systems Engineering



NC Manufacturing



Product Synthesis

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Preface

The Wireframe and Surface workbench allows you to create wireframe construction elements during preliminary design and enrich existing 3D mechanical part design with wireframe and basic surface features. As a complement to Part Design, this product meets the requirements of solids-based hybrid modeling.

The features-based approach offers a productive and intuitive design environment to capture and re-use design methodologies and specifications.

As a scalable product, Wireframe and Surface can be used in cooperation with companion products such as Part Design, Assembly Design and Generative Drafting. The widest application portfolio in the industry is also accessible through interoperability with CATIA Solutions Version 4 to enable support of the full product development process from initial concept to product in operation.

The *Wireframe and Surface User's Guide* has been designed to show you how to create and edit wireframe and surface features as well as hybrid parts. There are

often several ways to reach the final result. This guide aims at illustrating these various possibilities.

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Preface

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Using This Guide

This guide is intended for the user who needs to become quickly familiar with the Wireframe and Surface product. The user should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars.

To get the most out of this guide, we suggest you start reading and performing the step-by-step tutorial Getting Started. This tutorial will show you how to create a basic part.

The next sections deal with the creation and modification of various types of wireframe and surface geometry you will need to construct parts.

You may also want to take a look at the section describing the Wireframe and Surface workbench menus and toolbars.

Up Next

Where to Find More Information

Prior to reading this guide, we recommend that you read the *Infrastructure User's Guide.*

The *Part Design User's Guide* and *Assembly Design User's Guide* may prove useful.

Conventions

Back Up

What's New?

This table identifies what new or improved capabilities have been documented in Version 5 Release 12 of the Wireframe and Surface workbench.

New Functionalities

Creating an Axis

Creating a Cylinder

Enhanced Functionalities

Creating Wireframe Geometry

Creating Points

Extremum points are now aggregated under the command that created them

New Projection option allowing to specify a surface on which the point is projected normally to the plane

Creating Circles

Possibility to compute the default plane when one of the inputs is a planar curve for the following types: Bi-tangent and radius, Bi-tangent and point, Tri-tangent, Center and tangent

Performing Operations on Shape Geometry

Joining Geometry

No propagation option: only the elements explicitly selected are part of the propagation

Splitting Elements

New Automatic extrapolation option allowing the automatic extrapolation of the cutting curve to be optional

Trimming Elements

New Automatic extrapolation option allowing the automatic extrapolation of the elements to trim to be optional

Creating Boundary Curves

Possibility to select a boundary type before selecting an edge

Extracting Geometry

Possibility to select the part to keep when the result of an extract is not connex

Translating Geometry

Two options (Creation or Modification) are now available to modify the translation behavior

Rotating Geometry

Two options (Creation or Modification) are now available to modify the rotation behavior

Performing a Symmetry on Geometry

Two options (Creation or Modification) are now available to modify the symmetry behavior

Transforming Geometry by Scaling

Two options (Creation or Modification) are now available to modify the scaling behavior

Transforming Geometry by Affinity

Two options (Creation or Modification) are now available to modify the affinity behavior

Transforming Elements From an Axis to Another

Two options (Creation or Modification) are now available to modify the axis behavior

Using Tools

Working With A Support

Possibility to retrieve the features created on a single or a multi-selection works on support
Features are now aggregated under the command that created them

Stacking Commands

Features are now aggregated under the command that created them

Editing Parameters

New commands are now concerned with the edition of parameters in the 3D geometry

Selecting Using Multi-Selection

Multi-selection is available when editing a single feature

Selecting Using Multi-Output

Multi-selection is available when editing a single feature
In case an update error occurs, a dialog box displays the features in error
New behavior for shared and unshared features aggregated under the parent command

Getting Started

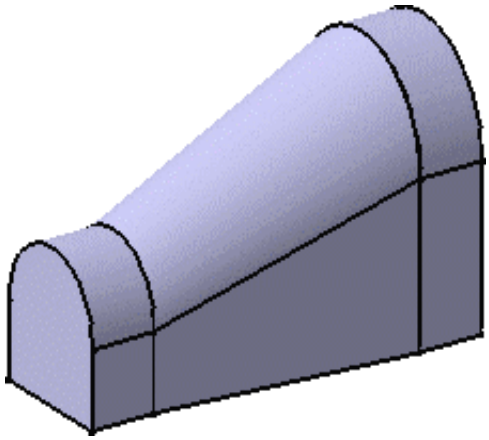
Before getting into the detailed instructions for using CATIA Version 5 Wireframe and Surface, the following tutorial aims at giving you a feel about what you can do with the product. It provides a step-by-step scenario showing you how to use key functionalities.

The main tasks described in this section are:

Entering the Workbench
Creating Wireframe Geometry
Creating a First Multi-section Surface
Creating Swept Surfaces
Creating the Second Multi-section Surface
Joining the Surfaces
Closing the Surfaces

This tutorial should take about ten minutes to complete.

The final part will look like this:



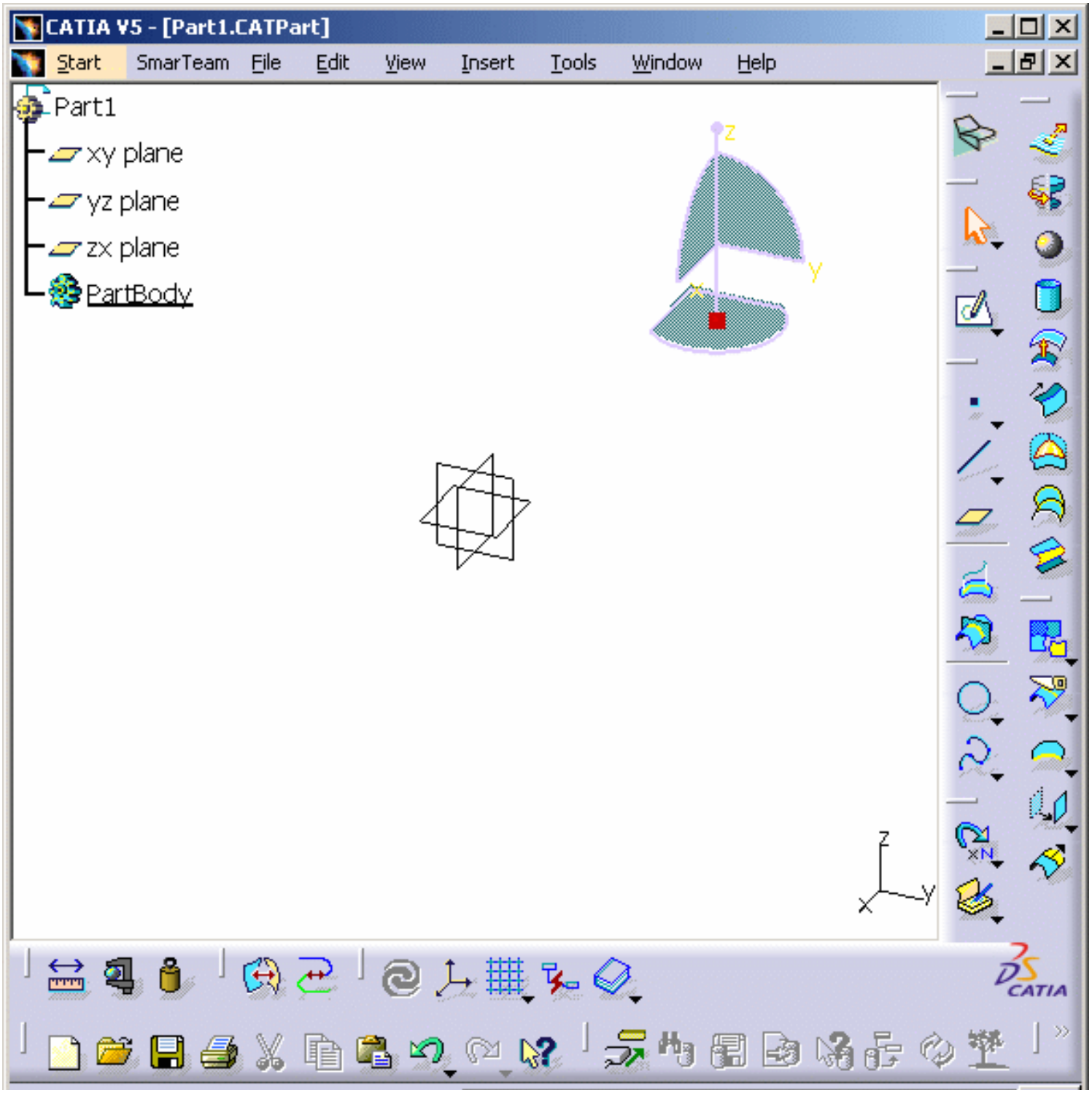
Entering the Workbench

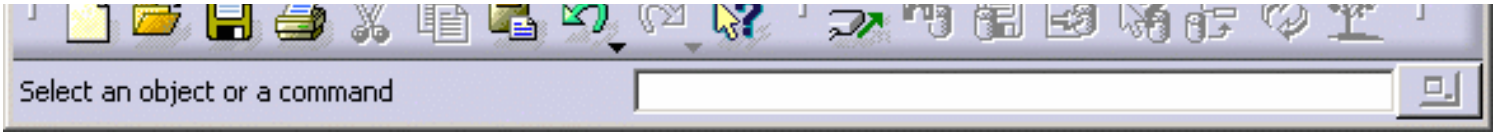
This first task shows you how to enter the Wireframe and Surface workbench and open a design part.

Before starting this scenario, you should be familiar with the basic commands common to all workbenches. These are described in the *Infrastructure User's Guide*.

1. Select **Mechanical Design -> Wireframe and Surface Design** from the **Start** menu.

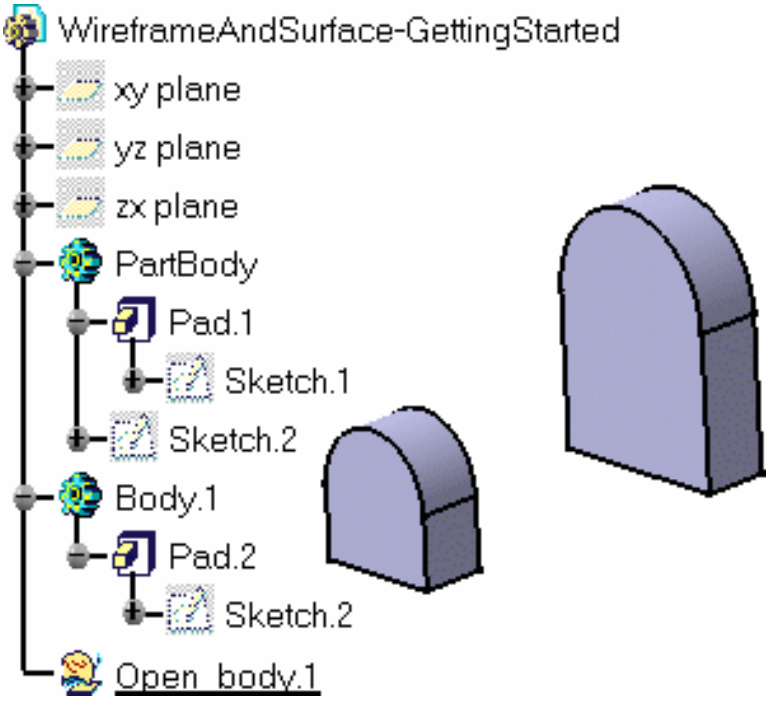
The Wireframe and Surface workbench is displayed.





2. Select **File -> Open** then select the GettingStartedWireframeAndSurface.CATPart document.

The following design part is displayed.



In the rest of this scenario, you will add to the existing elements of this part to complete the design.



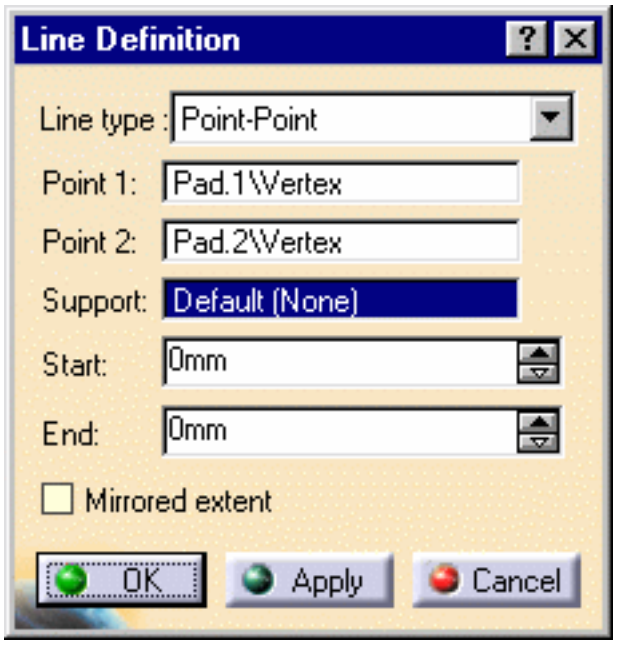
Up Next

Creating Wireframe Construction Elements

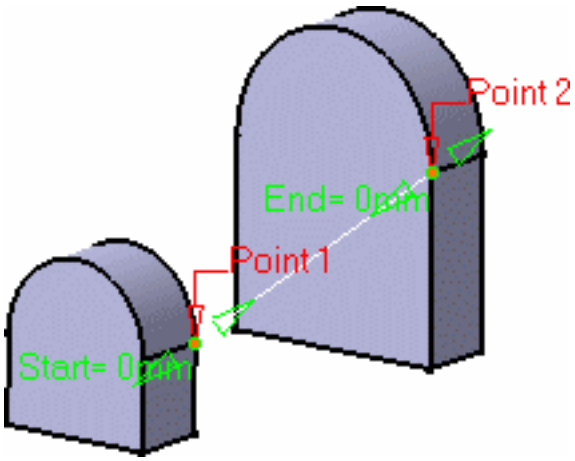
This task shows you how create wireframe construction elements using the vertices of solids.

1. Click the **Line** icon .

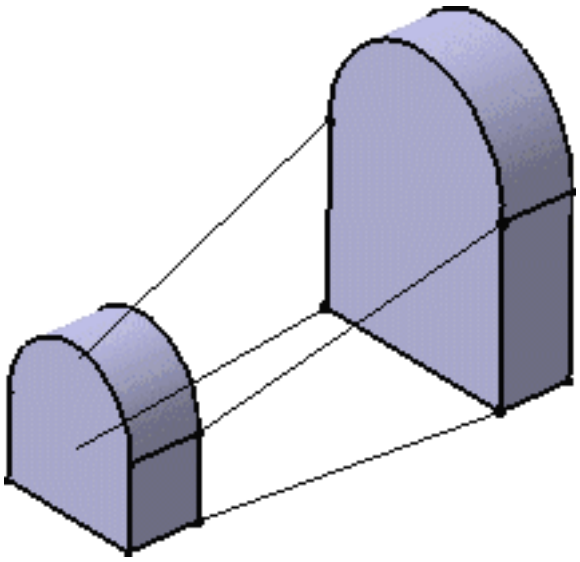
The Line Definition dialog box appears.



2. Create a line by selecting a vertex on Pad 1 and the corresponding vertex on Pad 2.



3. Repeat this steps to create four lines as shown in the opposite figure.



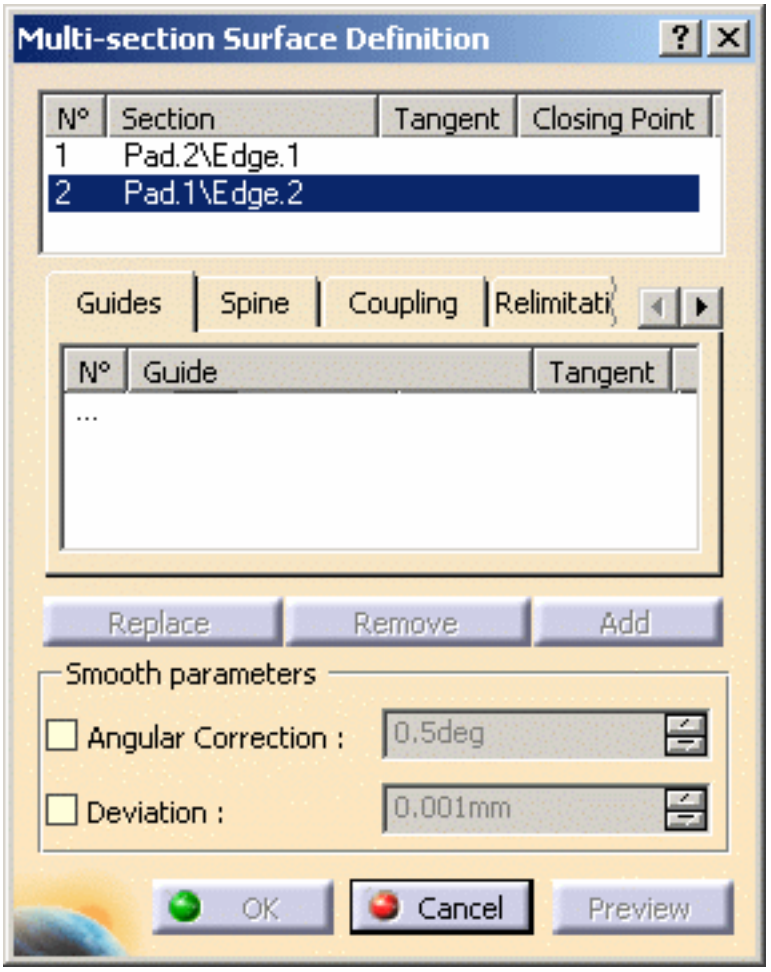
Back Up Next

Creating a First Multi-section Surface

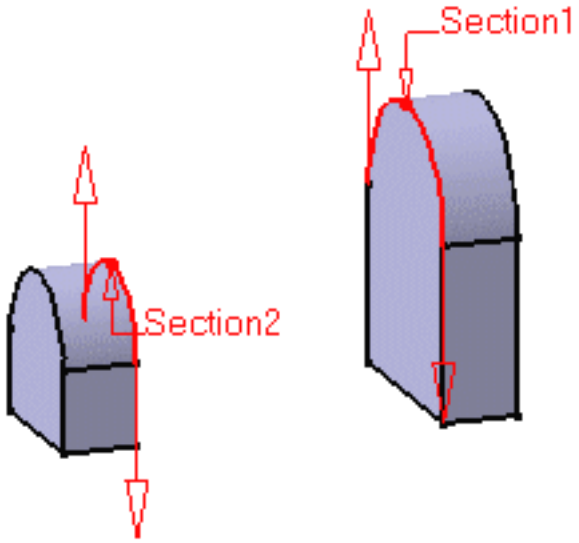
This task shows how to create a multi-section surface.

1. Click the **Multi-section surface** icon .

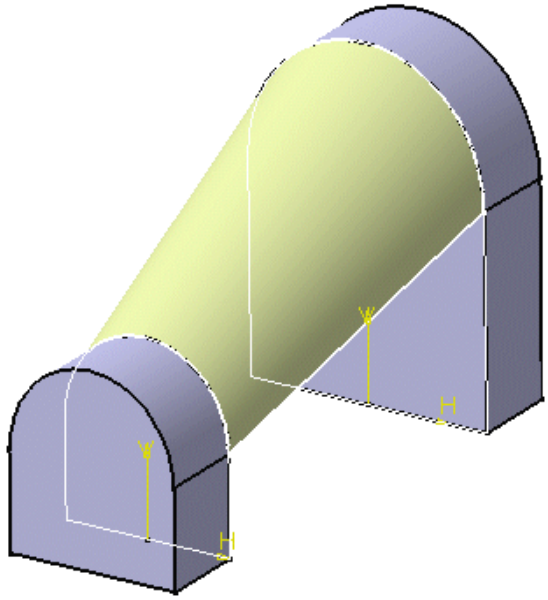
   The Multi-section Surface Definition dialog box appears.



2. Select the curved edge on each pad as sections for the multi-section surface.
   Arrows must point the same way on each side of the multi-section surface.



3. Click OK to create the multi-section surface.



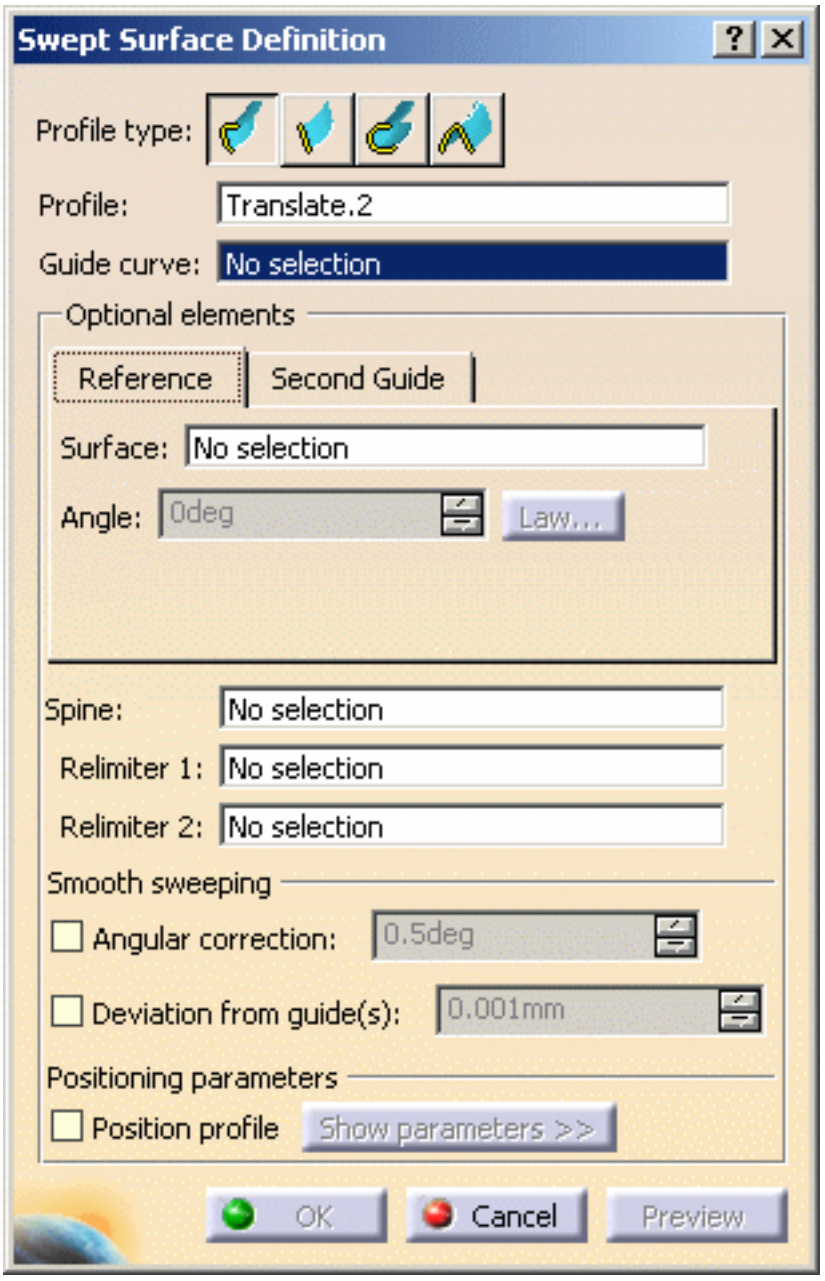
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Creating Two Swept Surfaces

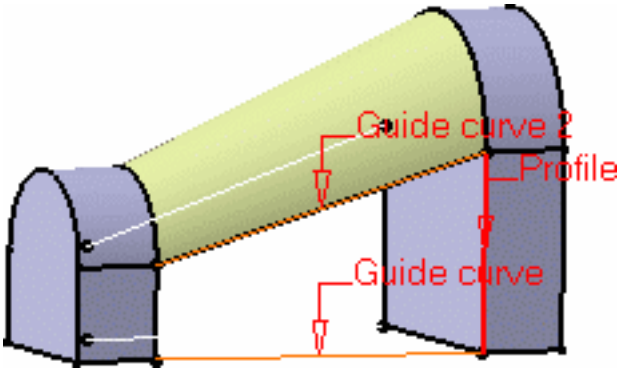
This task shows how to create two swept surfaces between opposite edges of the two pads.

1. Click the **Sweep** icon .

   The Swept Surface Definition dialog box appears.

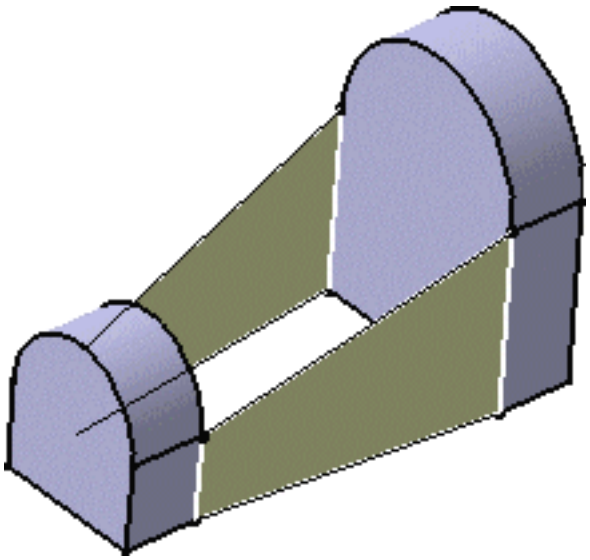


2. Select the vertical edge of Pad 2 as profile.
3. Select the bottom line as first guide curve.
4. Click the Second Guide tab then select the inclined line as second guide curve.
5. Click OK to create the swept surface.



6. Repeat these steps on the other side to create a second swept surface.

In the opposite figure the previously created lofted surface is hidden in order to illustrate the swept surfaces better.



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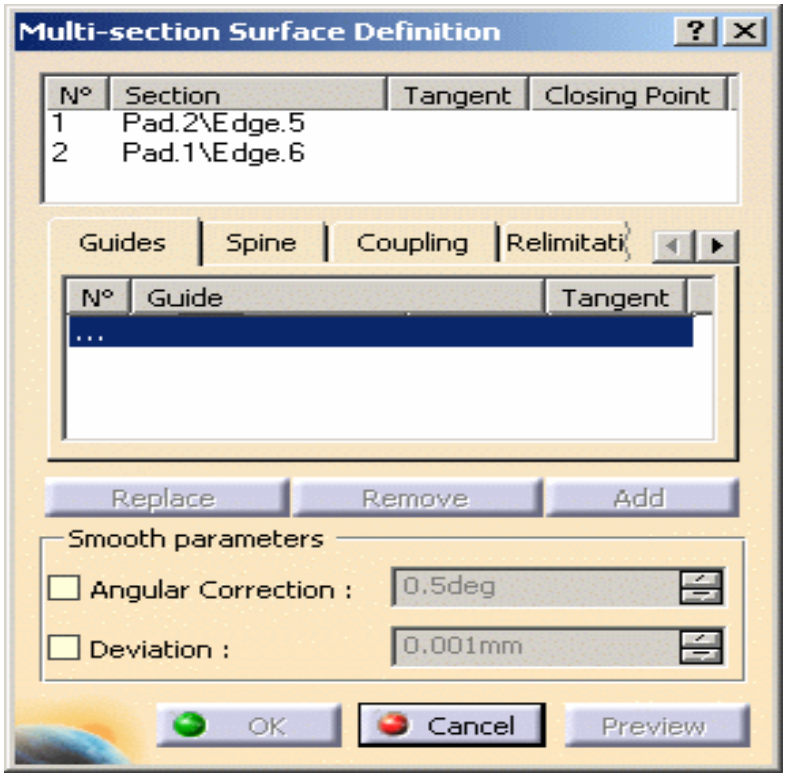
Creating a Second Loft Surface

This task shows how to create the second lofted surface at the bottom of the part.

In the illustrations below, the first loft and both sweeps have been hidden.

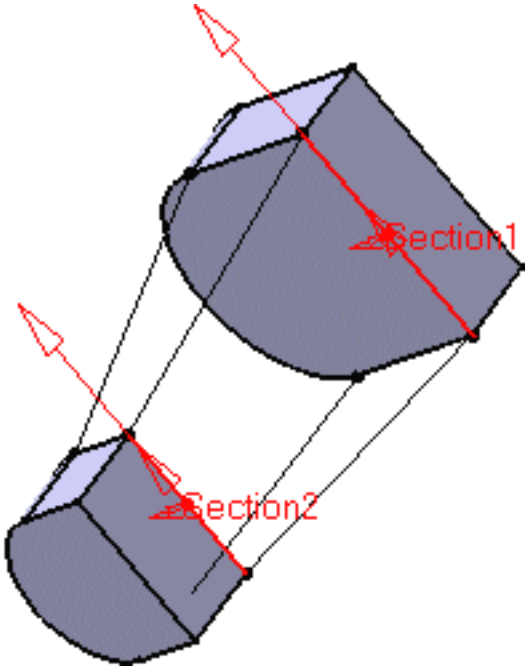
1. Click the **Loft** icon .

   The Lofted Surface Definition dialog box appears.



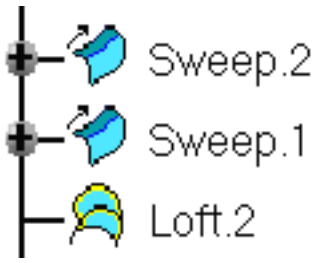
2. Select the horizontal edges on the pads as sections for the loft.

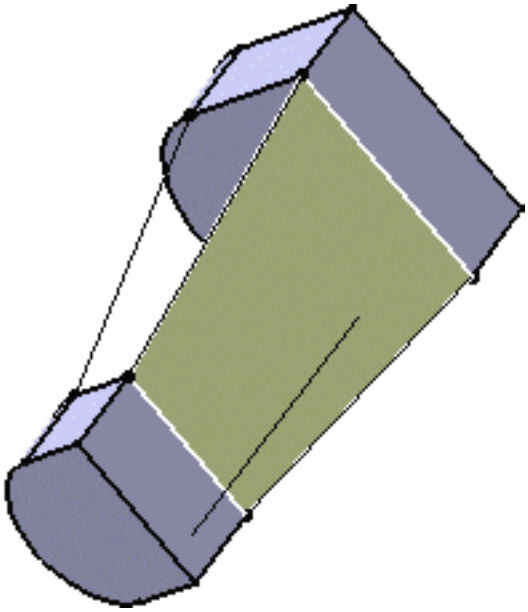
   Make sure arrows point the same way.



3. Click OK to create the surface.

The specification tree is updated to show the created surfaces.







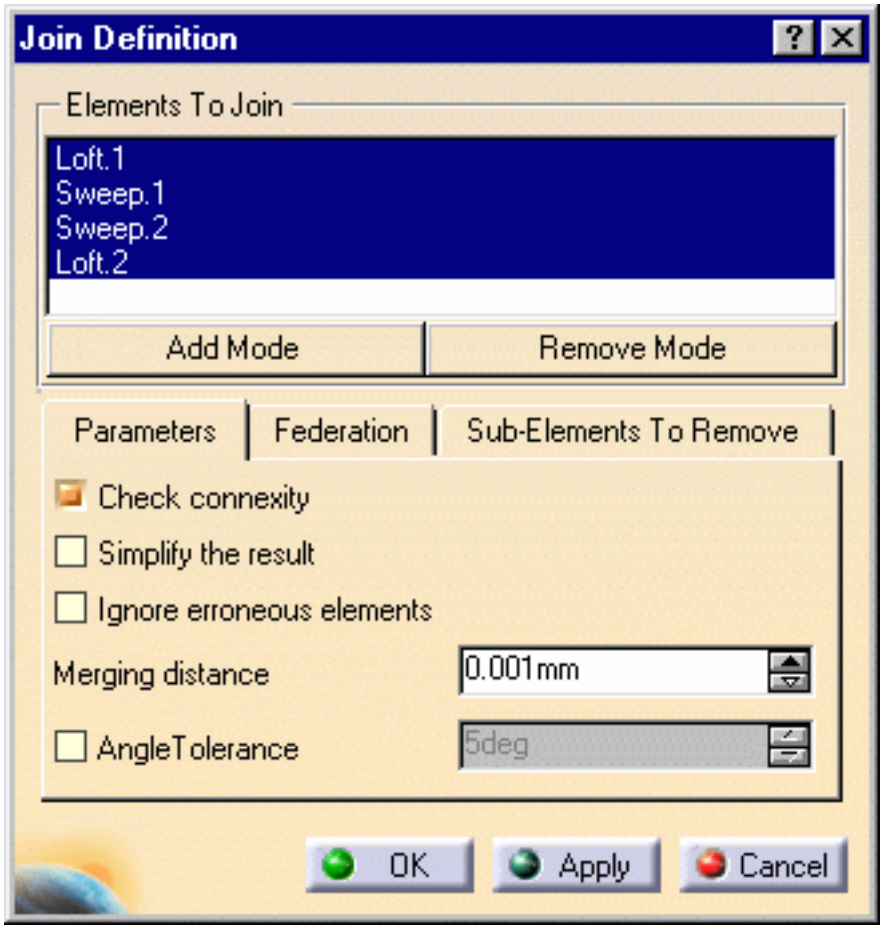
Back Up Next

Joining Surfaces

This task shows how to join the lofted and swept surfaces.

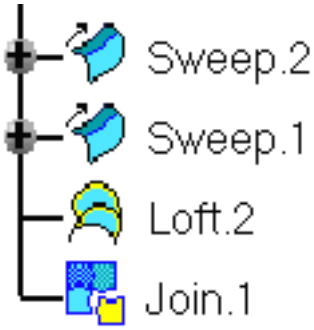
1. Click the **Join** icon .

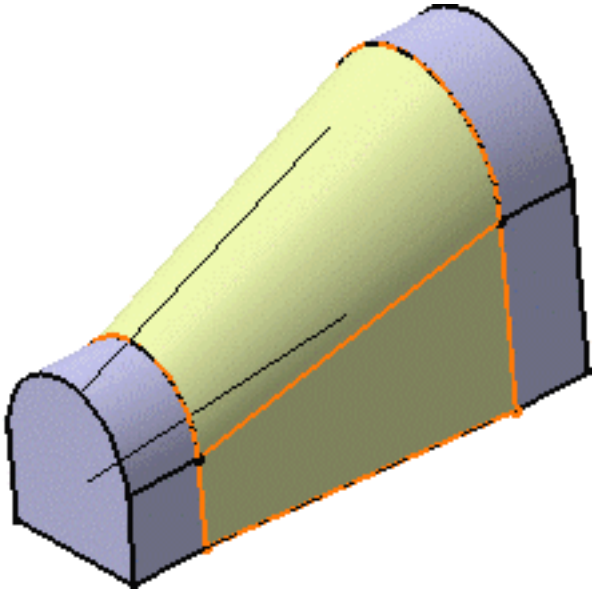
   The Join Definition dialog box appears.



2. Select the two lofted surfaces and the two swept surfaces.
3. Click OK to create the joined surface

The specification tree is updated to include the joined surface.







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Closing the Surfaces

This task shows you how to create a solid by closing the joined surface.

For this you must call up the Part Design workbench.

1. Select **Part Design** from the **Start -> Mechanical Design** menu.

   The Part Design workbench is displayed.

2. Click the **Close Surface** icon .

   This icon is available from the **Split** sub-toolbar:

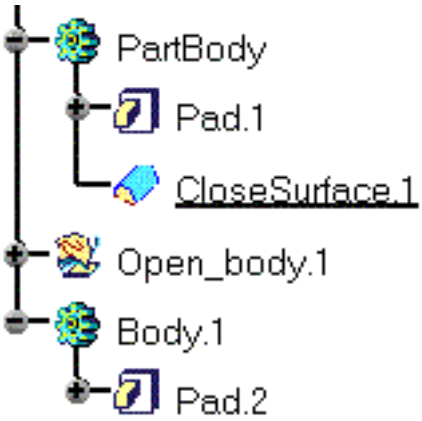
   Note that the Join element should be active in tree.

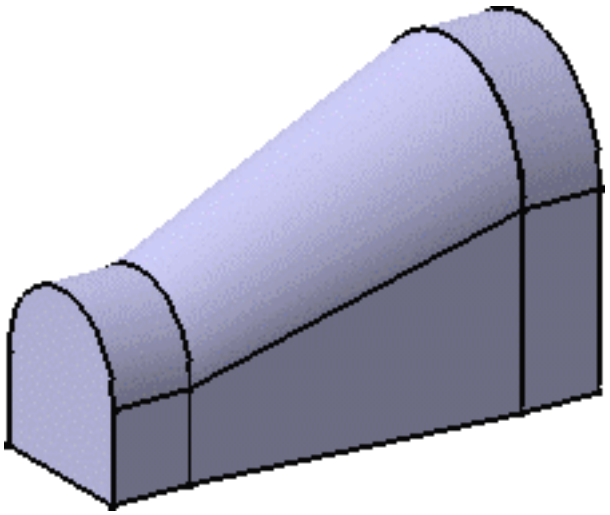
   

   The **CloseSurface Definition** dialog box appears.

3. Click OK to create the closed surface feature.

The specification tree is updated.





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Basic Tasks

The basic tasks you will perform in the Wireframe and Surface workbench are mainly the creation of wireframe and surface geometry you will use to build your part design.

This section will explain and illustrate how to create and manage various kinds of wireframe and surface geometry.

Creating Wireframe Geometry
Creating Surfaces
Performing Operations
Using Tools

When creating a geometric element, you often need to select other elements as inputs. When selecting a sketch as the input element, some restrictions apply, depending on the feature you are creating.

You should avoid selecting self-intersecting sketches as well as sketches containing heterogeneous elements such as a curve and a point for example.

However, the following elements accept sketches containing non connex elements (i.e. presenting gaps between two consecutive elements) as inputs, provided they are of the same type (homogeneous, i.e. two curves, or two points):

- Intersections
- Projections
- Extruded surfaces
- Surfaces of revolution
- Joined surfaces
- Split surfaces
- Trimmed surfaces
- All transformations: translation, rotation, symmetry, scaling, affinity and axis to axis.

Creating Wireframe Geometry

Wireframe geometry is the geometry that helps you create features when needed. Creating this geometry is a simple operation you can perform at any time.

Two creation modes are available: either you create geometry with its history or not. Geometry with no history is called a datum. Please refer to Creating Datums for more information.



Create points by coordinates: enter X, Y, Z coordinates.

Create points on a curve: select a curve and possibly a reference point, and enter a length or ratio.

Create points on a plane: select a plane and possibly a reference point, then click the plane.

Create points on a surface: select a surface and possibly a reference point, an element to set the projection orientation, and a length.

Create points as a circle center: select a circle

Create points at tangents: select a curve and a line.

Create point between another two points: select two points



Create multiple points: select a curve or a point on a curve, and possibly a reference point, set the number of point instances, indicate the creation direction or indicate the spacing between points.



Create lines between two points: select two points

Create lines based on a point and a direction: select a point and a line, then specify the start and end points of the line.

Create lines at an angle or normal to a curve: select a curve and its support, a point on the curve, then specify the angle value, the start and end points of the line.

Create lines tangent to a curve: select a curve and a reference point, then specify the start and end points of the line.

Create lines normal to a surface: select a surface and a reference point, then specify the start and end points of the line.

Create bisecting lines: select two lines and a starting point, then choose a solution.



Create an axis: select a geometric element, a direction, then choose the axis type.



Create polylines: select at least two points, then define a radius for a blending curve is needed


Create an offset plane: select an existing plane, and enter an offset value.

Create a parallel plane through a point: select an existing plane and a point. The resulting plane is parallel to the reference plane and passes through the point.

Create a plane at an angle: select an existing plane and a rotation axis, then enter an angle value (90° for a plane normal to the reference plane).

Create a plane through three points: select any three points

Create a plane through two lines : select any two lines

Create a plane through a point and a line : select any point and line

Create a plane through a planar curve: select any planar curve

Create a plane normal to a curve: select any curve and a point

Create a plane tangent to a surface: select any surface and a point

Create a plane based on its equation: key in the values for the Ax + Bu + Cz = D equation

Create a mean plane through several points: select any three, or more, points


Create n planes between two planes: select two planes, and specify the number of planes to be created


Create a circle based on a point and a radius: select a point as the circle center, a support plane or surface, and key in a radius value. For circular arcs, specify the start and end angles.

Create a circle from two points: select a point as the circle center, a passing point, and a support plane or surface. For circular arcs, specify the start and end angles.

Create a circle from two points and a radius: select the two passing points, a support plane or surface, and key in a radius value. For circular arcs, specify the arc based on the selected points.

Create a circle from three points: select three points. For circular arcs, specify the arc based on the selected points.

Create a circle tangent to two curves, at a point: select two curves, a passing point, a support plane or surface, and click where the circle should be created. For circular arcs, specify the arc based on the selected points.

Create a circle tangent to two curves, with a radius: select two curves, a support surface, key in a radius value, and click where the circle should be created. For circular arcs, specify the arc based on the selected points.

Create a circle tangent to three curves: select three curves.


Create splines: select two or more points, if needed a support surface, set tangency conditions and close the spline if needed.


Create a helix: select a starting and a direction, then specify the helix parameters.


Create corners: select a first reference element (curve or point), select a curve, a support plane or surface, and enter a radius value.


Creating connect curves: select two sets of curve and point on the curve, set their continuity type and, if needed, tension value.


Create projections: select the element to be projected and its support, specify the projection direction,


Create intersections: select the two elements to be intersected

Up Next

Creating Points

This task shows the various methods for creating points:

- by coordinates
- on a curve
- on a plane
- on a surface
- at a circle center
- tangent point on a curve
- between

Open the Points3D-1.CATPart document.

1. Click the **Point** icon .

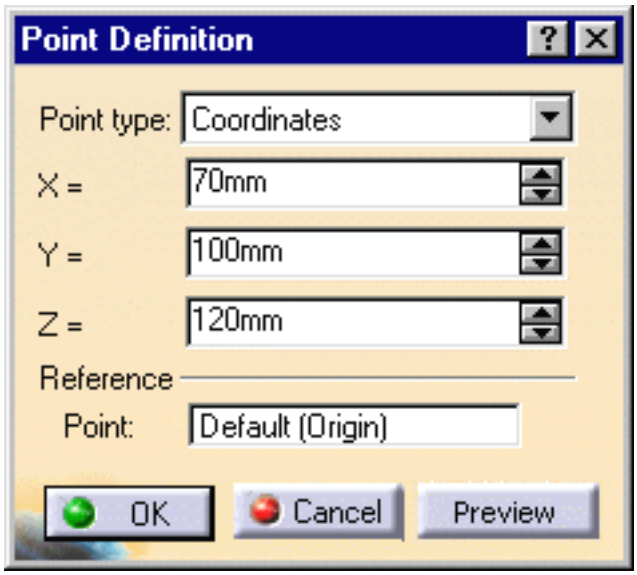
   The Point Definition dialog box appears.

2. Use the combo to choose the desired point type.

Coordinates

- Enter the X, Y, Z coordinates in the current axis-system.
- Optionally, select a reference point.

  The corresponding point is displayed.



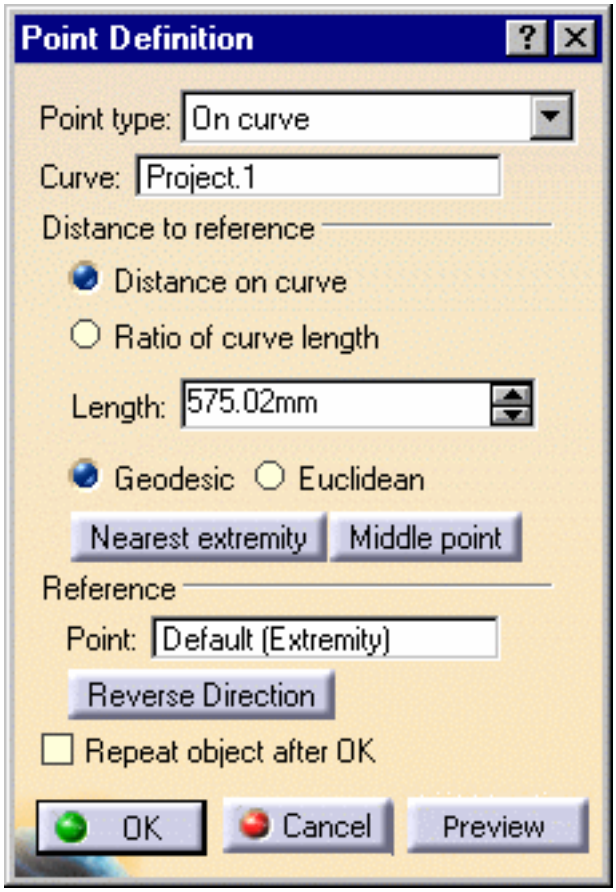
When creating a point within a user-defined axis-system, note that the **Coordinates in absolute axis-system** check button is added to the dialog box, allowing you to be define, or simply find out, the point's coordinates within the document's default axis-system.
If you create a point using the coordinates method and an axis system is already defined and set as current, the point's coordinates are defined according to current the axis system. As a consequence, the point's coordinates are not displayed in the specification tree.

The axis system must be different from the absolute axis.

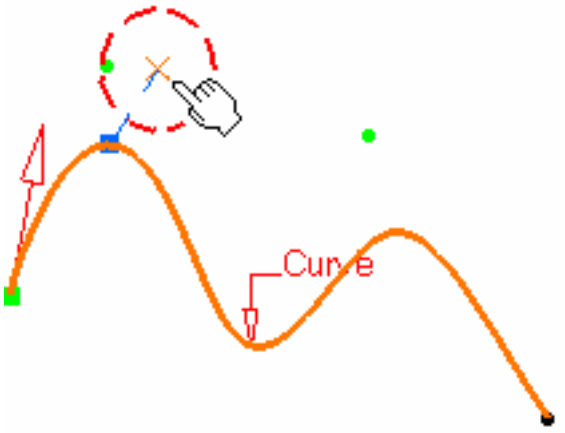
On curve

- Select a curve
- Optionally, select a reference point.

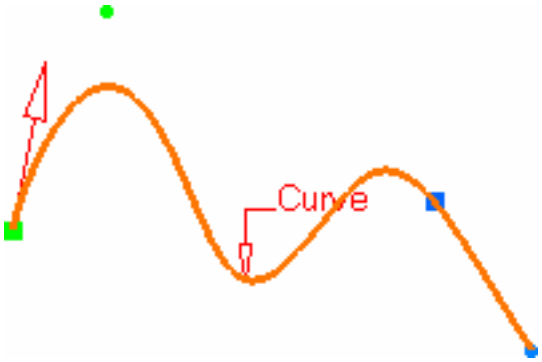
  If this point is not on the curve, it is projected onto the curve.
  If no point is selected, the curve's extremity is used as reference.



- Select an option point to determine whether the new point is to be created:
  - at a given distance along the curve from the reference point
  - a given ratio between the reference point and the curve's extremity.



- Enter the distance or ratio value.
  If a distance is specified, it can be:
  - a geodesic distance: the distance is measured along the curve
  - an Euclidean distance: the distance is measured in relation to the reference point (absolute value).



  The corresponding point is displayed.

You can also:

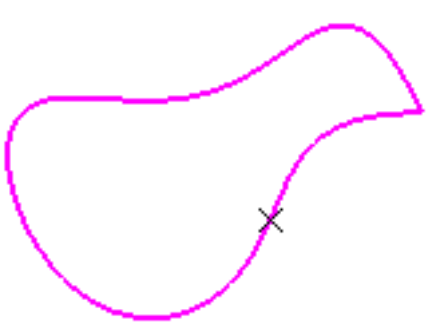
- click the **Nearest extremity** button to display the point at the nearest extremity of the curve.
- click the **Middle Point** button to display the mid-point of the curve.
- use the **Reverse Direction** button to display:
  - the point on the other side of the reference point (if a point was selected originally)
  - the point from the other extremity (if no point was selected originally).
- click the **Repeat object after OK** if you wish to create equidistant points on the curve, using the currently created point as the reference, as described in Creating Multiple Points in the Wireframe and Surface User's Guide .

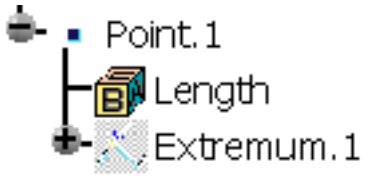
  You will also be able to create planes normal to the curve at these points, by checking the **Create normal planes also** button, and to create all instances in a new Open Body by checking the **Create in a new open body** button.
  If the button is not checked the instances are created in the current Open Body.



- If the curve is infinite and no reference point is explicitly given, by default, the reference point is the projection of the model's origin
- If the curve is a closed curve, either the system detects a vertex on the curve that can be used as a reference point, or it creates an extremum point, and highlights it (you can then select another one if you wish) or the system prompts you to manually select a reference point.

Extremum points created on a closed curve are now aggregated under their parent command and put in no show in the specification tree.

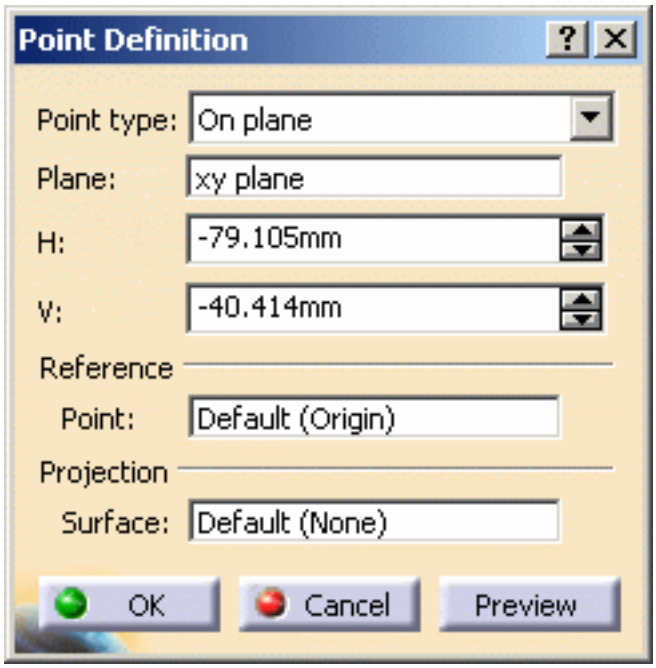




On plane

- Select a plane.
- Optionally, select a point to define a reference for computing coordinates in the plane.

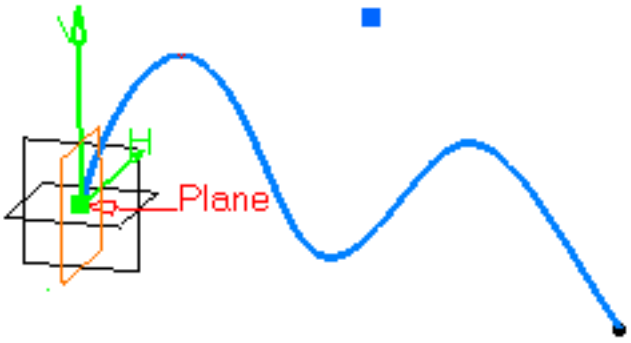
  If no point is selected, the projection of the model's origin on the plane is taken as reference.



- Optionally, select a surface on which the point is projected normally to the plane.

  If no surface is selected, the behavior is the same.

Furthermore, the reference direction (H and V vectors) is computed as follows:
With N the normal to the selected plane (reference plane), H results from the vectorial product of Z and N (H = Z^N).
If the norm of H is strictly positive then V results from the vectorial product of N and H (V = N^H).
Otherwise, V = N^X and H = V^N.

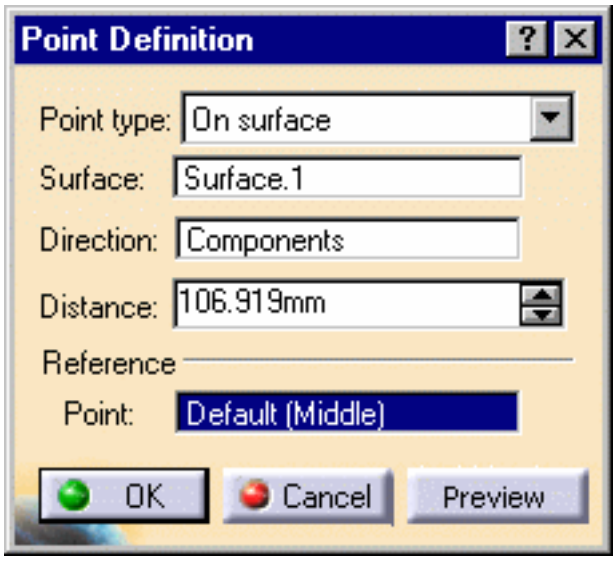


Would the plane move, during an update for example, the reference direction would then be projected on the plane.

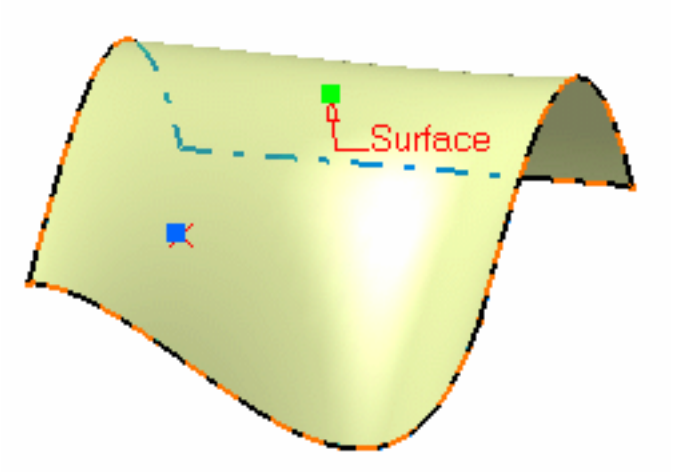
- Click in the plane to display a point.

On surface

- Select the surface where the point is to be created.

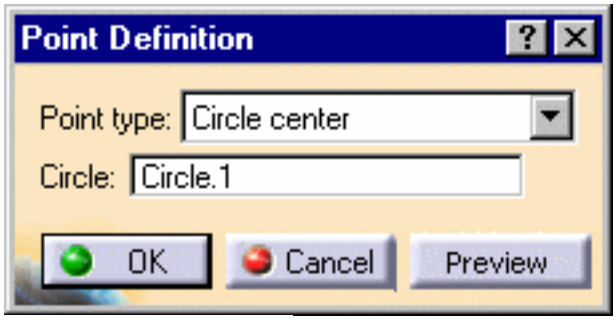


- Optionally, select a reference point. By default, the surface's middle point is taken as reference.
- You can select an element to take its orientation as reference direction or a plane to take its normal as reference direction. You can also use the contextual menu to specify the X, Y, Z components of the reference direction.
- Enter a distance along the reference direction to display a point.

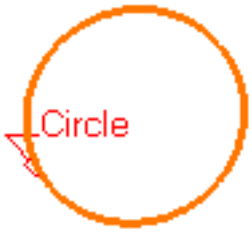


Circle center

- Select a circle, circular arc, or ellipse.

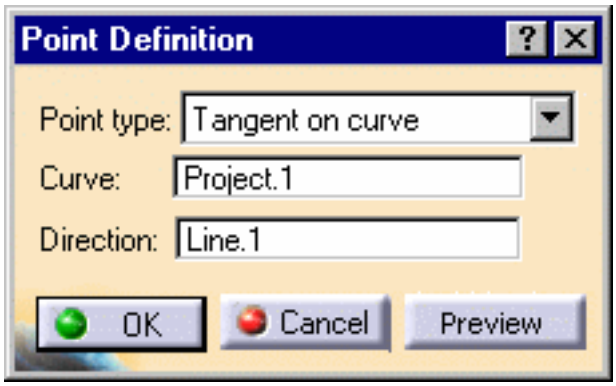


A point is displayed at the center of the selected element.



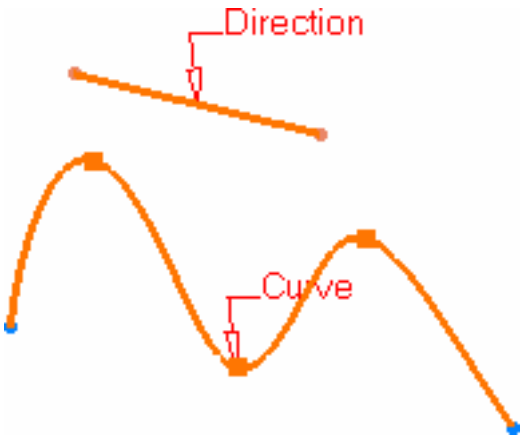
Tangent on curve

- Select a planar curve and a direction line.



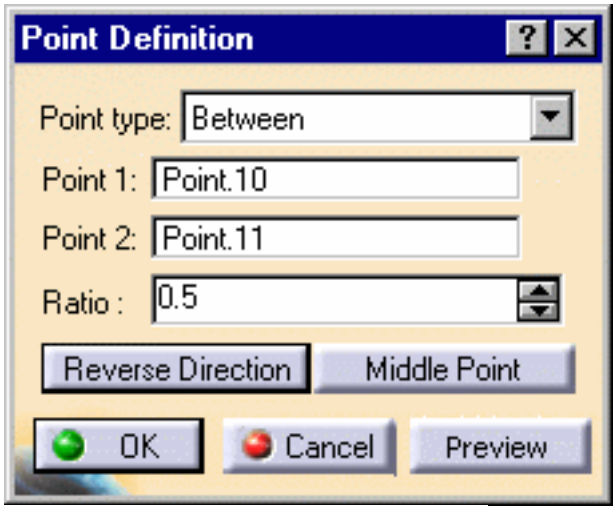
A point is displayed at each tangent.

The Multi-Result Management dialog box is displayed because several points are generated.



- Click **YES**: you can then select a reference element, to which only the closest point is created.
- Click **NO**: all the points are created.

Between



- Select any two points.
- Enter the ratio, that is the percentage of the distance from the first selected point, at which the new point is to be.
  You can also click **Middle Point** button to create a point at the exact midpoint (ratio = 0.5).



- Use the **Reverse direction** button to measure the ratio from the second selected point.

If the ratio value is greater than 1, the point is located on the virtual line beyond the selected points.

3. Click OK to create the point.

The point (identified as Point.xxx) is added to the specification tree.



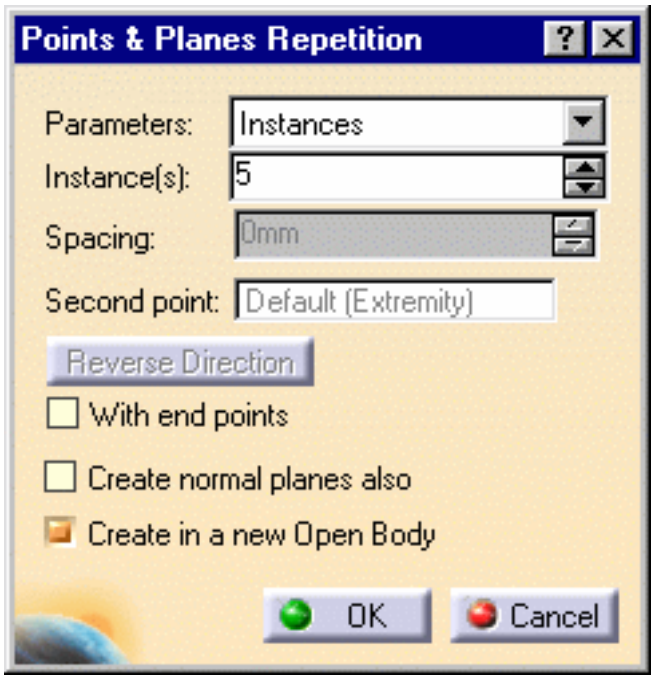
Creating Multiple Points

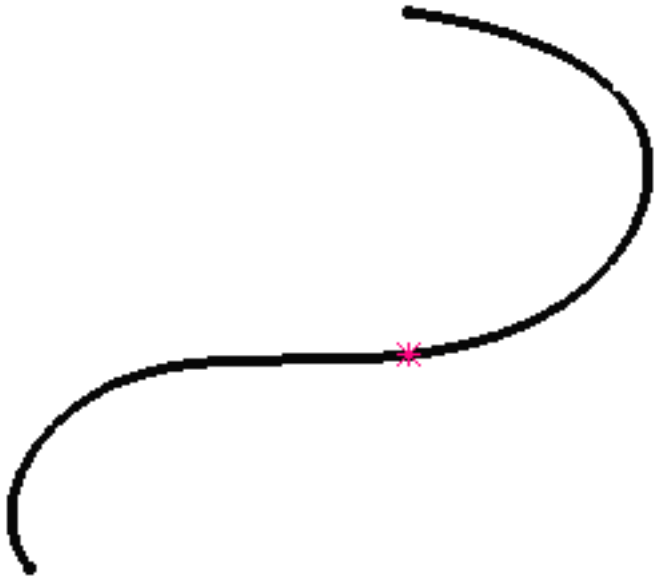
This task shows how to create several points at a time.

Open the MultiplePoints1.CATPart document.
Display the Points toolbar by clicking and holding the arrow from the **Point** icon.

1. Click the **Point & Planes Repetition** icon .
2. Select a curve or a Point on curve.

   The Multiple Points Creation dialog box appears.

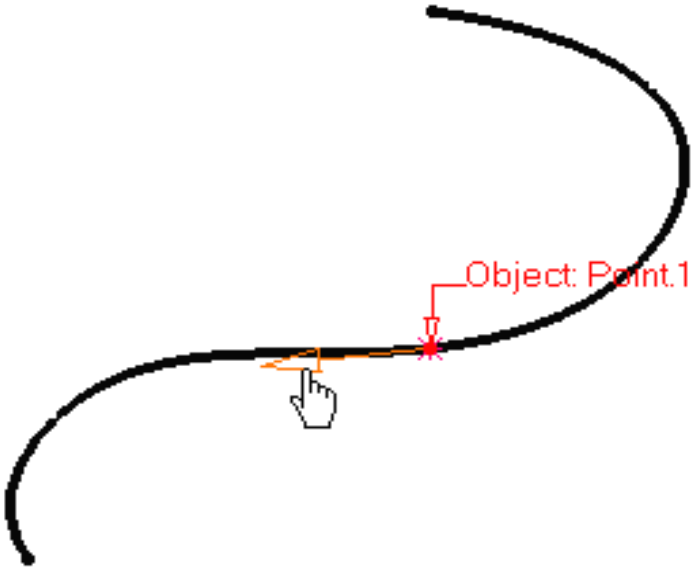




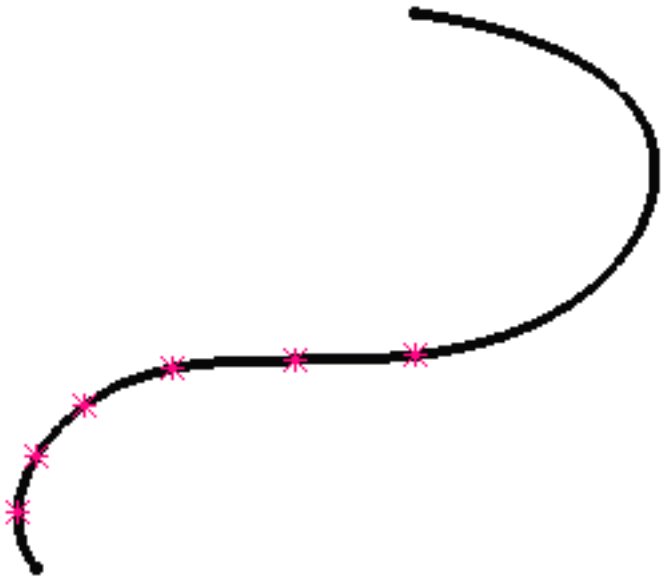
3. Define the number or points to be created (instances field). Here we chose 5 instances.

   You can choose the side on which the points are to be created in relation to the initially selected point on a curve. Simply use the Reverse Direction button, or clicking on the arrow in the geometry.

   If you check the **With end points** option, the last and first instances are the curve end points.

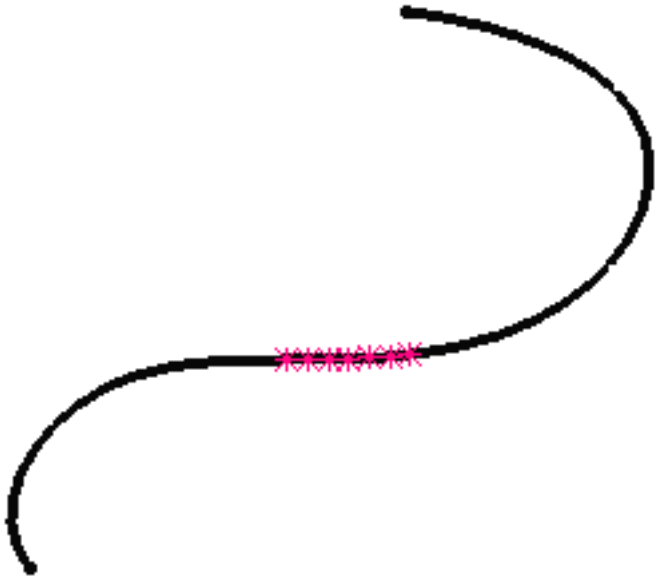


4. Click OK to create the point instances, evenly spaced over the curve on the direction indicated by the arrow.



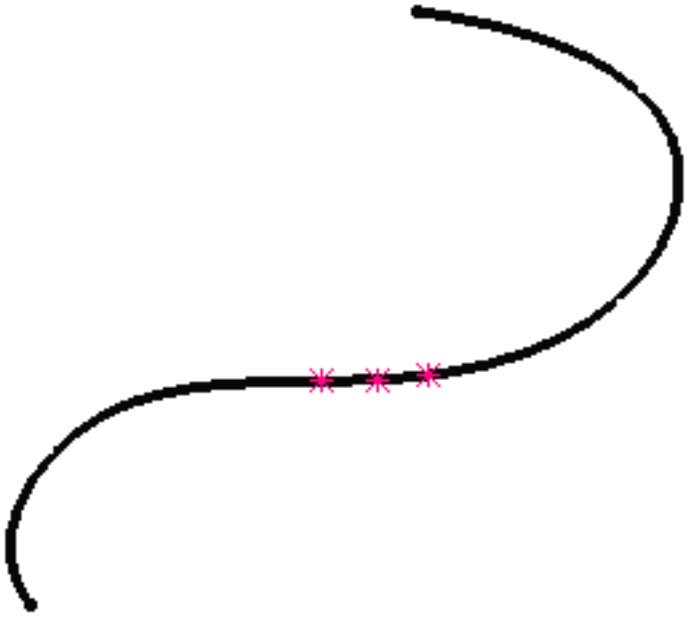
The points (identified as Point.xxx as for any other type of point) are added to the specification tree.

- If you selected a point on a curve, you can select a second point, thus defining the area of the curve where points should be created.
  Simply click the **Second point** field in the Multiple Points Creation dialog box, then select the limiting point.
  If you selected the Point2 created above as the limiting point, while keeping the same values, you would obtain the following:

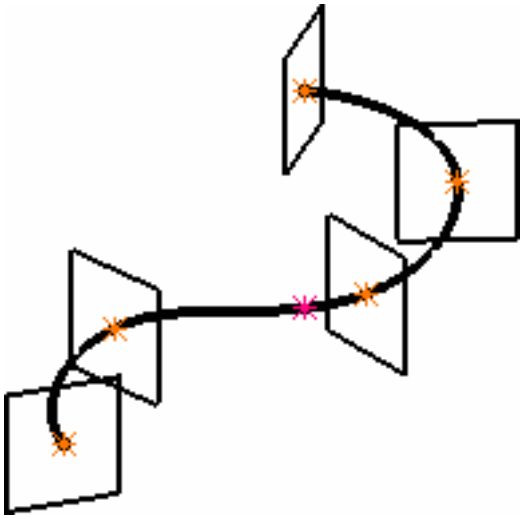


> If the selected point on curve already has a Reference point (as described in Creating Points - on curve), this reference point is automatically taken as the second point.
> By default, the **Second point** is one of the endpoints of the curve.

- When you select a point on a curve, the **Instances & spacing** option is available from the **Parameters** field.
  In this case, points will be created in the given direction and taking into account the **Spacing** value.
  For example, three instances spaced by 10mm.



- Check the **Create normal planes also** to automatically generate planes at the point instances.
- Check the **Create in a new open body** if you want all object instances in a separate Open Body.
  A new Open Body will be created automatically.



If the option is not checked the instances are created in the current Open Body.

Back Up Next

Creating Lines

This task shows the various methods for creating lines:

- point to point
- point and direction
- angle or normal to curve
- tangent to curve
- normal to surface
- bisecting

Open the Lines1.CATPart document.

1. Click the **Line** icon .

   The Line Definition dialog box appears.

2. Use the combo to choose the desired line type.

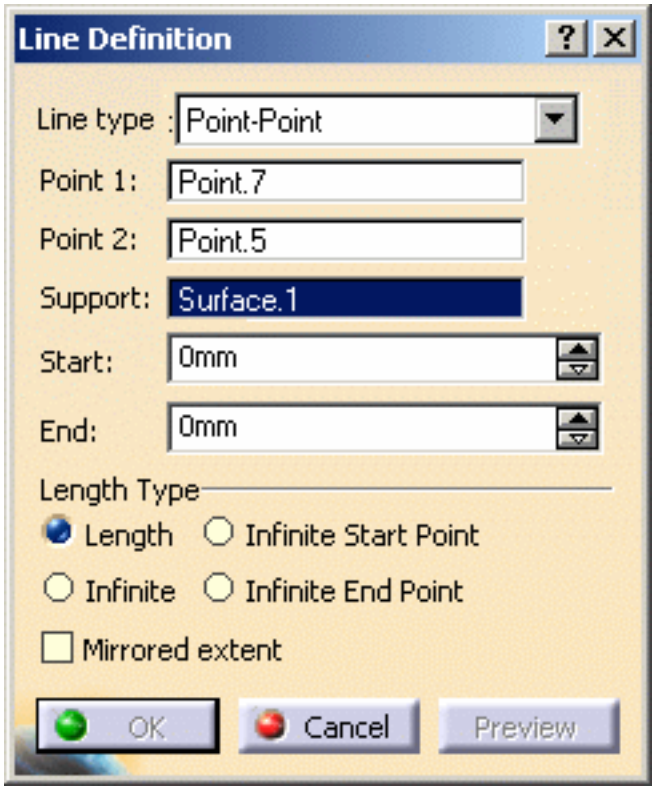
A line type will be proposed automatically in some cases depending on your first element selection.

Point - Point

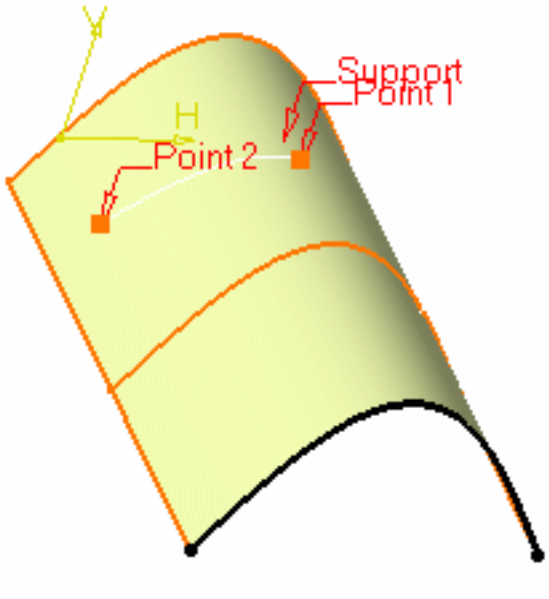
This command is only available with the Generative Shape Design 2 product.

- Select two points.

A line is displayed between the two points.
Proposed **Start** and **End** points of the new line are shown.

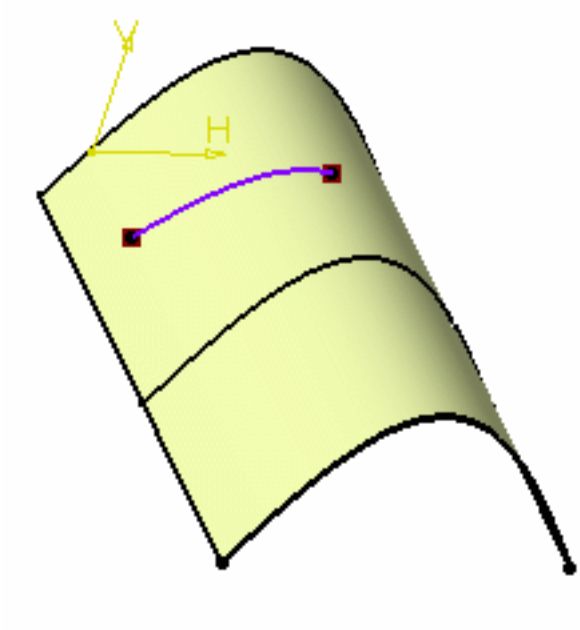


- If needed, select a support surface. In this case a geodesic line is created, i.e. going from one point to the other according to the shortest distance along the surface geometry (blue line in the illustration below). If no surface is selected, the line is created between the two points based on the shortest distance.



If you select two points on closed surface (a cylinder for example), the result may be unstable. Therefore, it is advised to split the surface and only keep the part on which the geodesic line will lie.

The geodesic line is not available with the Wireframe and Surface workbench.

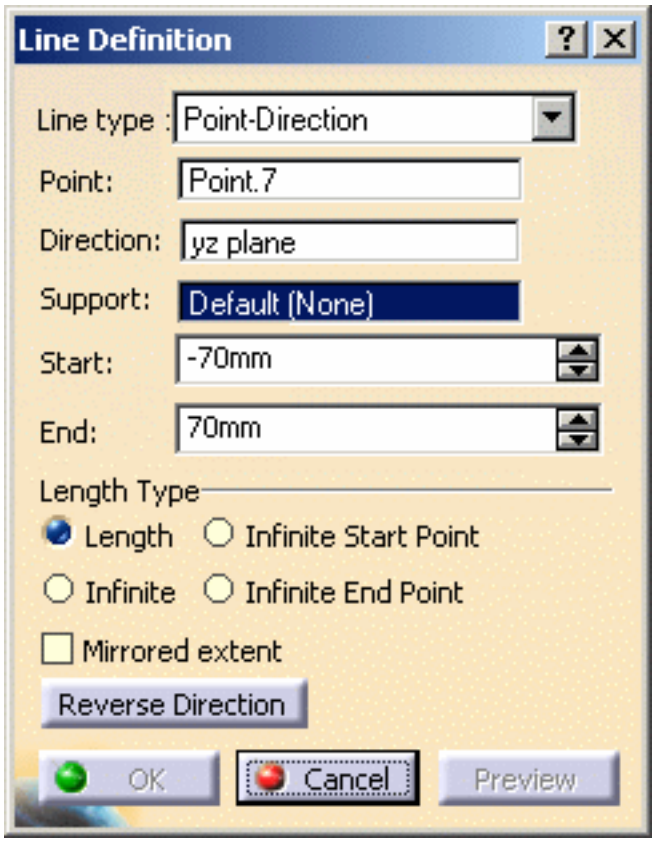


- Specify the **Start** and **End** points of the new line, that is the line endpoint location in relation to the points initially selected. These **Start** and **End** points are necessarily beyond the selected points, meaning the line cannot be shorter than the distance between the initial points.
- Check the **Mirrored extent** option to create a line symmetrically in relation to the selected **Start** and **End** points.

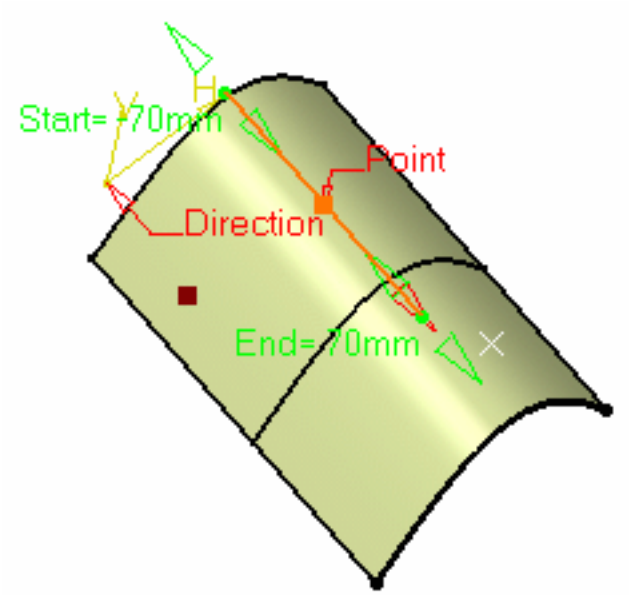
The projections of the 3D point(s) must already exist on the selected support.

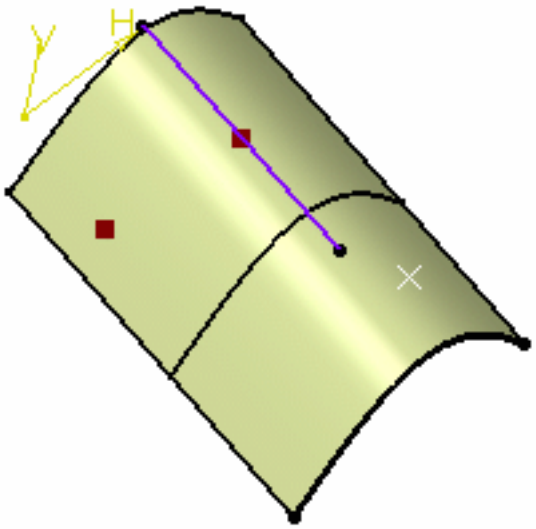
Point - Direction

- Select a reference **Point** and a **Direction** line.
  A vector parallel to the direction line is displayed at the reference point.
  Proposed **Start** and **End** points of the new line are shown.



- Specify the **Start** and **End** points of the new line.
  The corresponding line is displayed.





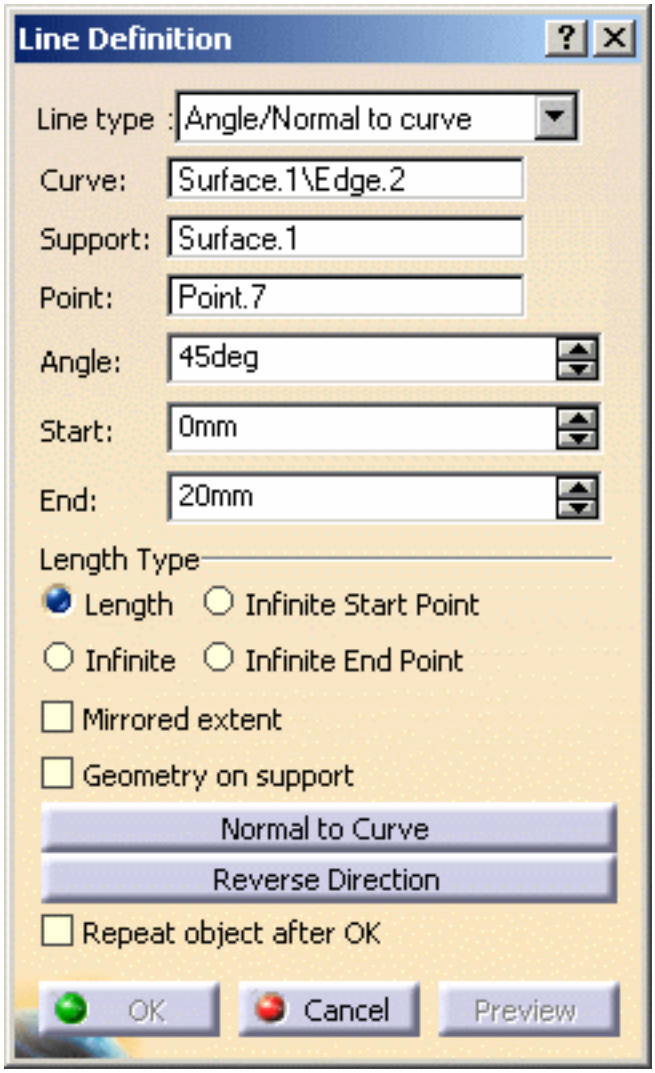
The projections of the 3D point(s) must already exist on the selected support.

Angle or normal to curve

- Select a reference **Curve** and a **Support** surface containing that curve.

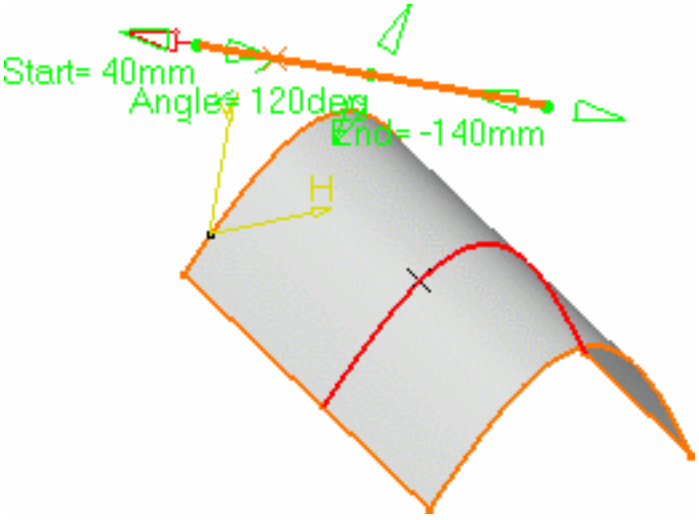
- If the selected curve is planar, then the **Support** is set to Default (Plane).

- If an explicit **Support** has been defined, a contextual menu is available to clear the selection.



- Select a **Point** on the curve.
- Enter an **Angle** value.

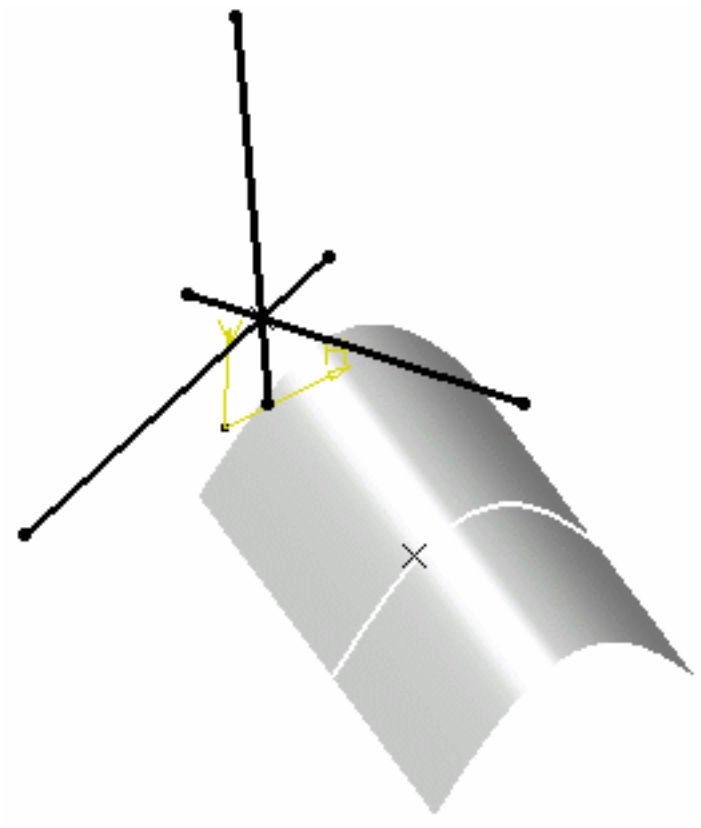
A line is displayed at the given angle with respect to the tangent to the reference curve at the selected point. These elements are displayed in the plane tangent to the surface at the selected point.
You can click on the **Normal to Curve** button to specify an angle of 90 degrees.
Proposed **Start** and **End** points of the line are shown.



- Specify the **Start** and **End** points of the new line.
The corresponding line is displayed.

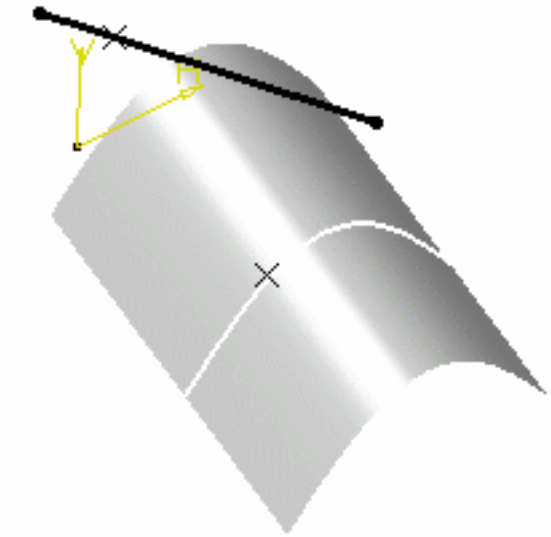
- Click the **Repeat object after OK** if you wish to create more lines with the same definition as the currently created line.
  In this case, the Object Repetition dialog box is displayed, and you key in the number of instances to be created before pressing OK.



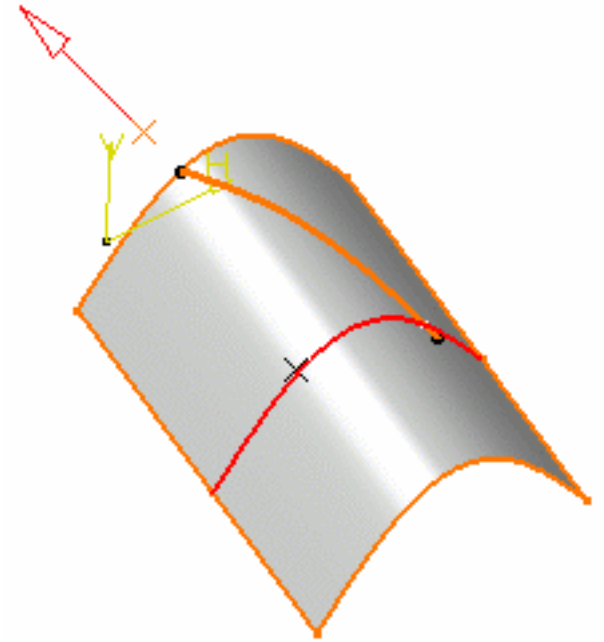


As many lines as indicated in the dialog box are created, each separated from the initial line by a multiple of the **angle** value.

You can select the **Geometry on Support** check box if you want to create a geodesic line onto a support surface.
The figure below illustrates this case.



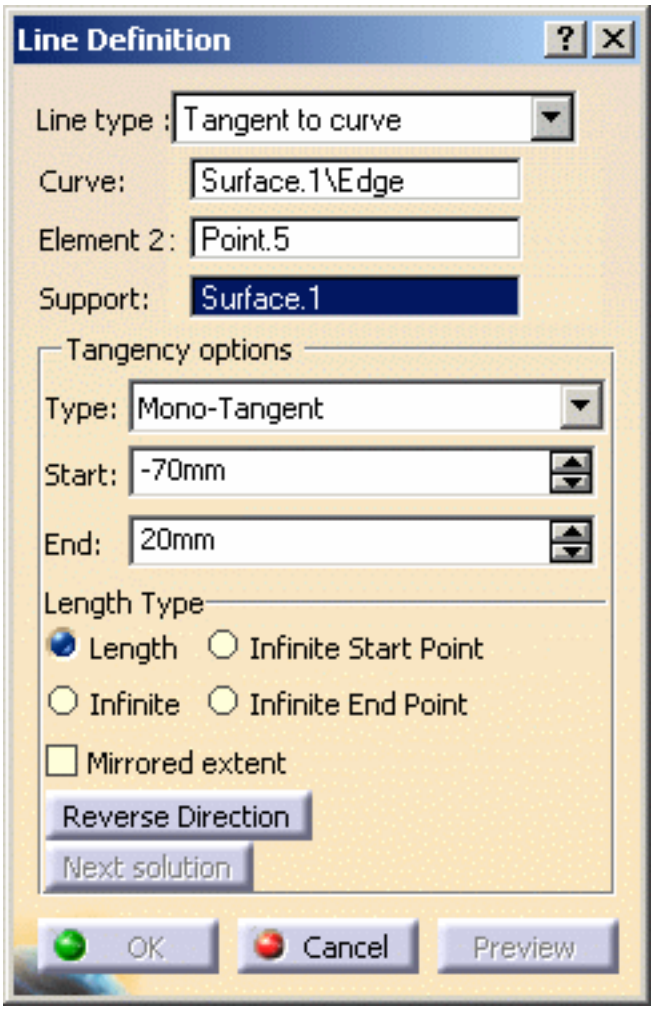
Geometry on support option not checked



Geometry on support option checked

This line type enables to edit the line's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

Tangent to curve

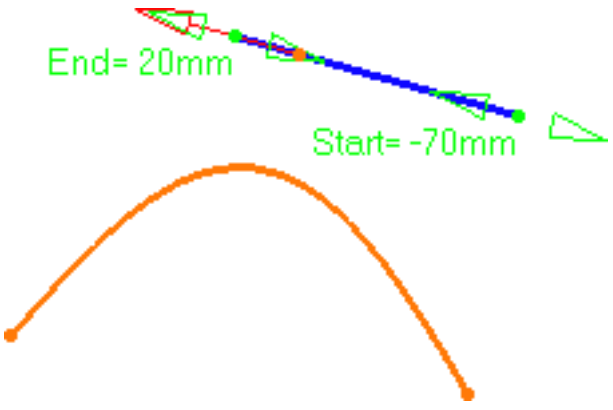


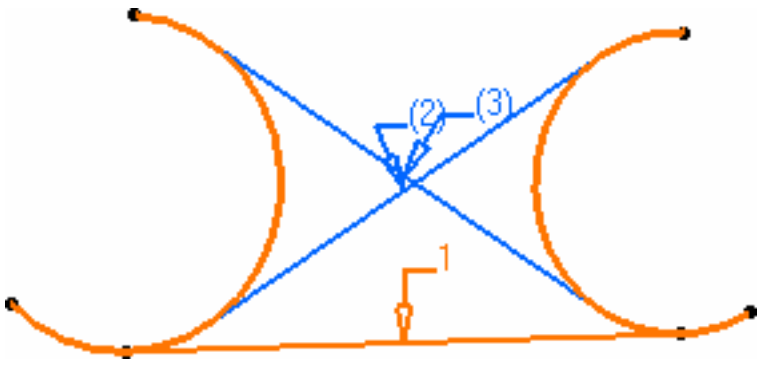
- Select a reference **Curve** and a **point** or another **Curve** to define the tangency.
  - if a point is selected (mono-tangent mode): a vector tangent to the curve is displayed at the selected point.
  - If a second curve is selected (or a point in bi-tangent mode), you need to select a support plane. The line will be tangent to both curves.

    - If the selected curve is a line, then the **Support** is set to Default (Plane).

    - If an explicit **Support** has been defined, a contextual menu is available to clear the selection.

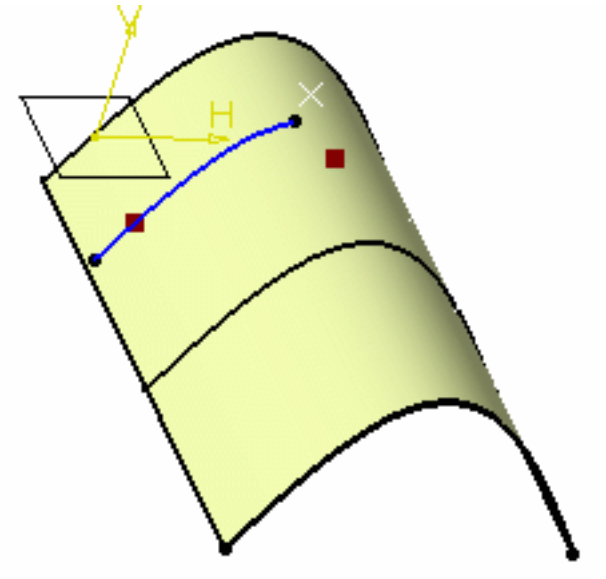
    When several solutions are possible, you can choose one (displayed in red) directly in the geometry, or using the **Next Solution** button.





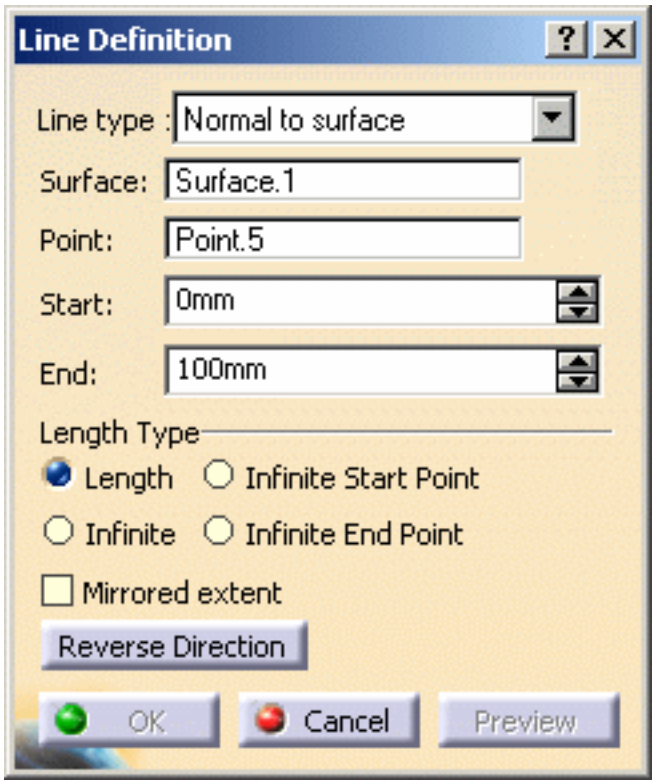
Line tangent to curve at a given point *Line tangent to two curves*

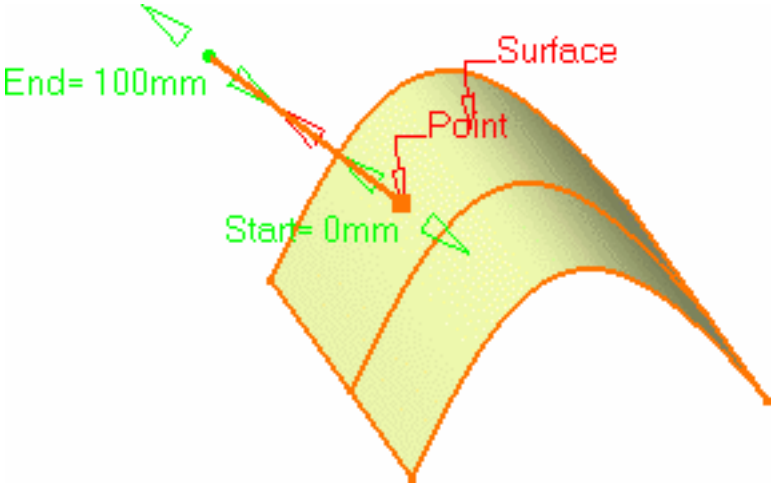
- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.



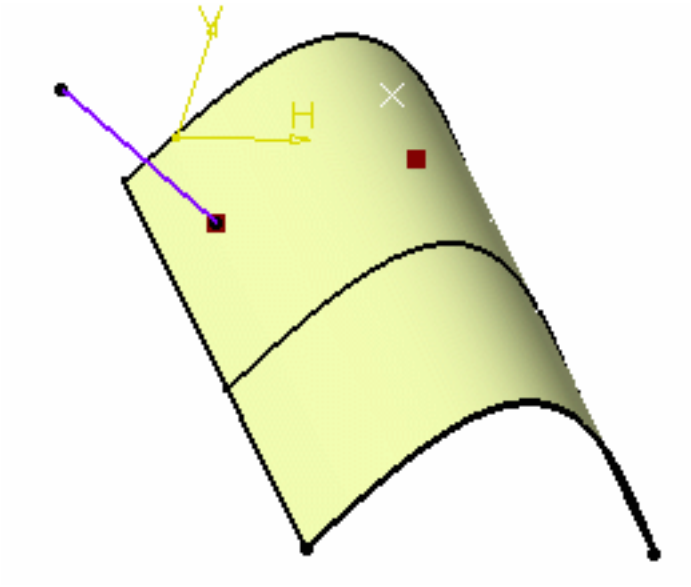
Normal to surface

- Select a reference **Surface** and a **Point**.
  A vector normal to the surface is displayed at the reference point.
  Proposed **Start** and **End** points of the new line are shown.



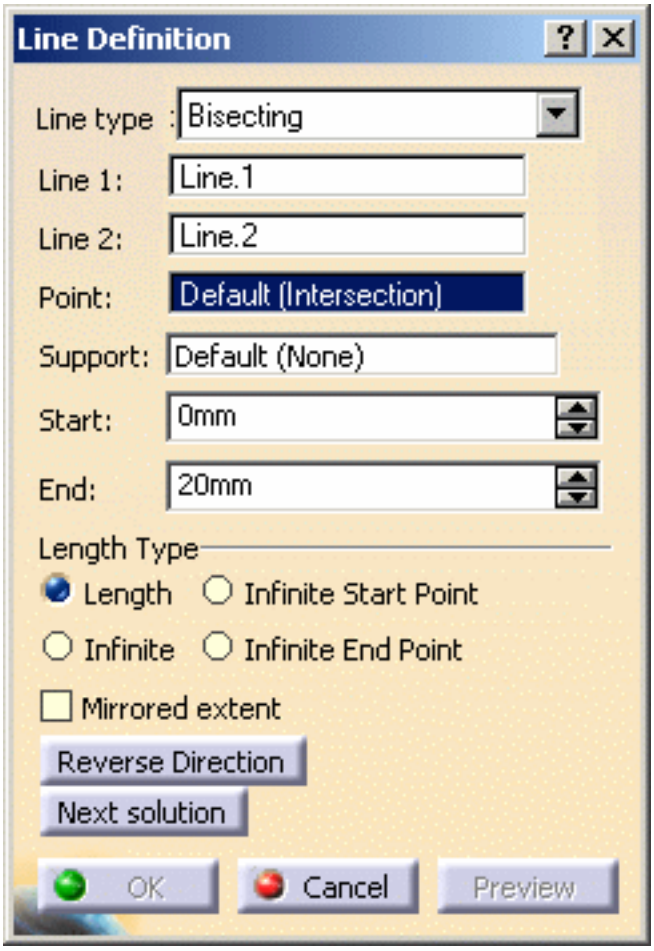


- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.

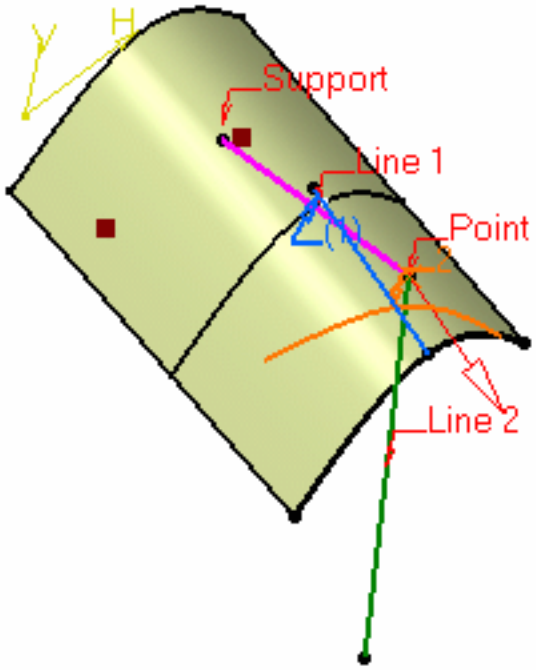


Bisecting

- Select two lines. Their bisecting line is the line splitting in two equals parts the angle between these two lines.
- Select a point as the starting point for the line. By default it is the intersection of the bisecting line and the first selected line.



- Select the support surface onto which the bisecting line is to be projected, if needed.
- Specify the line's length in relation to its starting point (**Start** and **End** values for each side of the line in relation to the default end points). The corresponding bisecting line, is displayed.
- You can choose between two solutions, using the **Next Solution** button, or directly clicking the numbered arrows in the geometry.



3. Click OK to create the line.

The line (identified as Line.xxx) is added to the specification tree.

- Regardless of the line type, **Start** and **End** values are specified by entering distance values or by using the graphic manipulators.
- Start and End values should not be the same.
- Select the Length Type:
  - **Length**: the line will be defined according to the **Start** and **End** points values
  - **Infinite**: the line will be infinite
  - **Infinite Start Point**: the line will be infinite from the **Start** point
  - **Infinite End Point**: the line will be infinite from the **End** point

By default, the Length type is selected.
The **Start** and/or the **End** points values will be greyed when one of the **Infinite** options is chosen.

- Check the **Mirrored extent** option to create a line symmetrically in relation to the selected **Start** point.
- In most cases, you can select a support on which the line is to be created. In this case, the selected point(s) is projected onto this support.
- You can reverse the direction of the line by either clicking the displayed vector or selecting the **Reverse Direction** button (not available with the point-point line type).

Creating an Axis

This task shows you how to create an axis feature.

Open the Axis1.CATPart document.

1. Click the **Axis** icon .

   The Axis Definition dialog box appears.

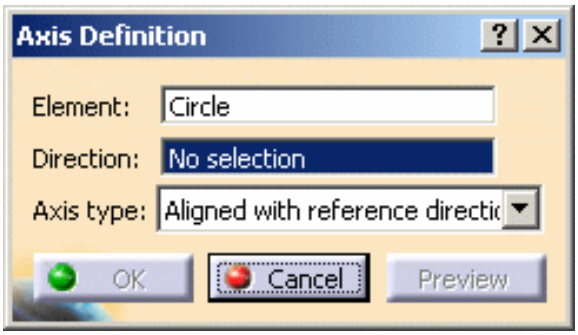
2. Select an **Element** where to create the axis.

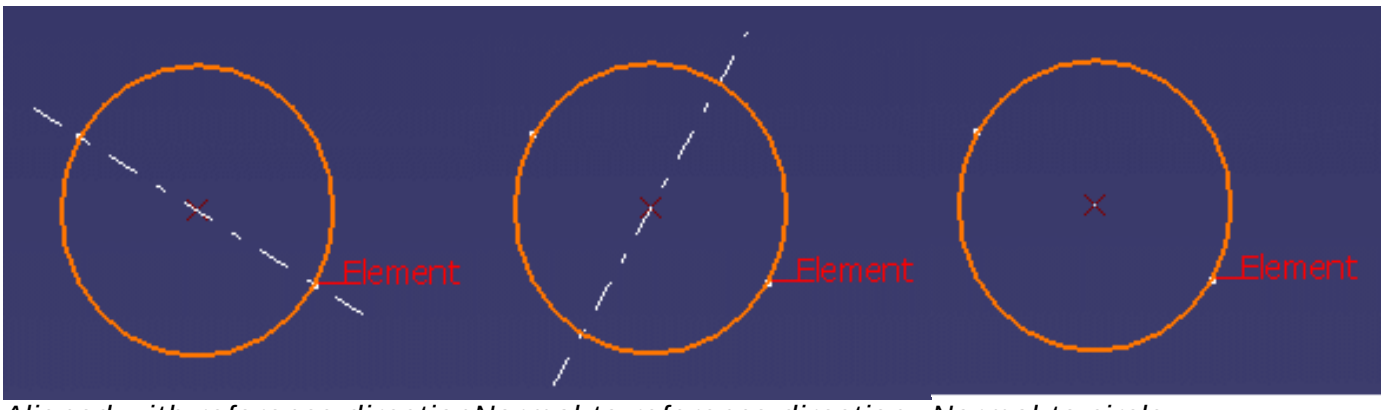
This element can be:

- a circle or a portion of circle
- an ellipse or a portion of ellipse
- an oblong curve
- a revolution surface or a portion of revolution surface

Circle

- Select the direction (here we chose the yz plane), when not normal to the surface.
- Select the axis type:
  - Aligned with reference direction
  - Normal to reference direction
  - Normal to circle

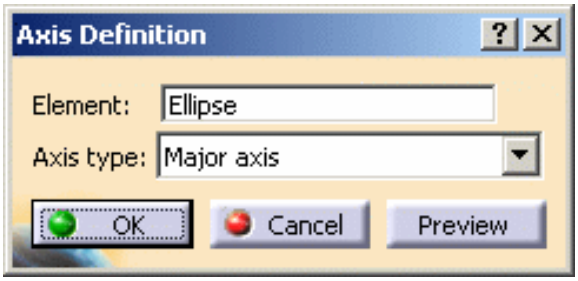


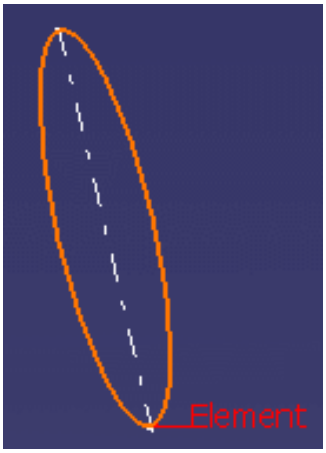


Aligned with reference direction *Normal to reference direction* *Normal to circle*

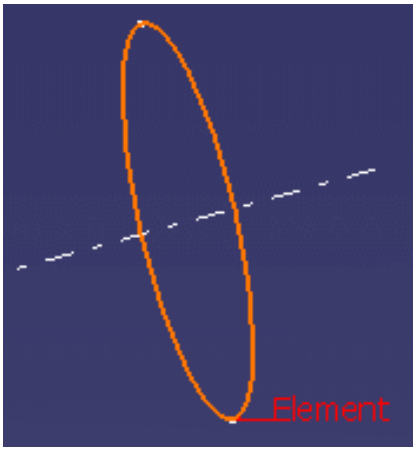
Ellipse

- Select the axis type:
  - Major axis
  - Minor axis
  - Normal to ellipse

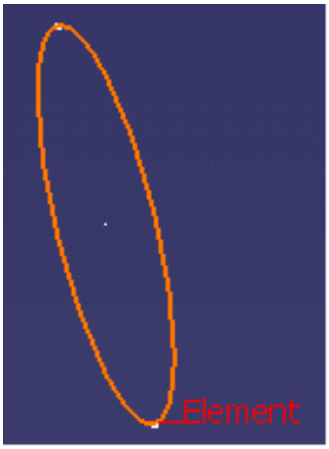




Major axis



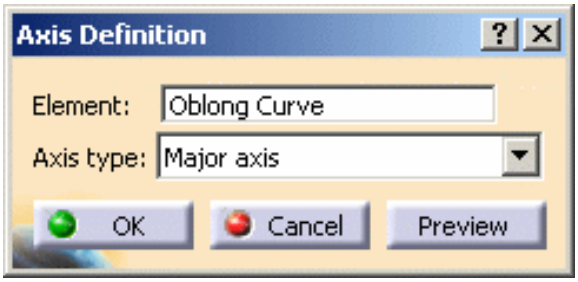
Minor axis

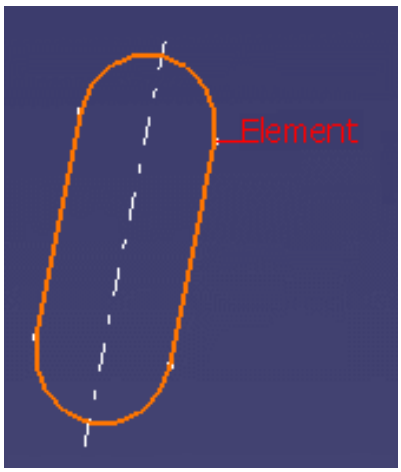


Normal to ellipse

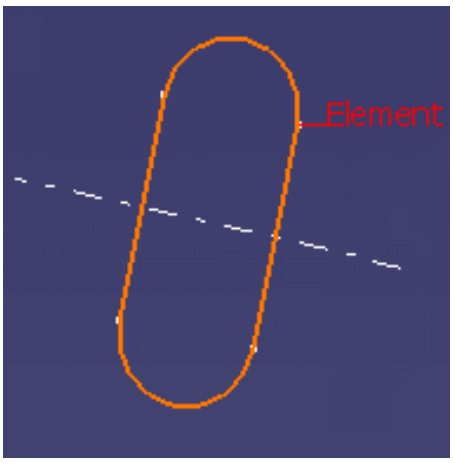
Oblong Curve

- Select the axis type:
  - Major axis
  - Minor axis
  - Normal to oblong

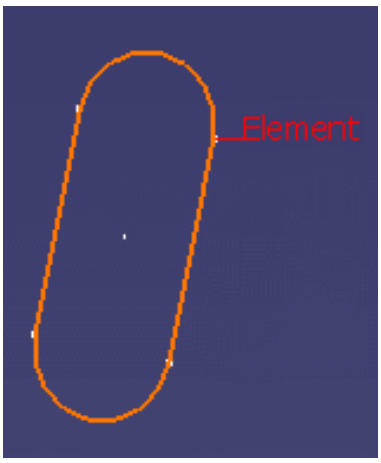




Major axis



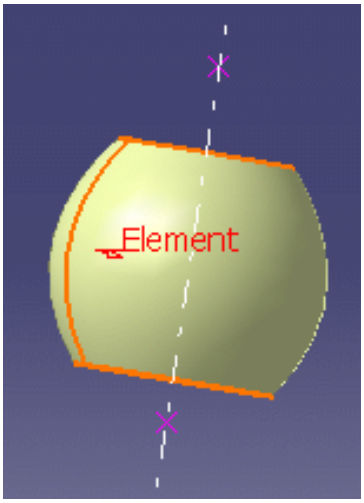
Minor axis

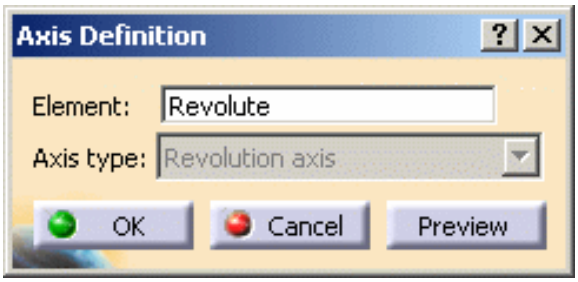


Normal to oblong

Revolution Surface

The revolution surface's axis is used, therefore the axis type combo list is disabled.





The axis can be displayed in the 3D geometry, either infinite or limited to the geometry block of the input element. This option is to be parameterized in **Tools -> Options -> Shape -> Generative Shape Design -> General**.
To have further information, please refer to the General Settings chapter in the *Customizing* section.

3. Click **OK** to create the axis.

The element (identified as Axis.xxx) is added to the specification tree.

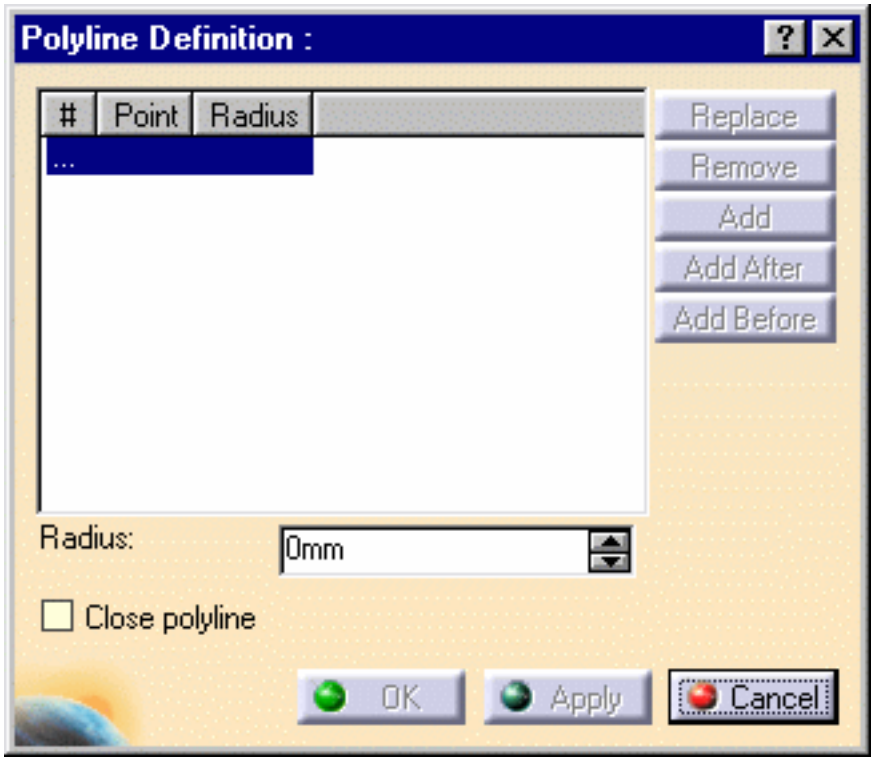
Creating Polylines

This task shows how to create a polyline, that is a broken line made of several connected segments.
These linear segments may be connected by blending radii.
Polylines may be useful to create cylindrical shapes such as pipes, for example.

Open the Spline1.CATPart document.

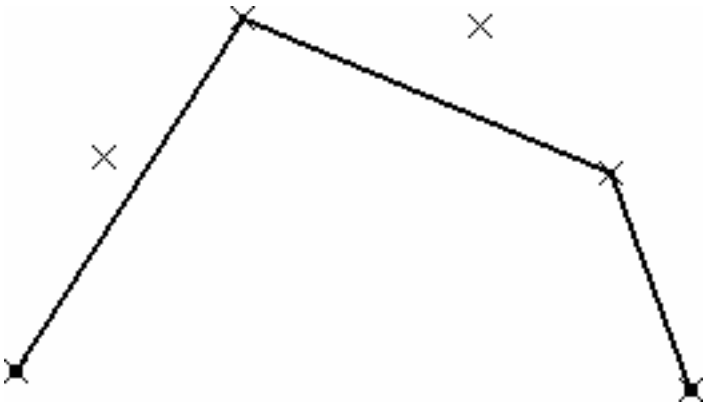
1. Click the **Polyline** icon .

   The Polyline Definition dialog box appears.



2. Select several points in a row.
   Here we selected Point.1, Point.5, Point.3 and Point.2 in this order.

   The resulting polyline would look like this:



3. From the dialog box, select Point.5, click the Add After button and select Point.6.
4. Select Point.3 and click the Remove button.

   The resulting polyline now looks like this:

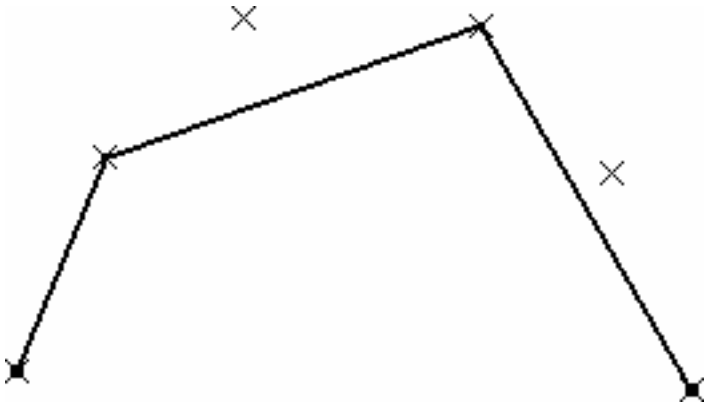


5. Still from the dialog box select Point.5, click the Replace button, and select Point.4 in the geometry.

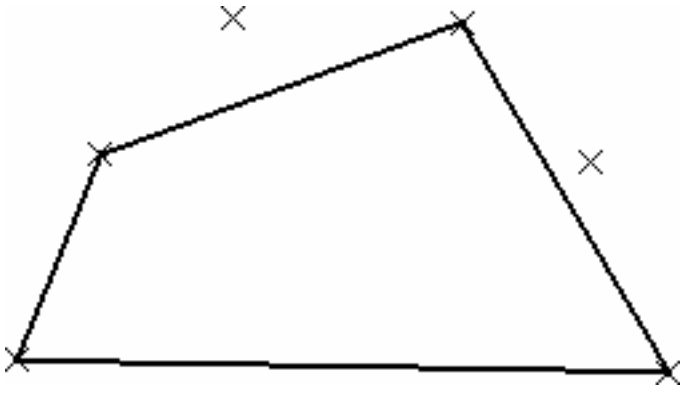
   The added point automatically becomes the current point in the dialog box.

6. Click OK in the dialog box to create the polyline.

The element (identified as Polyline. xxx) is added to the specification tree.



- The polyline's orientation depends on the selection order of the points.
- You can re-order selected points using the Replace, Remove, Add, Add After, and Add Before buttons.



- You cannot select twice the same point to create a polyline. However, you can check the Close polyline button to generate a closed contour.

Back Up Next

Creating Planes

This task shows the various methods for creating planes:

- offset from a plane
- parallel through point
- angle/normal to a plane
- through three points
- through two lines
- through a point and a line
- through a planar curve
- normal to a curve
- tangent to a surface
- from its equation
- mean through points

Open the Planes1.CATPart document.

1. Click the **Plane** icon .

   The Plane Definition dialog box appears.

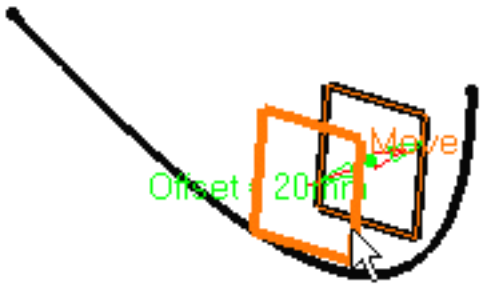
2. Use the combo to choose the desired **Plane type**.

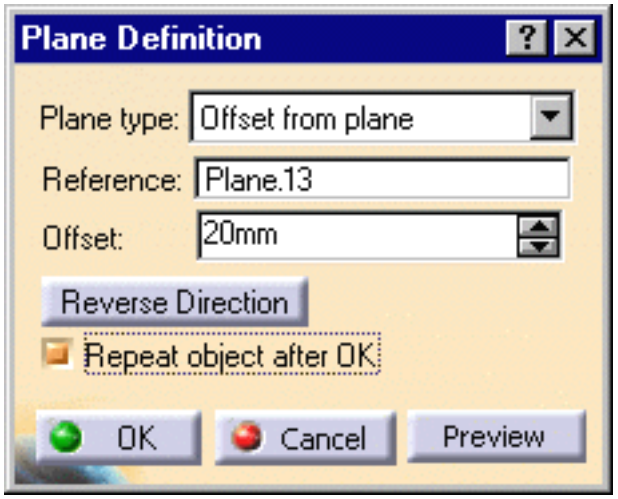
Once you have defined the plane, it is represented by a red square symbol, which you can move using the graphic manipulator.

Offset from plane

- Select a reference **Plane** then enter an **Offset** value.

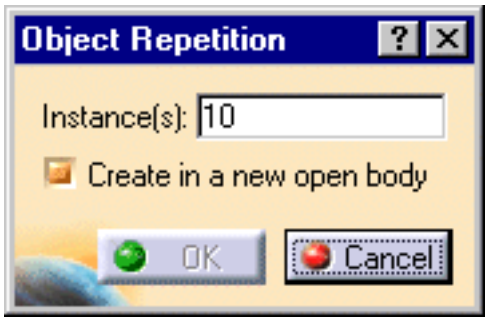
A plane is displayed offset from the reference plane.



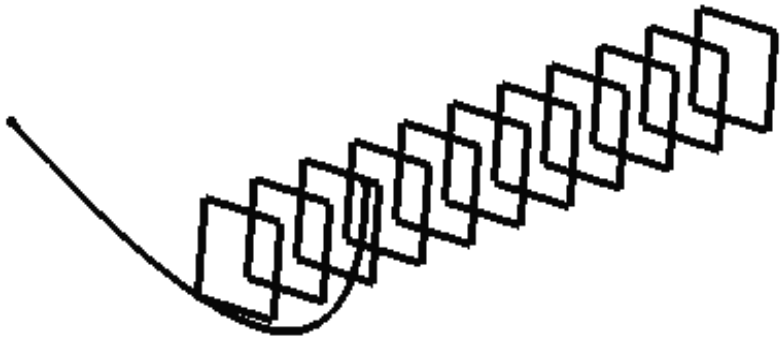


Use the **Reverse Direction** button to reverse the change the offset direction, or simply click on the arrow in the geometry.

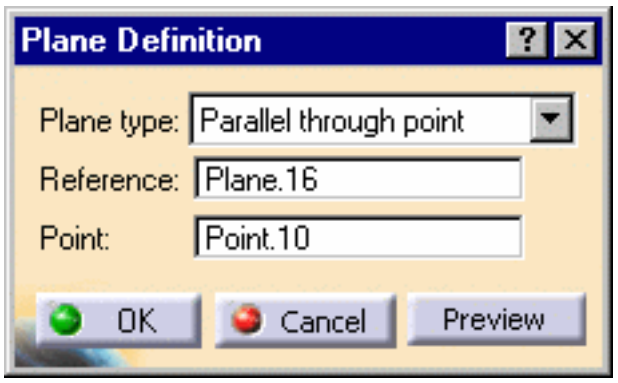
- Click the **Repeat object after OK** if you wish to create more offset planes .
  In this case, the **Object Repetition** dialog box is displayed, and you key in the number of instances to be created before pressing OK.



As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Offset** value.



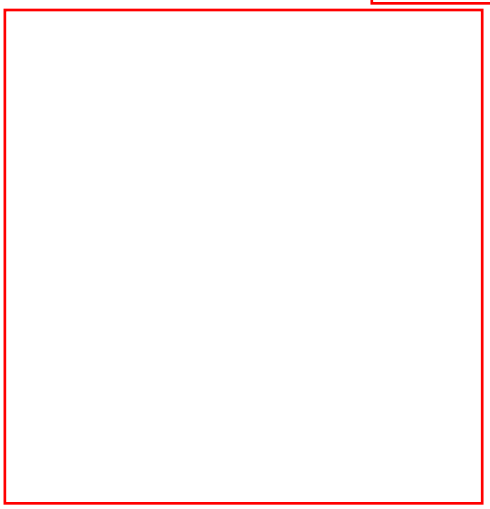
Parallel through point



- Select a reference **Plane** and a **Point**.

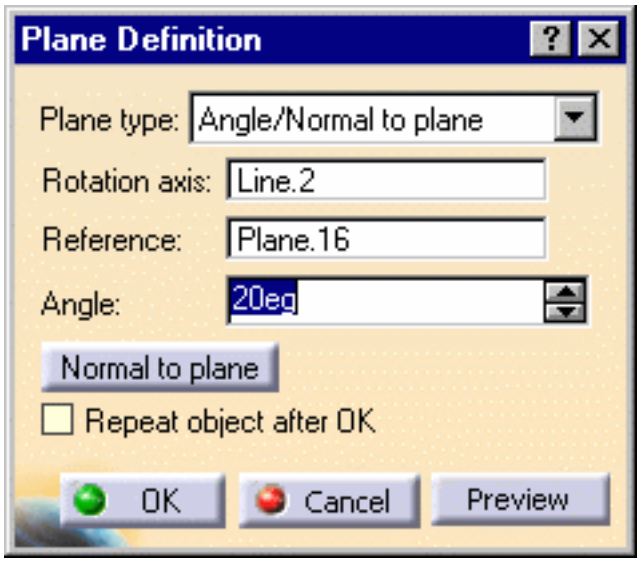
A plane is displayed parallel to the reference plane and passing through the selected point.



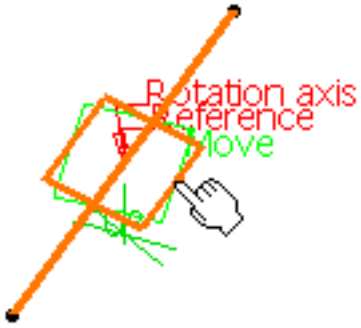


Angle or normal to plane

- Select a reference **Plane** and a **Rotation axis**.
  This axis can be any line or an implicit element, such as a cylinder axis for example. To select the latter press and hold the Shift key while moving the pointer over the element, then click it.
- Enter an **Angle** value.



A plane is displayed passing through the rotation axis. It is oriented at the specified angle to the reference plane.



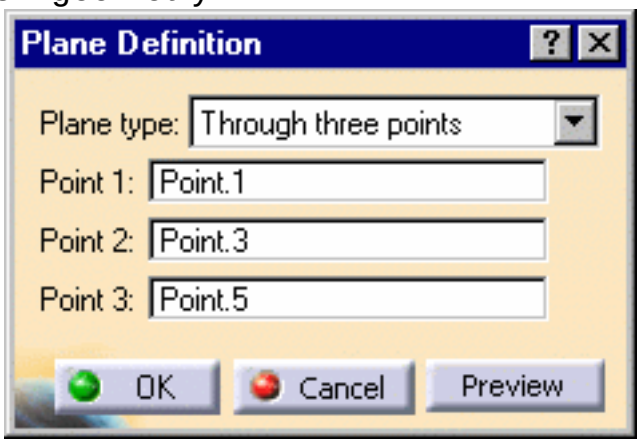
- Click the **Repeat object after OK** if you wish to create more planes at an angle from the initial plane.
  In this case, the **Object Repetition** dialog box is displayed, and you key in the number of instances to be created before pressing OK.

  As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Angle** value.

  Here we created five planes at an angle of 20 degrees.

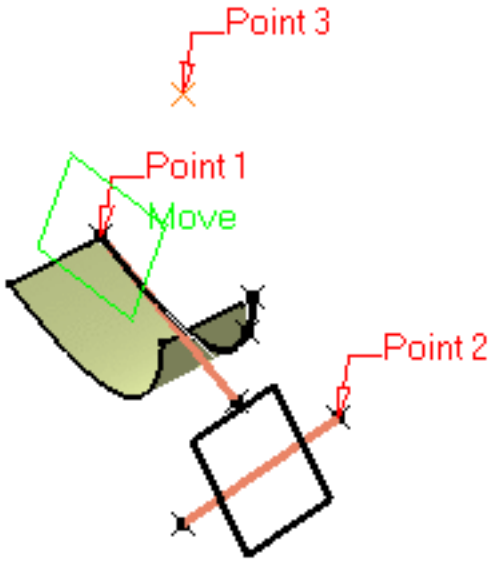


This plane type enables to edit the plane's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

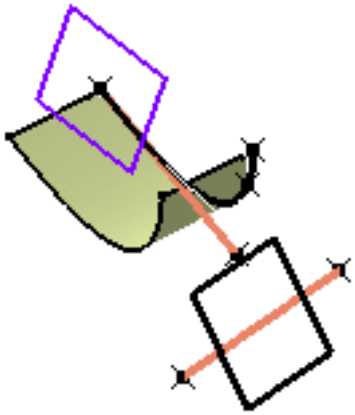


Through three points

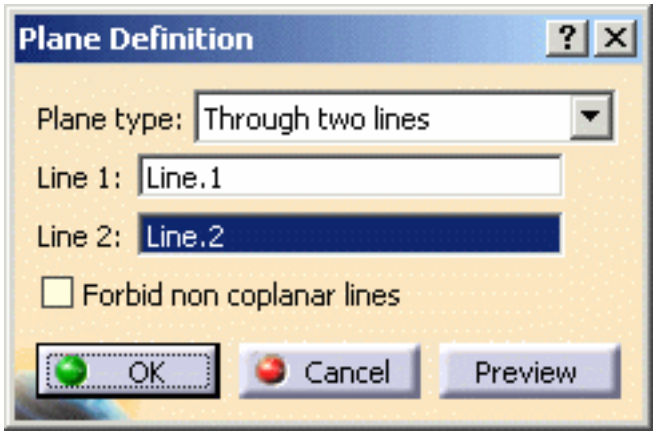
- Select three points.



The plane passing through the three points is displayed. You can move it simply by dragging it to the desired location.

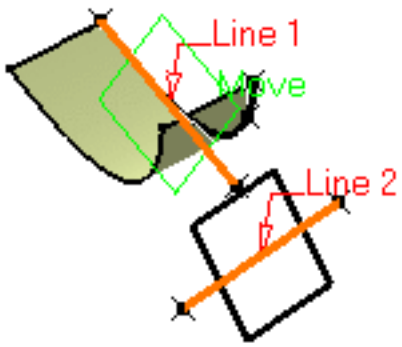


Through two lines

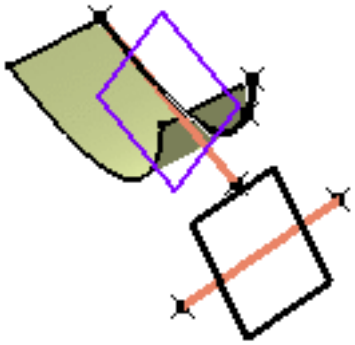


- Select two lines.

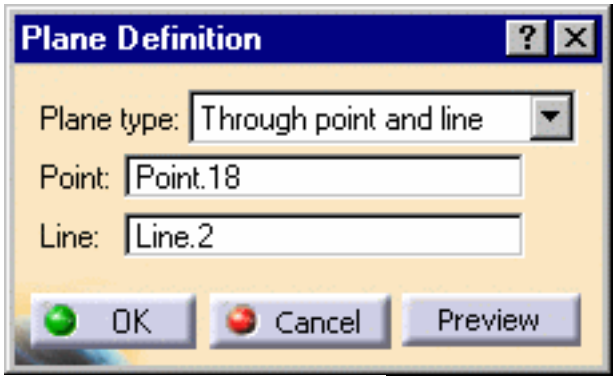
The plane passing through the two line directions is displayed. When these two lines are not coplanar, the vector of the second line is moved to the first line location to define the plane's second direction.



Check the **Forbid non coplanar lines button** to specify that both lines be in the same plane.

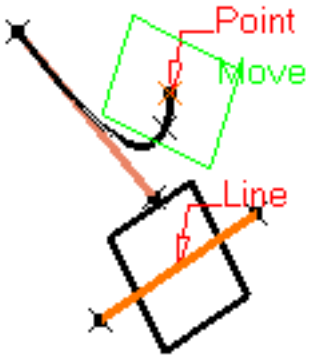


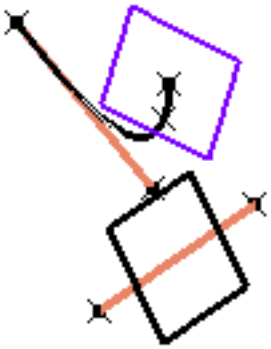
Through point and line



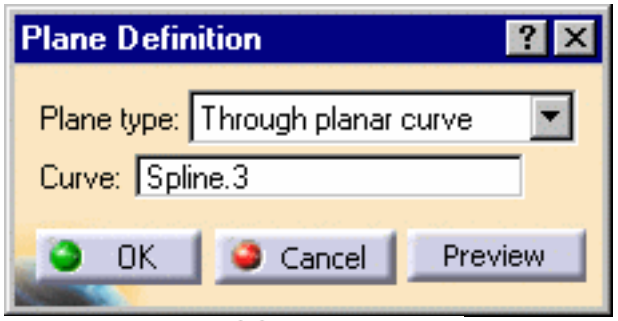
- Select a **Point** and a **Line**.

The plane passing through the point and the line is displayed.

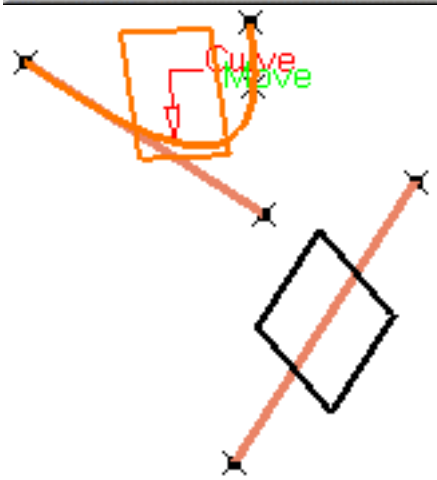




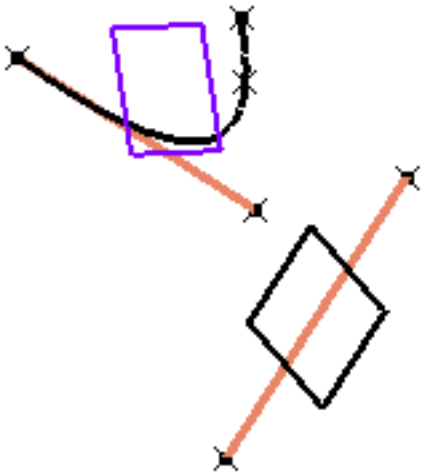
Through planar curve



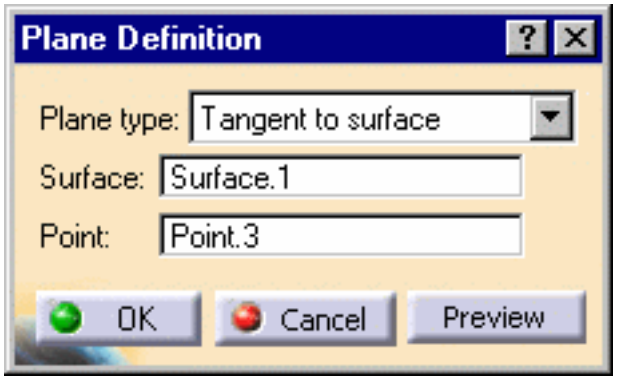
- Select a planar **Curve**.



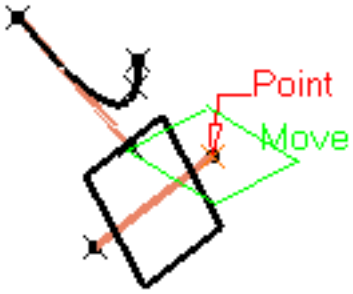
The plane containing the curve is displayed.



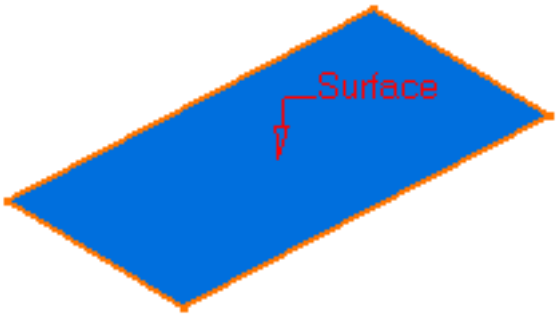
Tangent to surface

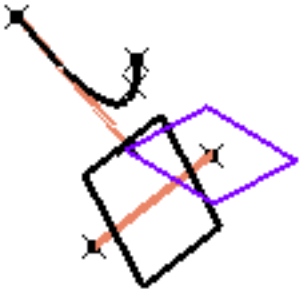


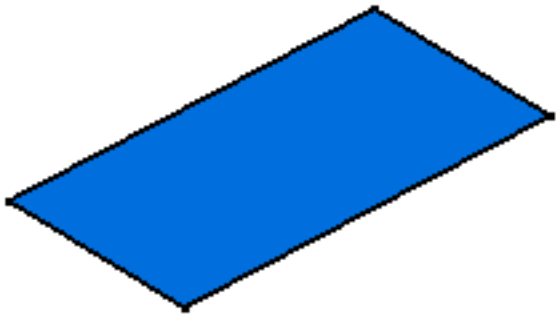
- Select a reference **Surface** and a **Point**.



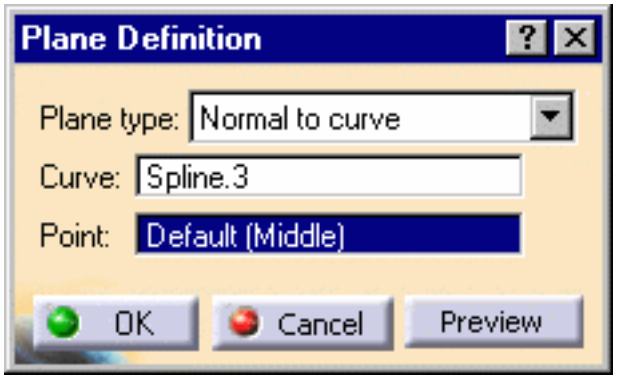
A plane is displayed tangent to the surface at the specified point.





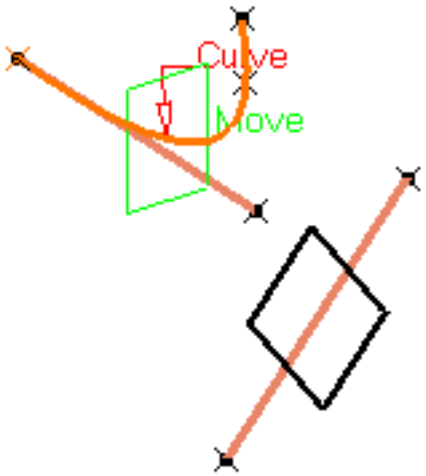


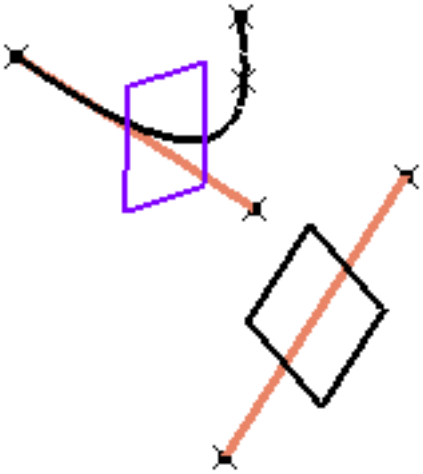
Normal to curve



- Select a reference **Curve.**
- You can select a **Point**. By default, the curve's middle point is selecte.

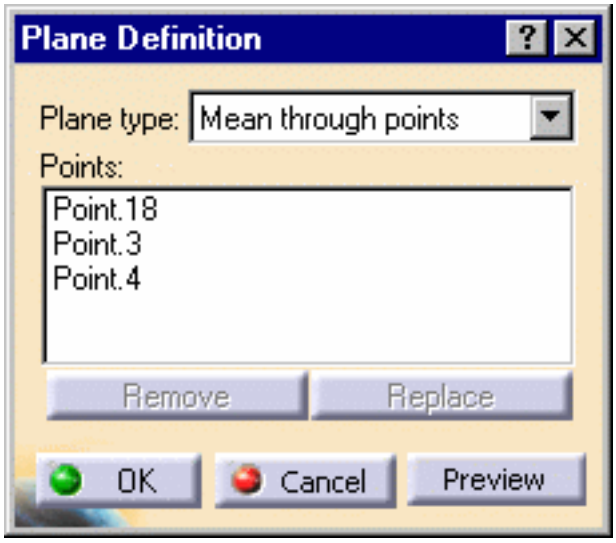
A plane is displayed normal to the curve at the specified point.





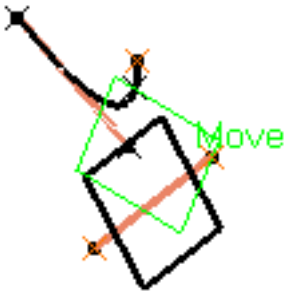
Mean through points

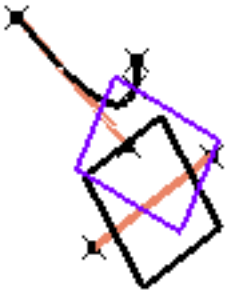
- Select three or more points to display the mean plane through these points.



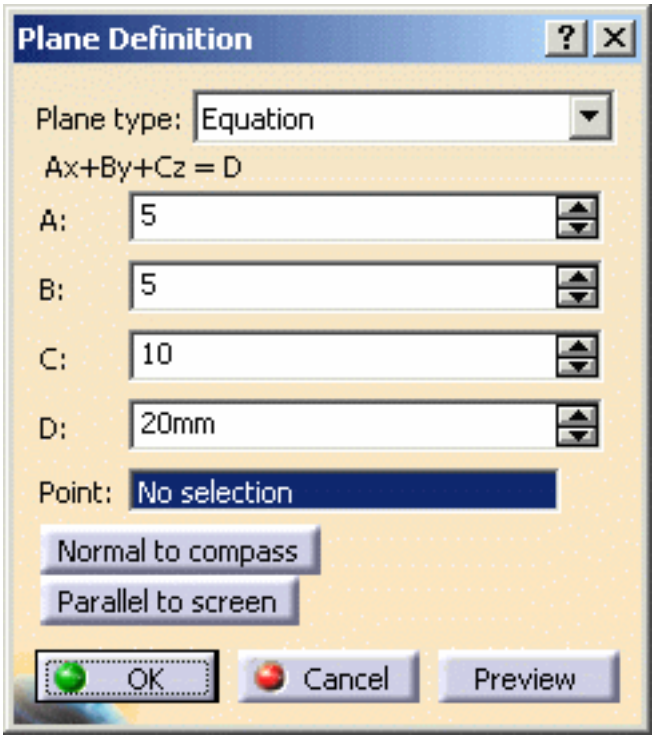
It is possible to edit the plane by first selecting a point in the dialog box list then choosing an option to either:

- **Remove** the selected point
- **Replace** the selected point by another point.





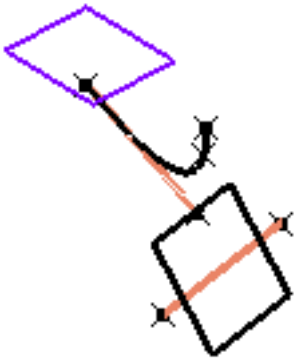
Equation



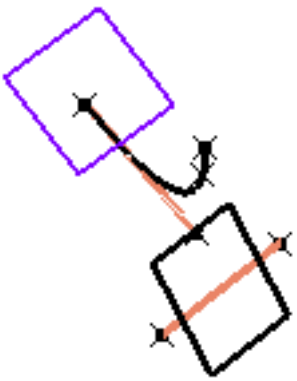
- Enter the **A**, **B**, **C**, **D** components of the **Ax + By + Cz = D** plane equation.

Select a point to position the plane through this point, you are able to modify **A**, **B**, and **C** components, the **D** component becomes grayed.

Use the **Normal to compass** button to position the plane perpendicular to the compass direction.



Use the **Parallel to screen** button to parallel to the screen current view.



3. Click **OK** to create the plane.

The plane (identified as Plane.xxx) is added to the specification tree.



Creating Planes Between Other Planes

This task shows how to create any number of planes between two existing planes, in only one operation:

Open the Planes1.CATPart document.

1. Click the **Planes Repetition** icon .

   The Planes Between dialog box appears.



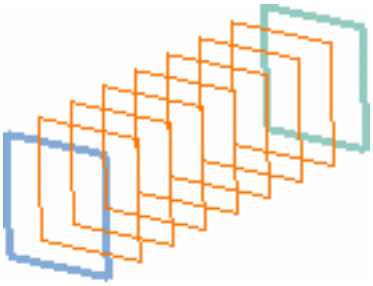
2. Select the two planes between which the new planes must be created.



3. Specify the number of planes to be created between the two selected planes.

4. Click **OK** to create the planes.

The planes (identified as Plane.xxx) are added to the specification tree.



Check the **Create in a new Open Body** button to create a new Open Body containing only the repeated planes.

Back Up Next

Creating Circles

This task shows the various methods for creating circles and circular arcs:

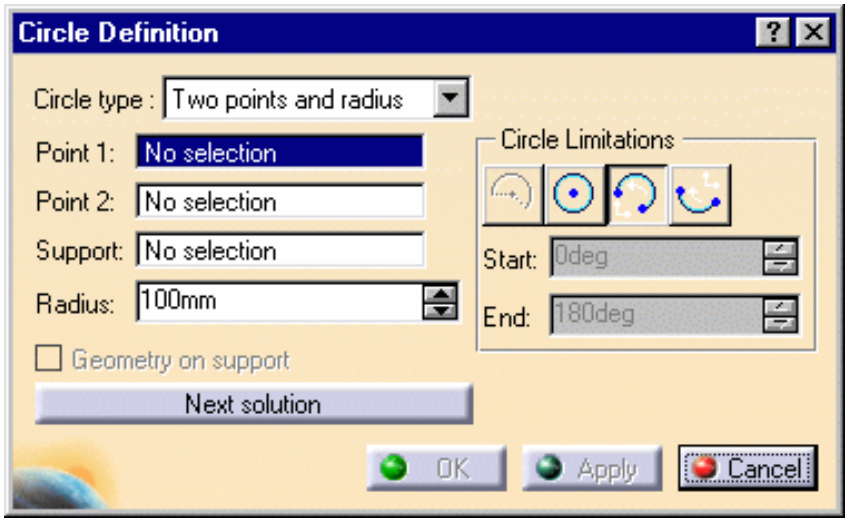
- center and radius
- center and point
- two points and radius
- three points
- bitangent and radius
- bitangent and point
- tritangent
- center and tangent

Open the Circles1.CATPart document.

1. Click the **Circle** icon .

   The Circle Definition dialog box appears.

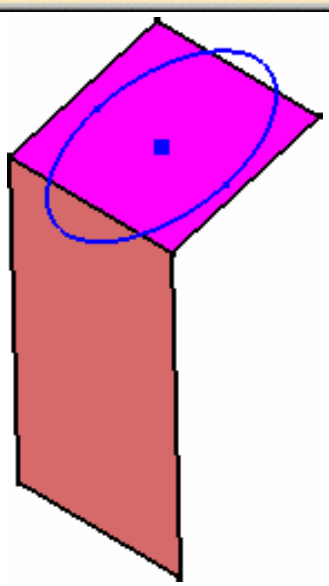
2. Use the combo to choose the desired circle type.



Center and radius

- Select a point as circle **Center**.
- Select the **Support** plane or surface where the circle is to be created.
- Enter a **Radius** value.

Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the **Start** and **End** angles of the arc.

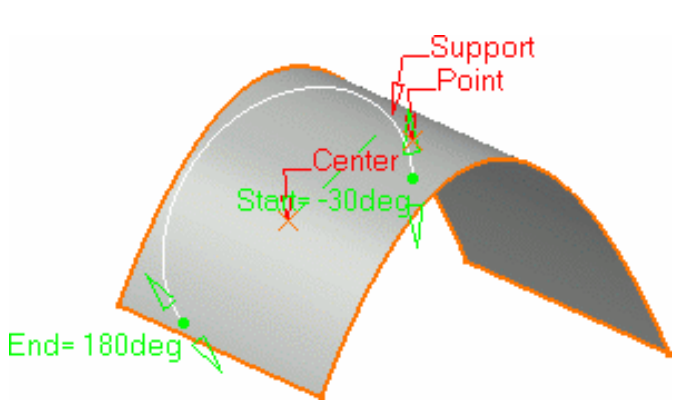


If a support surface is selected, the circle lies on the plane tangent to the surface at the selected point.

Start and **End** angles can be specified by entering values or by using the graphic manipulators.

Center and point

- Select a point as **Circle** center.
- Select a **Point** where the circle is to be created.
- Select the **Support** plane or surface where the circle is to be created.

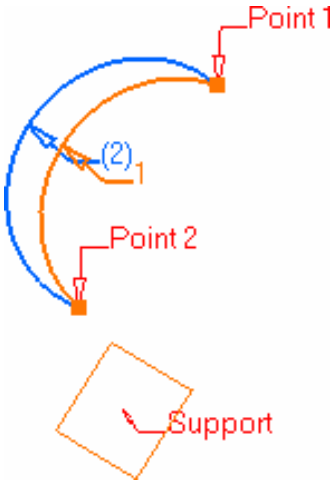


The circle, which center is the first selected point and passing through the second point or the projection of this second point on the plane tangent to the surface at the first point, is previewed.

Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the **Start** and **End** angles of the arc.

Two points and radius

- Select two points on a surface or in the same plane.
- Select the **Support** plane or surface.
- Enter a **Radius** value.

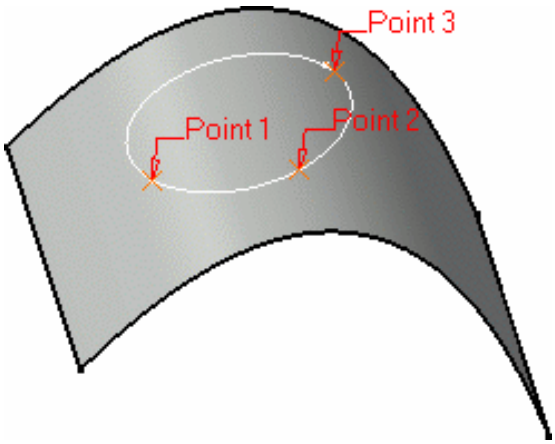


The circle, passing through the first selected point and the second point or the projection of this second point on the plane tangent to the surface at the first point, is previewed.

Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the trimmed or complementary arc using the two selected points as end points.

You can use the **Second Solution** button, to display the alternative arc.

Three points



- Select three points where the circle is to be created.

Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the trimmed or complementary arc using the two of the selected points as end points.

3. In each of the methods above, you can select the **Geometry on Support** check box if you want the circle to be projected onto a support surface.

   In this case just select a support surface.

Bi-tangent and radius

- Select two **Elements** (point or curve) to which the circle is to be tangent.
- Select a **Support** surface.

If one of the selected inputs is a planar curve, then the **Support** is set to Default (Plane).
If an explicit **Support** needs to be defined, a contextual menu is available to clear the selection in order to select the desired support.

This automatic support definition saves you from performing useless selections.

- Enter a **Radius** value.
- Several solutions may be possible, so click in the region where you want the circle to be.

Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the trimmed or complementary arc using the two tangent points as end points.

Bi-tangent and point

- Select a point or a curve to which the circle is to be tangent.
- Select a **Curve** and a **Point** on this curve.
- Select a **Support** plane or planar surface.

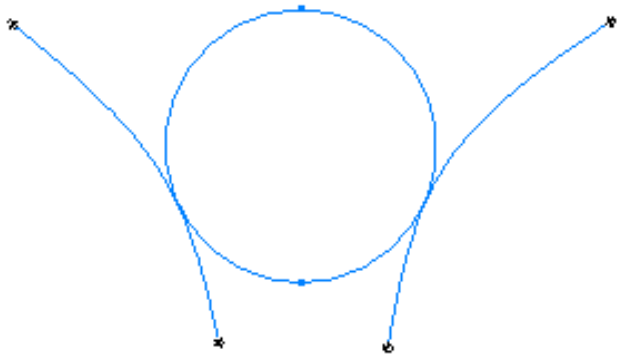
The point will be projected onto the curve.

If one of the selected inputs is a planar curve, then the **Support** is set to Default (Plane).
If an explicit **Support** needs to be defined, a contextual menu is available to clear the selection in order to select the desired support.

This automatic support definition saves you from performing useless selections.

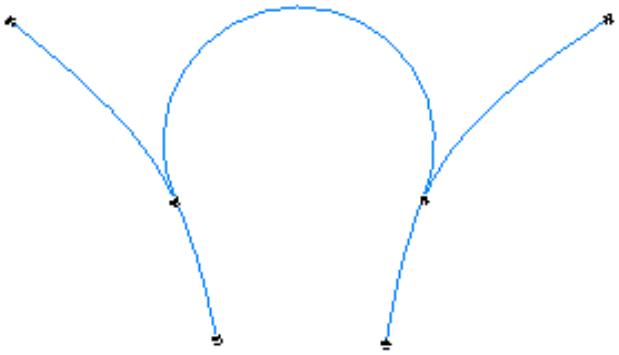
- Several solutions may be possible, so click in the region where you want the circle to be.

  Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.

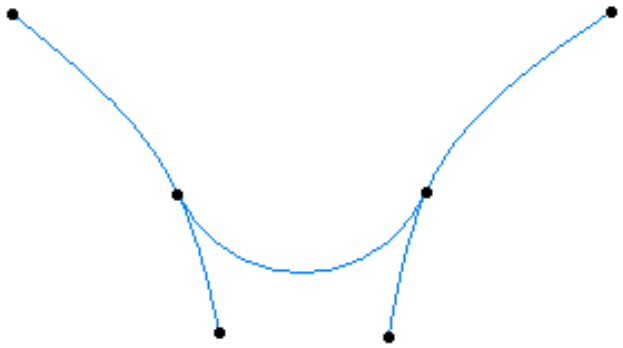


Complete circle

For a circular arc, you can choose the trimmed or complementary arc using the two tangent points as end points.

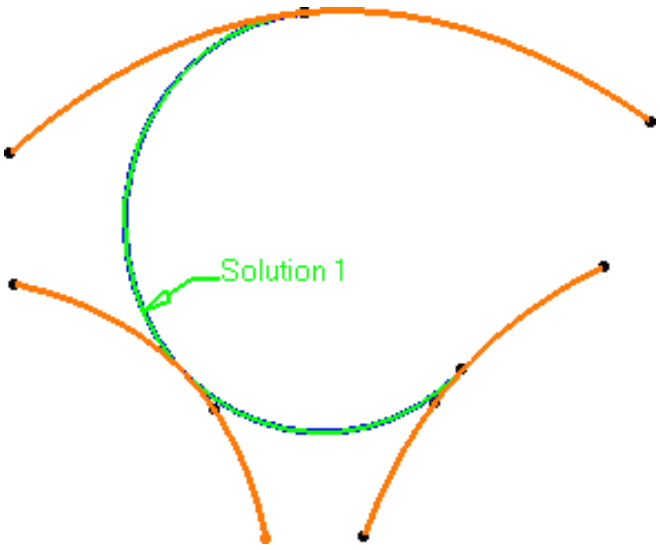


Trimmed circle



Complementary trimmed circle

Tri-tangent



- Select three **Elements** to which the circle is to be tangent.
- Select a **Support** planar surface.

If one of the selected inputs is a planar curve, then the **Support** is set to Default (Plane).
If an explicit **Support** needs to be defined, a contextual menu is available to clear the selection in order to select the desired support.
This automatic support definition saves you from performing useless selections.

- Several solutions may be possible, so select the arc of circle that you wish to create.

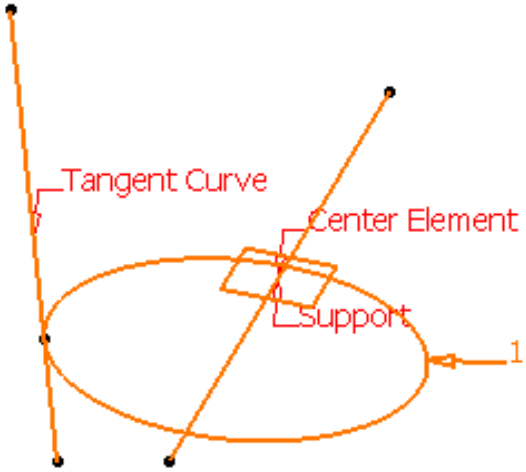
Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed. The first and third elements define where the relimitation ends.
For a circular arc, you can specify the trimmed or complementary arc using the two tangent points as end points.

Center and tangent

There are two ways to create a center and tangent circle:

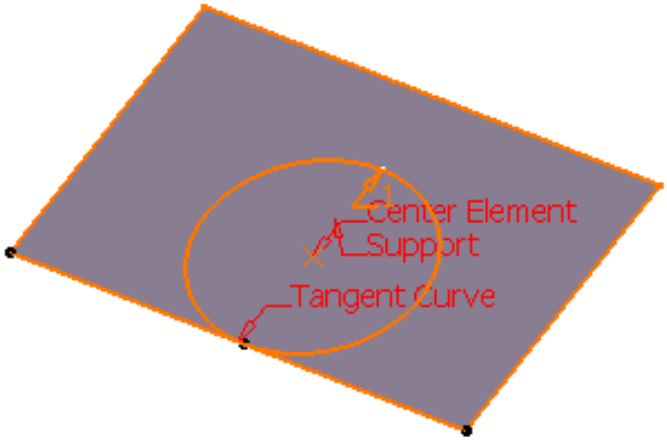
1. Center curve and radius

- Select a curve as the **Center Element**.
- Select a **Tangent Curve**.
- Enter a **Radius** value.



2. Line tangent to curve definition

- Select a point as the **Center Element**.
- Select a **Tangent Curve**.



If one of the selected inputs is a planar curve, then the **Support** is set to Default (Plane).
If an explicit **Support** needs to be defined, a contextual menu is available to clear the selection in order to select the desired support.

This automatic support definition saves you from performing useless selections.

The circle center will be located either on the center curve or point and will be tangent to tangent curve.

Please note that only full circles can be created.

4. Click **OK** to create the circle or circular arc.

The circle (identified as Circle.xxx) is added to the specification tree.

When several solutions are possible, click the **Next Solution** button to move to another arc of circle, or directly select the arc you want in the 3D geometry.

Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.



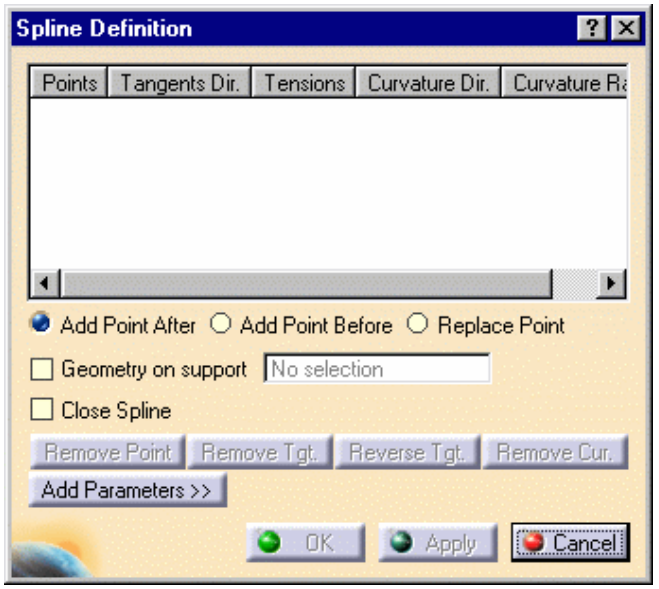
Creating Splines

This task shows the various methods for creating spline curves.

Open the Spline1.CATPart document.

1. Click the Spline icon .

   The Spline Definition dialog box appears.



2. Select two or more points where the spline is to pass.

   An updated spline is visualized each time a point is selected.

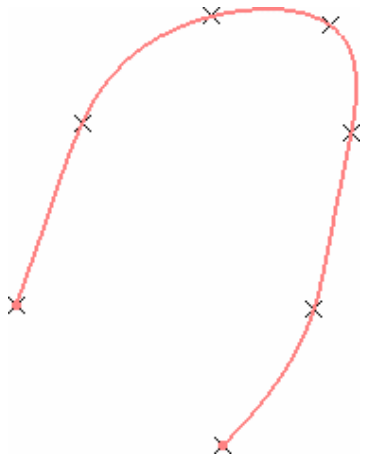
3. It is possible to edit the spline by first selecting a point in the dialog box list then choosing a button to either:

   - Add a point after the selected point
   - Add a point before the selected point
   - Remove the selected point
   - Replace the selected point by another point.

4. You can select the **Geometry on support** check box, and select a support (plane, surface), if you want the spline to be projected onto a support surface.

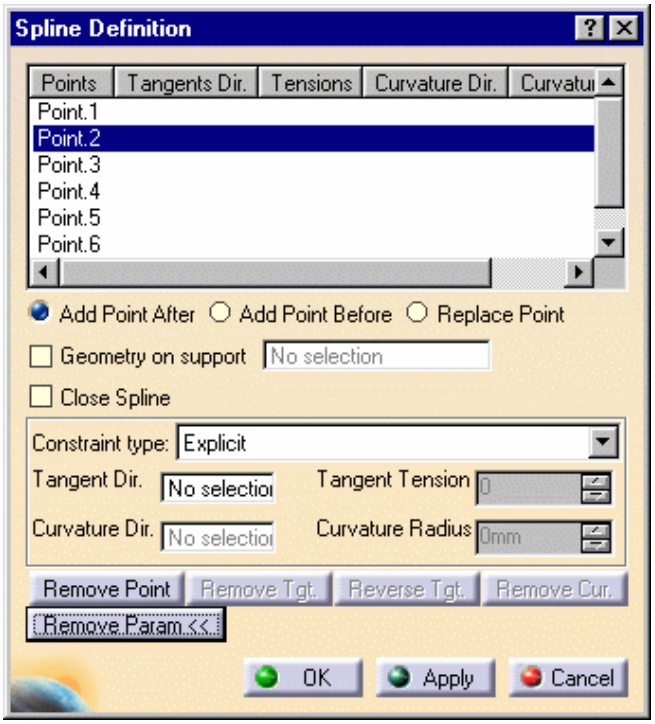
   It is better when the tangent directions belong to the support, that is when a projection is possible.

   In this case just select a surface or plane.



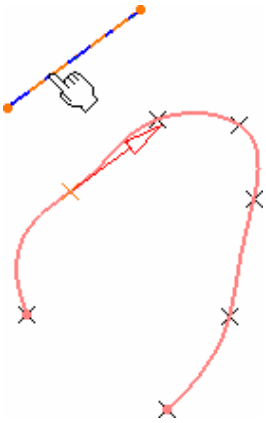
In the figure above, the spline was created on a planar support grid.

5. Click on the Add Parameter button to display further options.
6. To set tangency conditions onto any point of the spline, select the point and click on **Tangent Dir.**

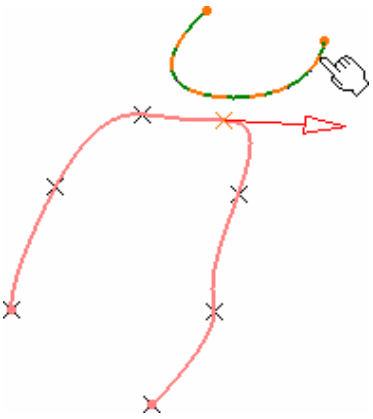


There are two ways of imposing tangency and curvature constraints:

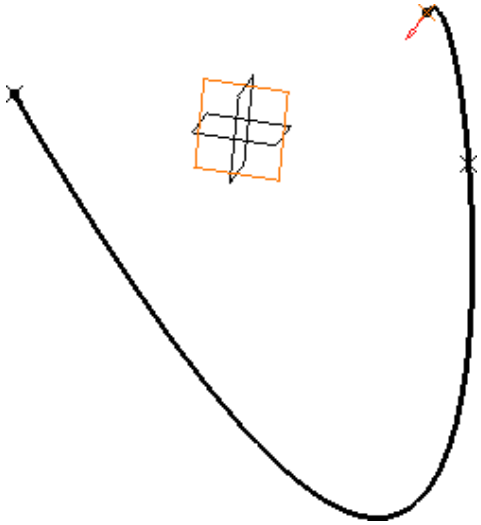
1. **Explicit**: select a line or plane to which the tangent on the spline is parallel at the selected point



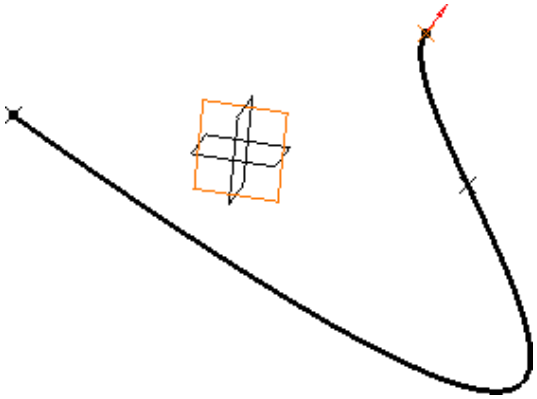
2. **From curve**: select a curve to which the spline is tangent at the selected point.



Use the **Remove Tgt.**, **Reverse Tgt.**, or **Remove Cur.** to manage the different imposed tangency and curvature constraints.



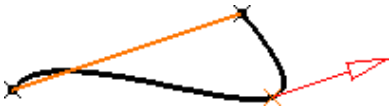
Spline with a tangency constraint on endpoint (tension = 2)



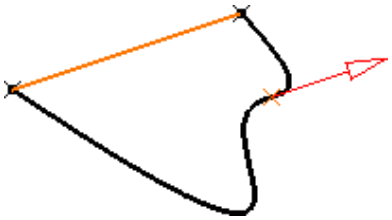
Spline with reversed tangent

7. To specify a curvature constraint at any point of the spline, once a tangency constraint has been set, indicate a curvature direction and enter a radius value:

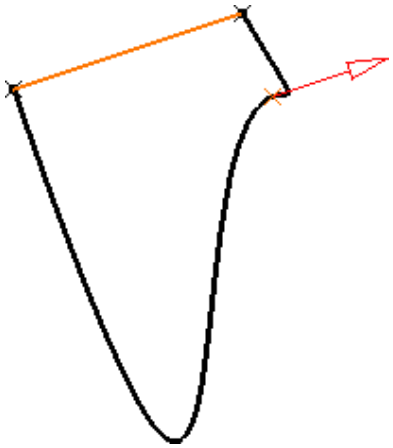
The curvature direction is projected onto a plane normal to the tangent direction.
If you use the **Create line** contextual menu, and want to select the same point as a point already used to define the tangent direction, you may have to select it from the specification tree, or use the pre-selection navigator.



Spline with tangency constraint



Spline with tangency constraint and curvature constraint (radius = 50mm)



Spline with tangency constraint and curvature constraint (radius = 2mm)

Note that for the Points Specifications, you must enter your information in the following order:

- **Tangent Dir.** (tangent direction)
- **Tangent Tension**
- **Curvature Dir.** (curvature direction)
- **Curvature Radius** (to select it, just click in the field).

The fields become active as you select values.

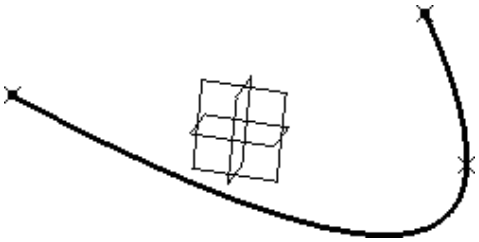
8. Click OK to create the spline.

The spline (identified as Spline.xxx) is added to the specification tree.

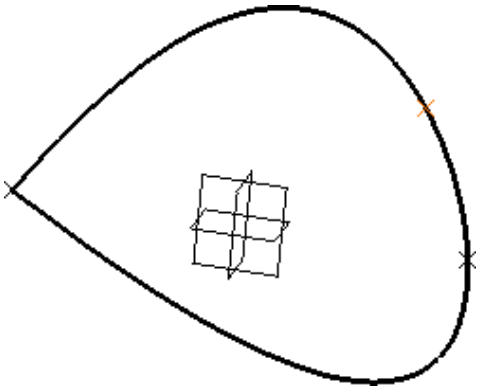
- To add a parameter to a point, select a line in the Points list. This list is highlighted.
  You have two possibilities:

  1. extended parameters
  2. select any line or plane for the direction.

- Use the **Close Spline** option to create a closed curve, provided the geometric configuration allows it



Spline with Close Spline option unchecked



Spline with Close Spline option checked

Back Up Next

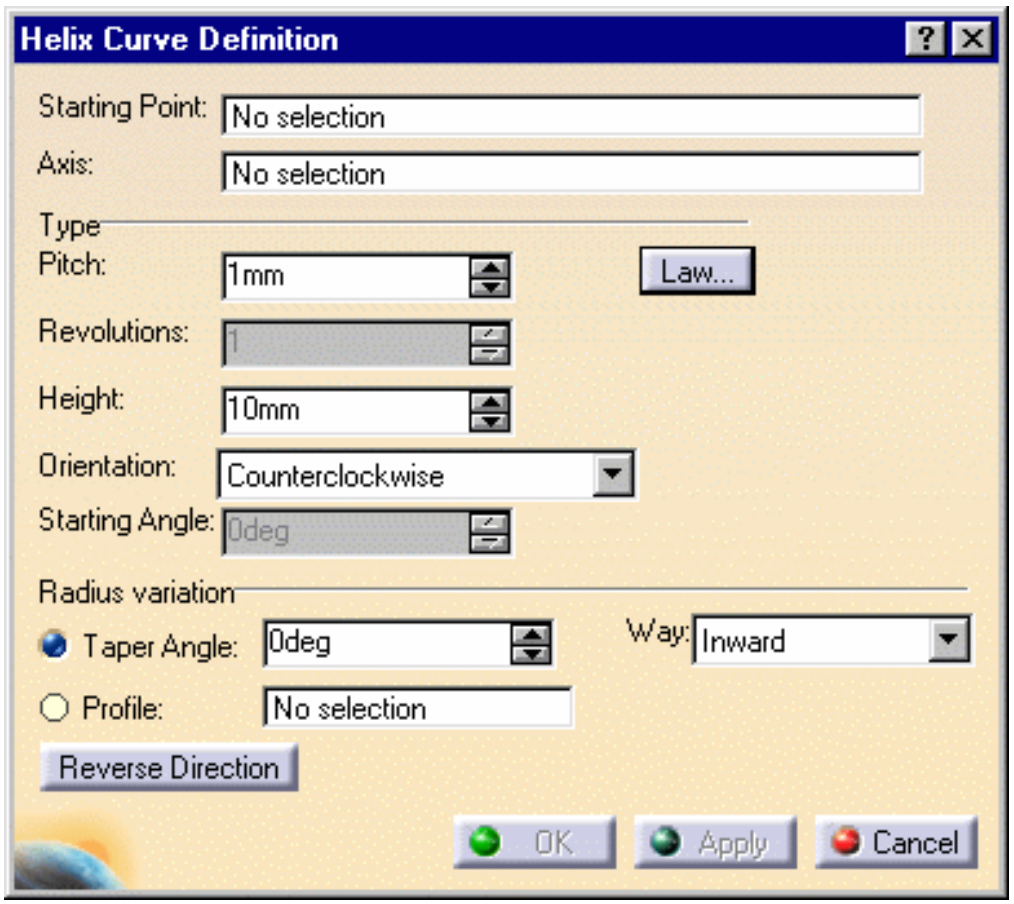
Creating a Helix

This task shows the various methods for creating helical 3D curves, such as coils and springs for example.

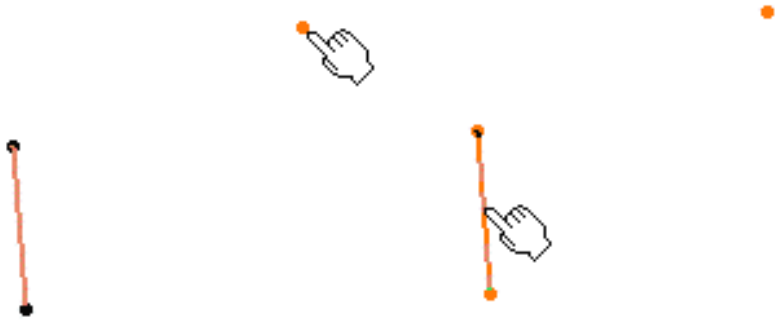
Open the Helix1.CATPart document.

1. Click the Helix icon .

   The Helix Curve Definition dialog box appears.

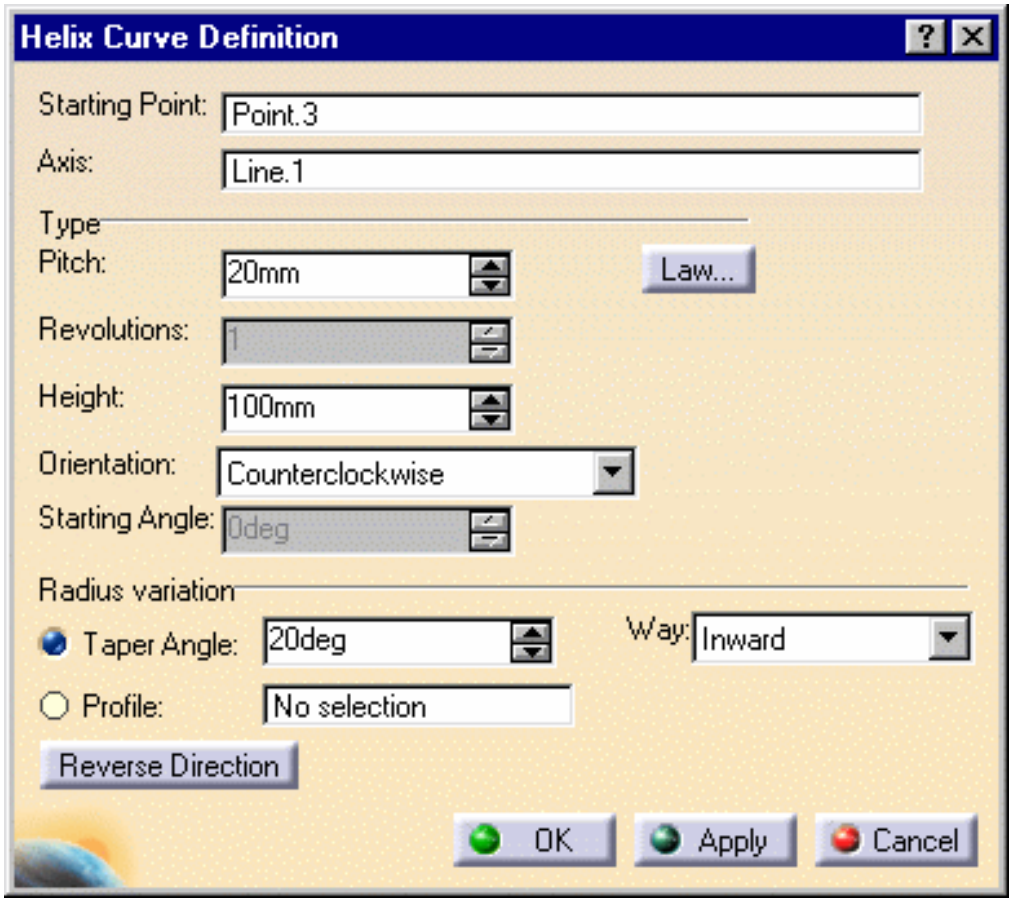


2. Select a starting point and an axis.



3. Set the helix parameters:

- **Pitch**: the distance between two revolutions of the curve



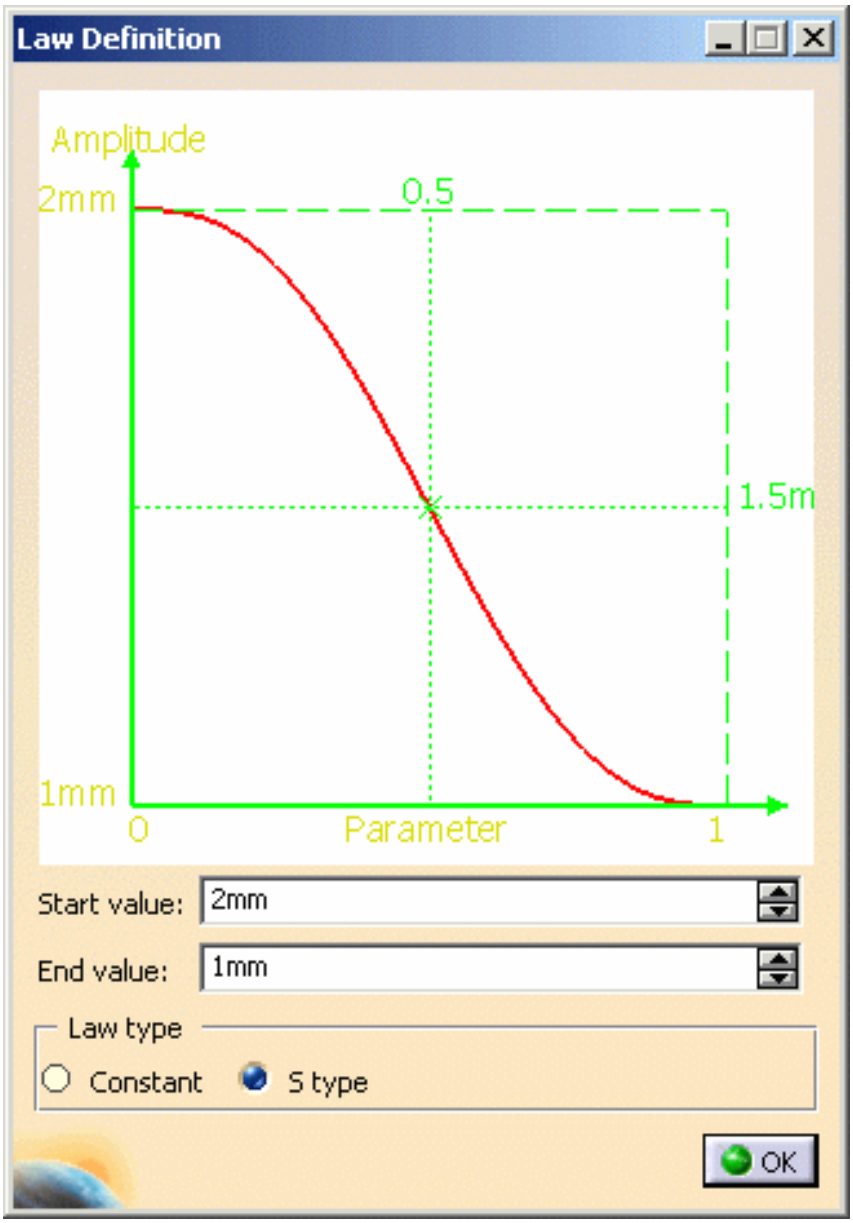
You can define the evolution of the pitch along the helix using a law.

1. Click the Law button to display the Law Definition dialog box.
2. Choose type of law to be applied to the pitch: It can stay **Constant**, or evolve according to a **S type** law.

   For the S type pitch, you need to define a second pitch value. The pitch distance will vary between these two pitch values, over the specified number of revolutions.

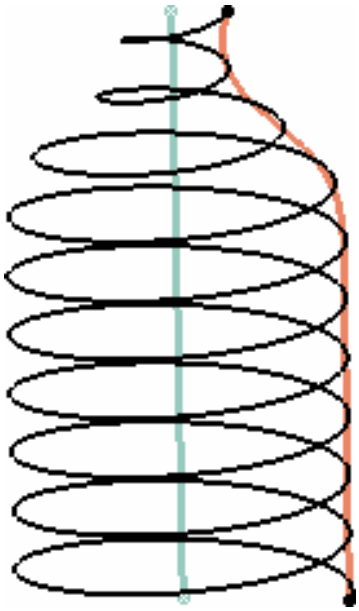
3. The Law Viewer allows you to:
   - visualize the law evolution and the maximum and minimum values,
   - navigate into the viewer by panning and zooming (using to the mouse),
   - trace the law coordinates by using the manipulator,
   - change the viewer size by changing the panel size
   - reframe on by using the viewer contextual menu
   - change the law evaluation step by using the viewer contextual menu (from 0.1 (10 evaluations) to 0.001 (1000 evaluations)).

4. Click OK to return to the Helix Curve Definition dialog box.



- **Height**: the global height of the helical curve, in the case of a constant pitch type helix
- **Orientation**: defines the rotation direction (clockwise or counter clockwise)
- **Starting Angle**: defines where the helical curve starts, in relation to the starting point. This parameter can be set only for the **Constant** pitch only.
- **Taper Angle**: the radius variation from one revolution to the other. It ranges from -90° to 90° excluded.
  For a constant radius, set the taper angle to 0.
- **Way**: defines the taper angle orientation.
  Inward: the radius decreases
  Outward: the radius increases.

- **Profile**: the curve used to control the helical curve radius variation. The radius evolves according to the distance between the axis and the selected profile (here the orange curve).
  Note that the **Starting point** must be on the profile.

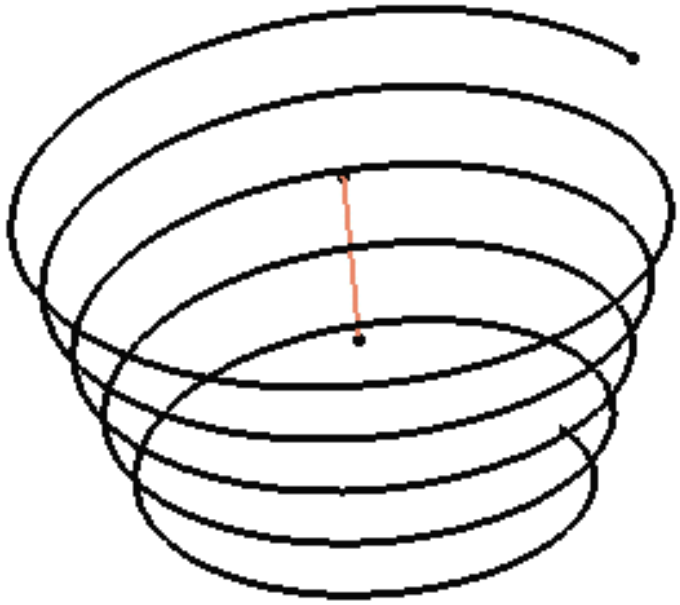


- **Starting Angle**: defines where the helical curve starts, in relation to the starting point. This parameter can be set only for the **Constant** pitch only.

4. Click the Reverse Direction button to invert the curve direction.

5. Click OK to create the helix.

   The helical curve (identified as Helix.xxx) is added to the specification tree.



Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.

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Creating Corners

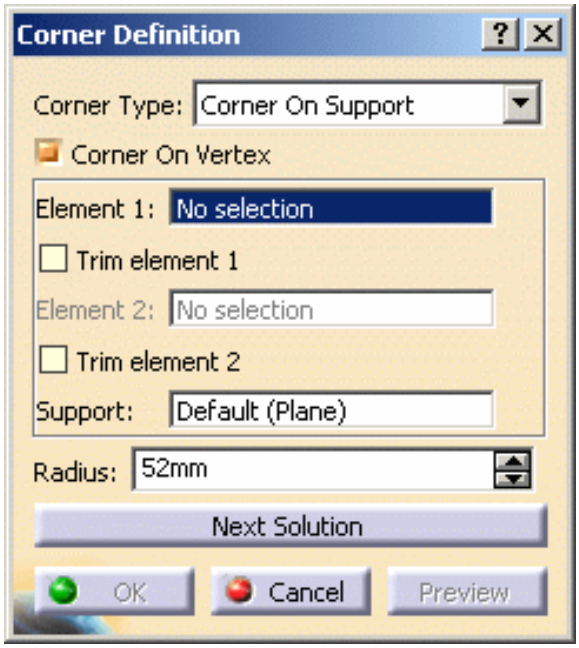
This task shows you how to create a corner between two curves or between a point and a curve.

Open the Corner1.CATPart document.

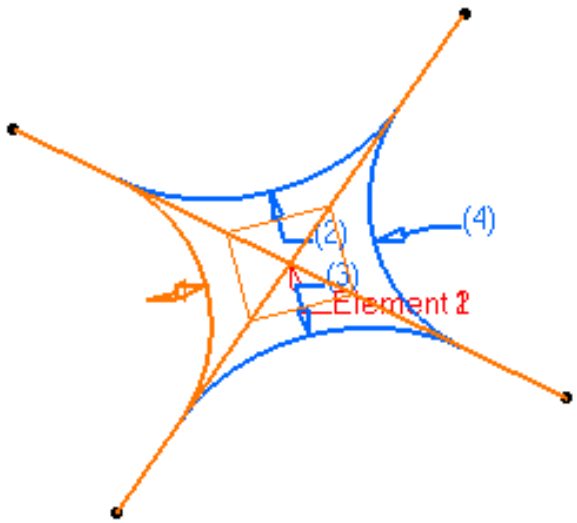
1. Click the **Corner** icon .

   The Corner Definition dialog box appears.

2. Select a curve or a point as first reference element.
3. Select a curve as second reference element.
   The corner will be created between these two references.
4. Select the **Support** surface. It can be a surface or a plane. Here we selected the zx plane.



The resulting corner is a curve seen as an arc of circle lying on a support place or surface.

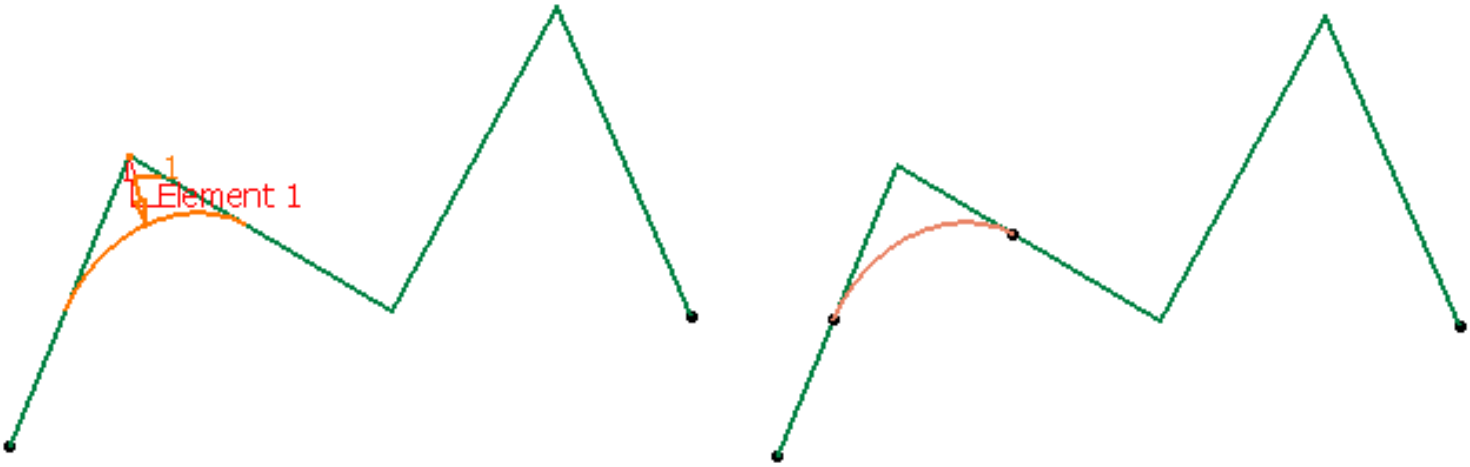


The reference elements must lie on this support, as well as the center of the circle defining the corner.

5. Enter a **Radius** value.

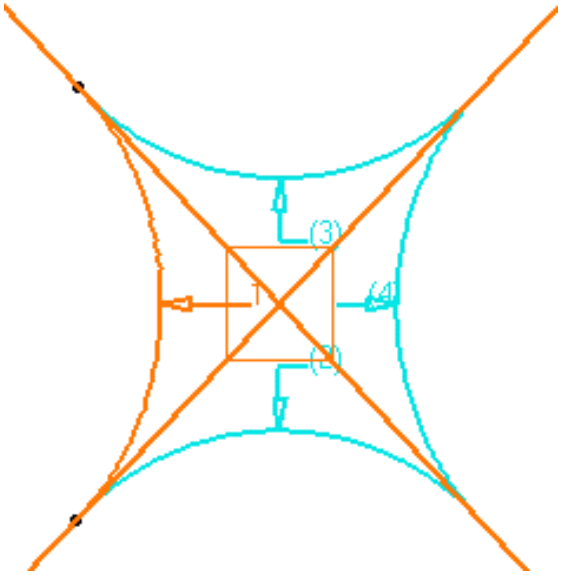
The **Corner On Vertex** check box enables you to create a corner by selecting a point or a curve as Element 1 (Element 2 is grayed as well as the Trim Element 1 and 2 options).

6. Select the **Support** surface.

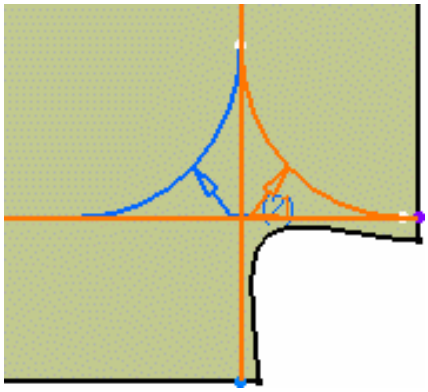


The example above shows a corner defined by a point as Element 1

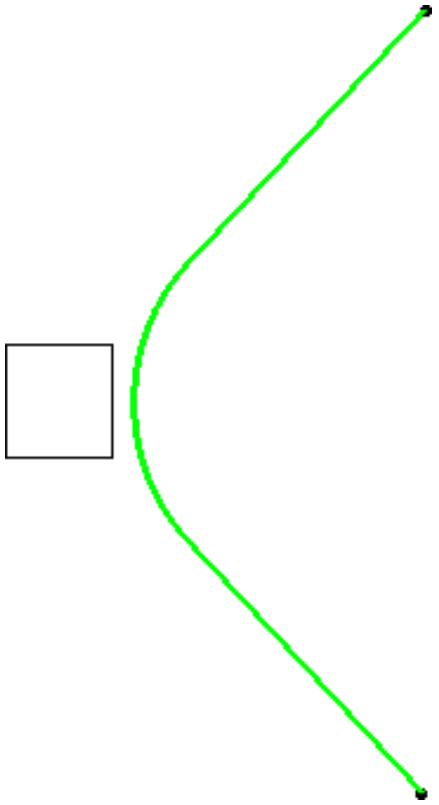
7. Several solutions may be possible, so click the **Next Solution** button to move to another corner solution, or directly select the corner you want in the geometry.



Not all four solutions are always available, depending on the support configuration (if the center of one of the corners does not lie on the support for example).



8. You can select the **Trim elements** check box if you want to trim and assemble the two reference elements to the corner.



The elements can be trimmed and assembled individually.

9. Click OK to create the corner.

The corner (identified as Corner.xxx) is added to the specification tree.

When the selected curves are coplanar, the default support is the background plane. However, you can explicitly select any support.
You can edit the rotated element's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

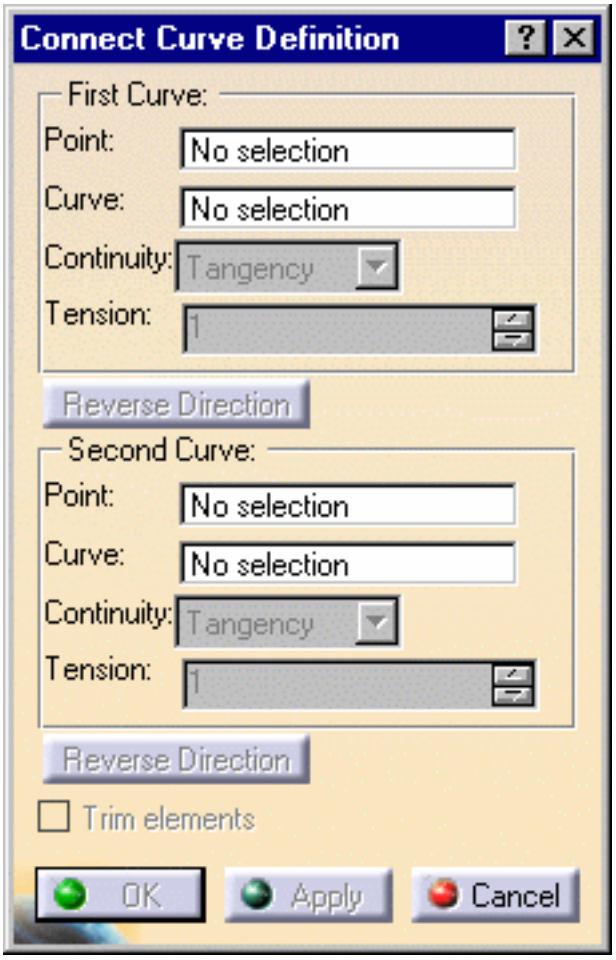
Back Up Next

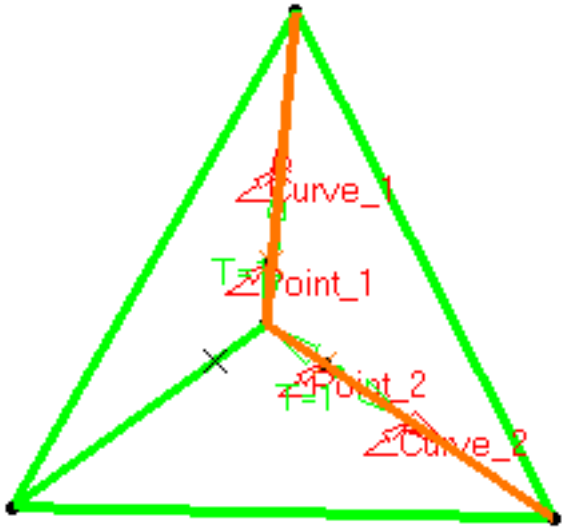
Creating Connect Curves

This task shows how to create connecting curves between two existing curves.

Open the Connect1.CATPart document.

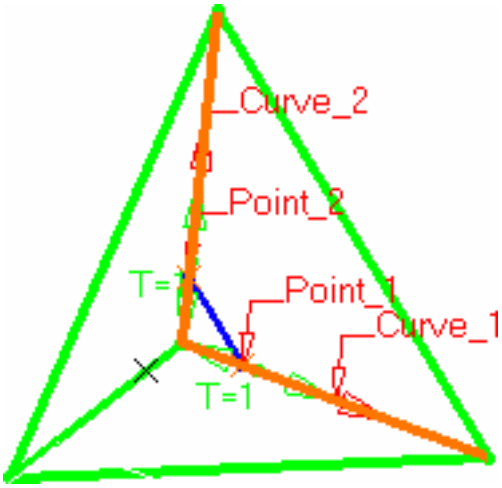
1. Click the **Connect Curve** icon .
   The Connect Curve Definition dialog box appears.
2. Select a first **Point** on a curve then a second **Point** on a second curve.
   The **Curve** fields are automatically filled.



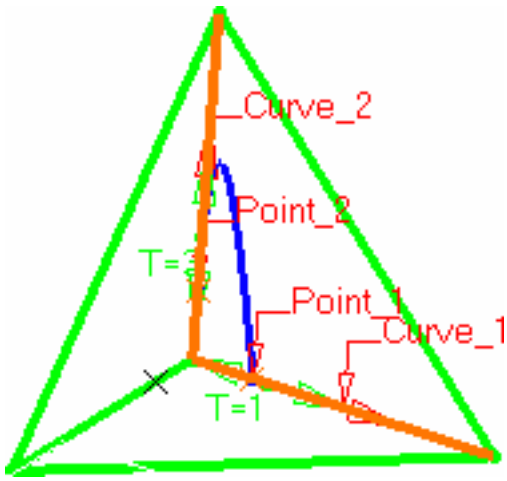


3. Use the combos to specify the desired **Continuity** type: **Point**, **Tangency** or **Curvature**.
4. If needed, enter tension values in the example below the tension is set to 3 whenever we illustrate a curvature or tangency continuity).

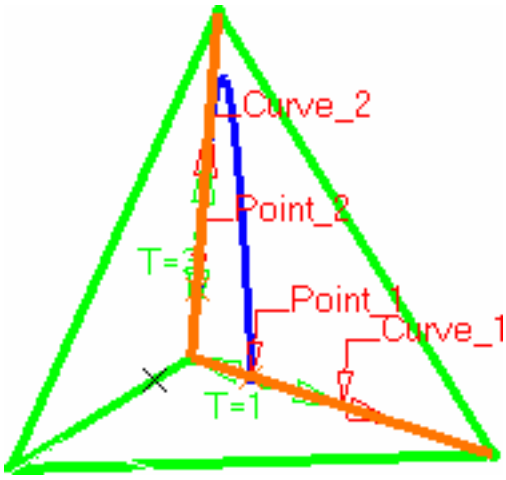
The connect curve is displayed between the two selected points according to the specified continuity and tension values.



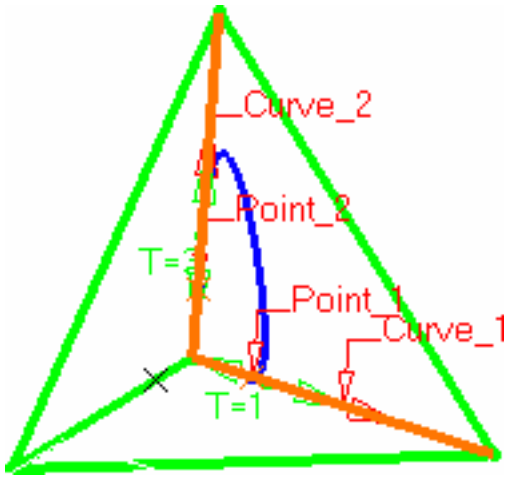
Connect curve with point continuity at both points



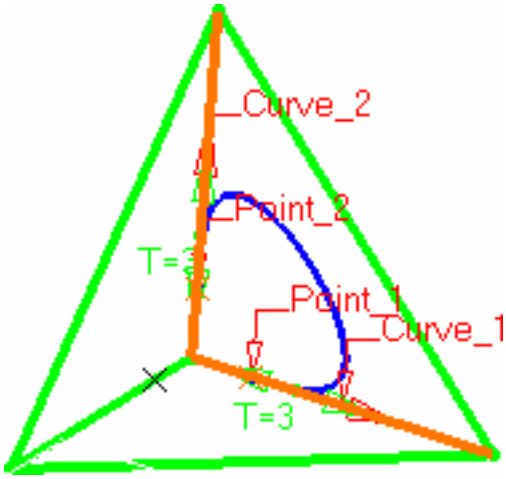
Connect curve with point continuity at one point and tangent continuity at the other



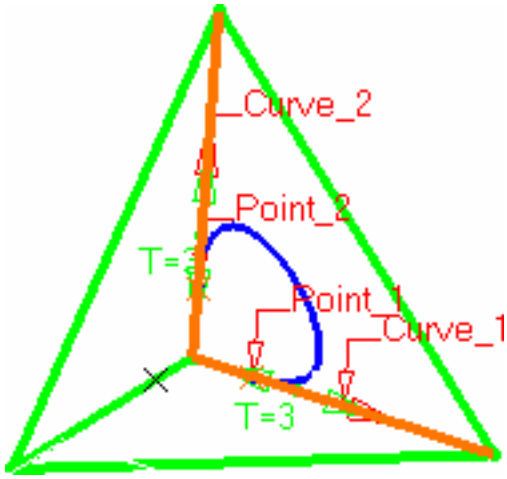
Connect curve with point continuity at one point and curvature continuity at the other



Connect curve with tangent continuity at one point and curvature continuity at the other



Connect curve with curvature continuity at both points

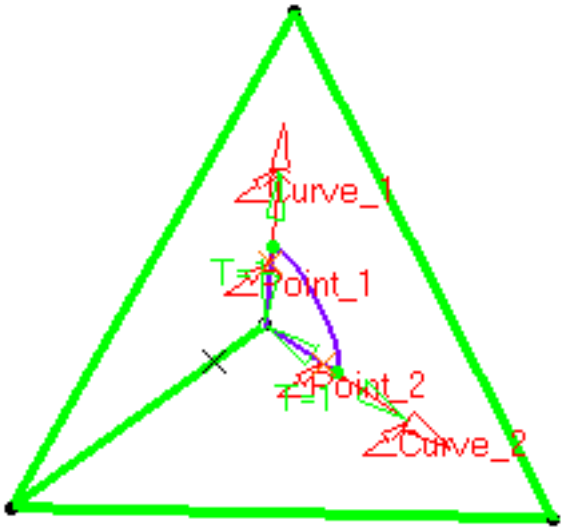


Connect curve with tangent continuity at both points

5. An arrow is displayed at each extremity of the curve. You can click the arrow to reverse the orientation of the curve at that extremity.

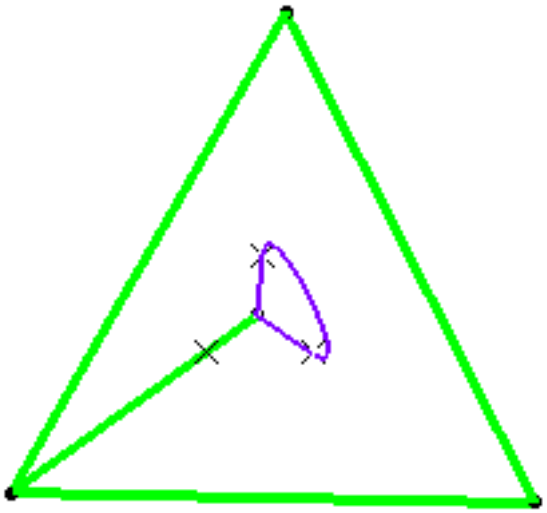
A graphic manipulator also allows you to modify the tension at the extremity of the connect curve, rather than in the dialog box.

6. You can select the **Trim elements** check box if you want to trim and assemble the two initial curves to the connect curve.



7. Click **OK** to create the connect curve.

   The curve (identified as Connect.xxx) is added to the specification tree.



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Creating Projections

This task shows you how to create geometry by projecting one or more elements onto a support. The projection may be normal or along a direction.
You can project:

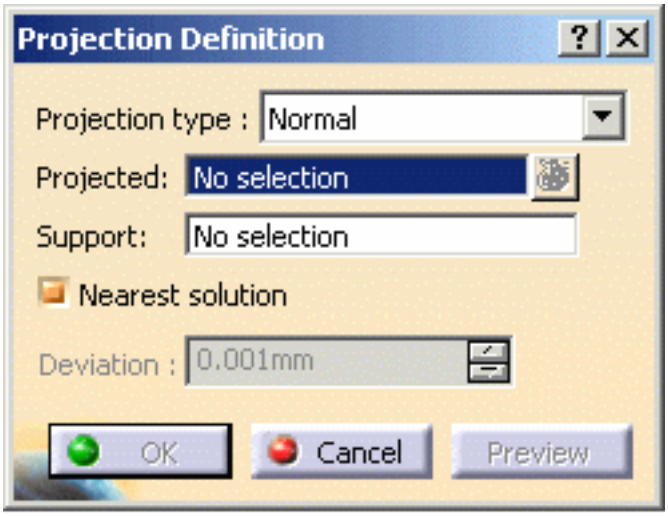
- a point onto a surface or wireframe support
- wireframe geometry onto a surface support
- any combination of points and wireframe onto a surface support.

Generally speaking, the projection operation has a derivative effect, meaning that there may be a continuity loss when projecting an element onto another. If the initial element presents a curvature continuity, the resulting projected element presents at least a tangency continuity. If the initial element presents a tangency continuity, the resulting projected element presents at least a point continuity.

Open the Projection1.CATPart document.

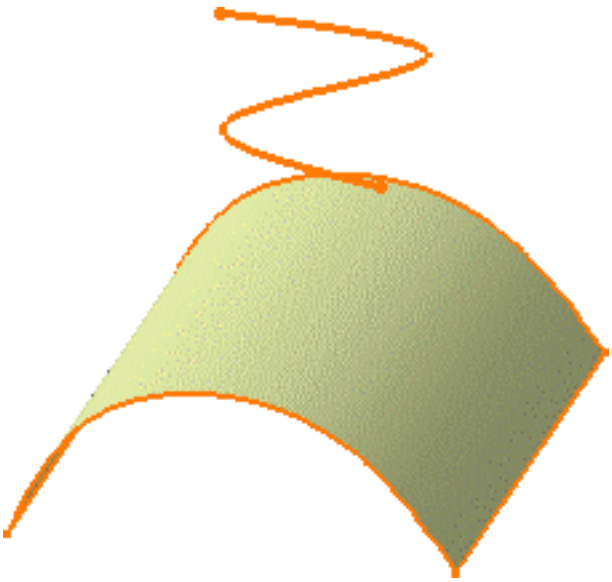
1. Click the **Projection** icon .

   The Projection Definition dialog box appears.

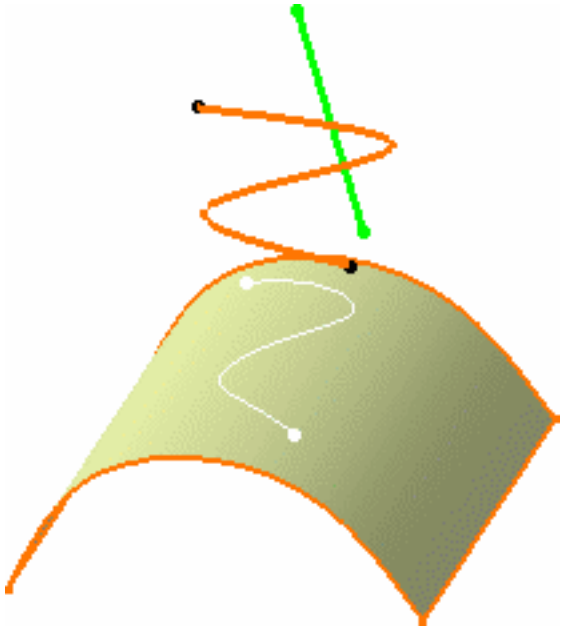


2. Select the element to be **Projected**.

   You can select several elements to be projected. In this case, the **Projected** field indicates: **x elements**



3. Select the **Support** element.

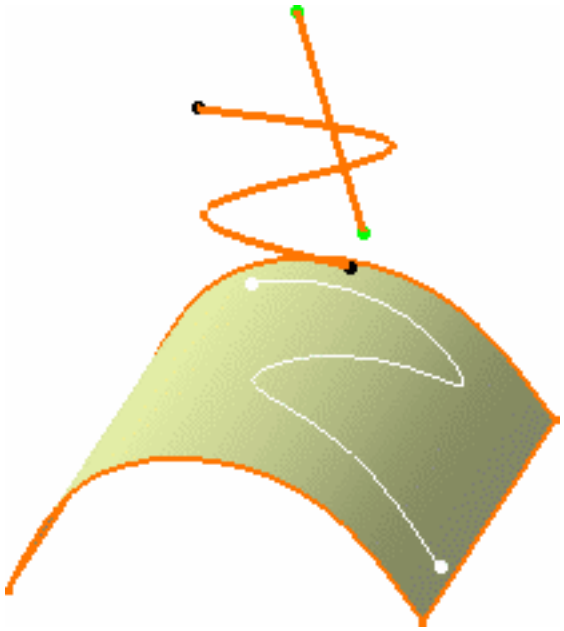


4. Use the combo to specify the direction type for the projection:

- **Normal**: the projection is done normal to the support element.

- **Along a direction**: you need to select a line to take its orientation as the translation direction or a plane to take its normal as the translation direction.

  You can also specify the direction by means of X, Y, Z vector components by using the contextual menu on the **Direction** field.



7. Click OK to create the projection element.

The projection (identified as Project.xxx) is added to the specification tree.

- Whenever several projections are possible, you can select the **Nearest Solution** check box to keep the nearest projection.

The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.

Multi-selection is available. Refer to Editing a List of Elements to find out how to display and manage the list of selected elements.



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Creating Intersections

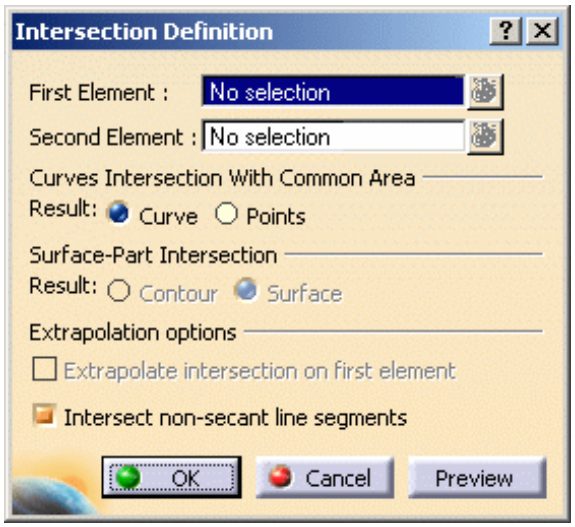
This task shows you how to create wireframe geometry by intersecting elements.

You can intersect:

- wireframe elements
- surfaces
- wireframe elements and a surface.

Open the Intersection1.CATPart document.

1. Click the **Intersection** icon .



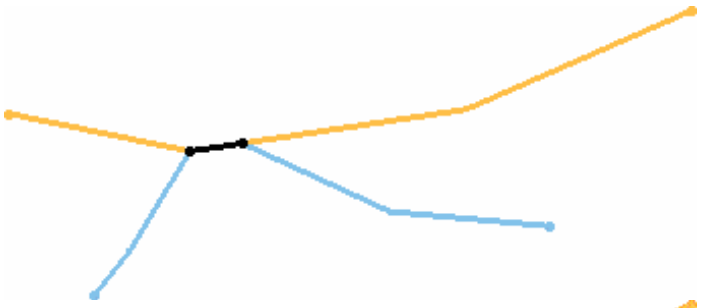
The Intersection Definition dialog box appears as well as the Multi-Selection dialog box allowing to perform multi-selection.

2. Select the two elements to be intersected.
The intersection is displayed.

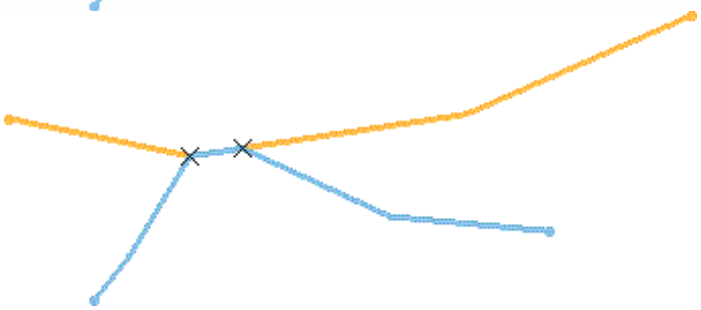
Multi-selection is available on the first selection, meaning you can select several elements to be intersected, but only one intersecting element.

3. Choose the type of intersection to be displayed:

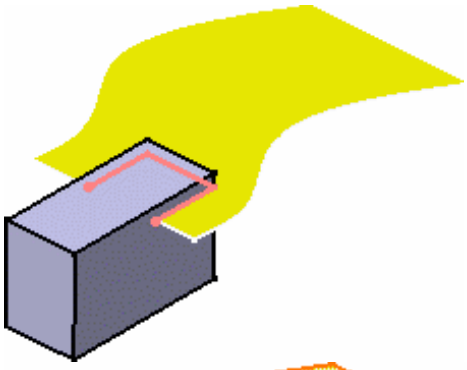
- A **Curve**: when intersecting a curve with another one



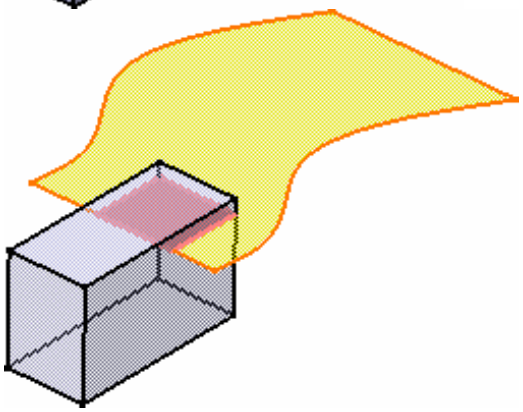
- **Points**: when intersecting a curve with another one



- A **Contour**: when intersecting a solid element with a surface



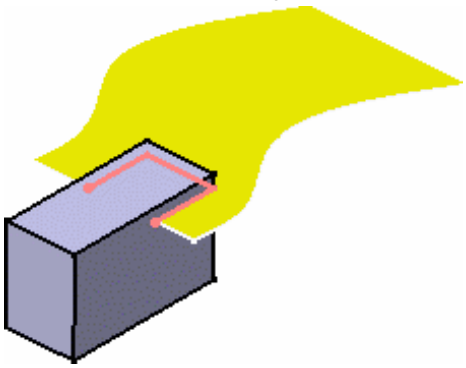
- A **Face**: when intersecting a solid element with a surface (we increased the transparency degree on the pad and surface)



4. Click OK to create the intersection element.

   This element (identified as Intersect.xxx) is added to the specification tree.

This example shows the line resulting from the intersection of a plane and a surface



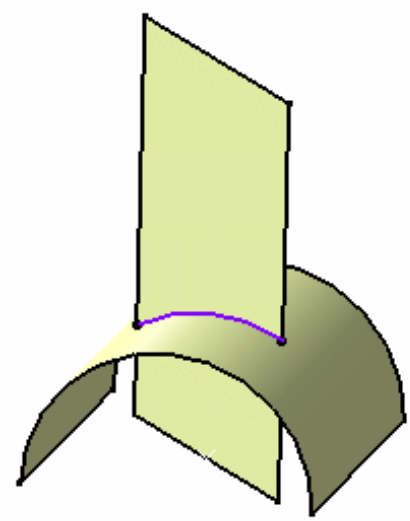
This example shows the curve resulting from the intersection of two surfaces



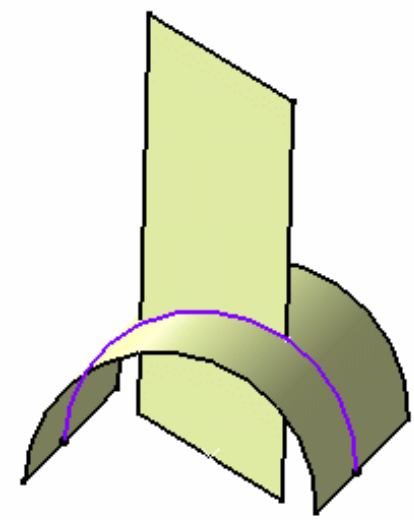
Two options can be defined to improve the preciseness of the intersection.

Open the Intersection2.CATPart document.

- The **Extrapolate intersection on first element** check box enables you to perform an extrapolation on the first selected element, in the case of a surface-surface intersection. In all the other cases, the option will be grayed.

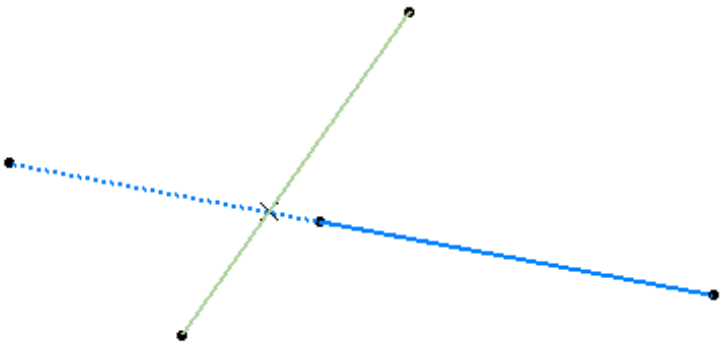


Intersection without the Extrapolation option checked

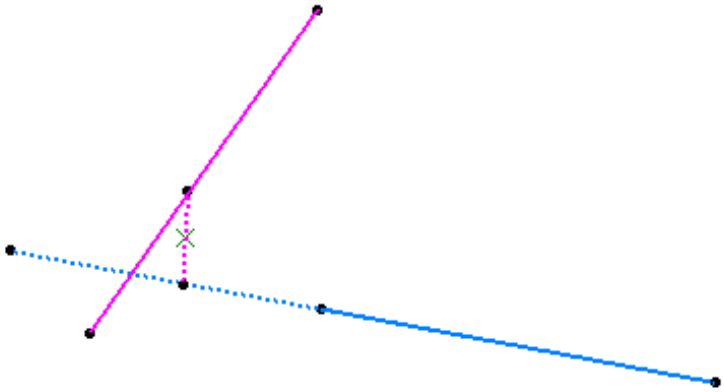


Intersection with the Extrapolation option checked

- The **Intersect non-secant line segments** check box enables you to perform an intersection on two non-cutting lines. In all the other cases, the option will be grayed.



Intersection between the light green line and the blue line: the intersection point is calculated after the blue line is extrapolated



Intersection between the pink line and the blue line: the intersection is calculated as the mid-point of minimum distance between the two lines

- Avoid using input elements which are tangent to each other since this may result in geometric instabilities in the tangency zone.
- If you intersect closed surfaces, they need to be created in two different open bodies.

The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.

Creating Surfaces

Wireframe and Surface allows you to model both simple and complex surfaces using techniques such as extruding, lofting and sweeping.

Two creation modes are available: either you create geometry with its history or not. Geometry with no history is called a datum. Please refer to Creating Datums for more information.


Create extruded surfaces: select a profile, specify the extrusion direction, the start and end limits.


Create surfaces of revolution: select a profile and a rotation axis, and key in an angle


Create spherical surfaces: select the center point of the sphere, the axis-system defining the meridian and parallel curves, and define the angular limits of the spherical surface


Create cylindrical surfaces: select the center point of the circle and specify the extrusion direction.


Offset surfaces: select a surface, specify the offset value and choose the offset direction


Create swept surfaces: select a guide curve, a planar profile, optionally a spine and second guide curve, and position the profile


Create filling surfaces: select curves/surface edges to form a closed boundary and specify the continuity type


Creating lofted surfaces: select one or two planar section curves, optionally guide curves and a spine


Creating blended surfaces: select two curves, and possibly their support, specify the tension, continuity, closing point and coupling ratio, if needed.

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Creating Extruded Surfaces

This task shows how to create a surface by extruding a profile along a given direction.

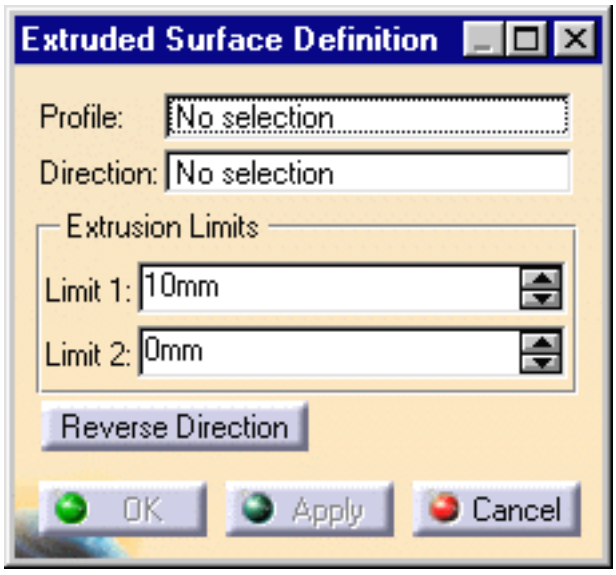
Open the Extrude1.CATPart document.

1. Click the **Extrude** icon .

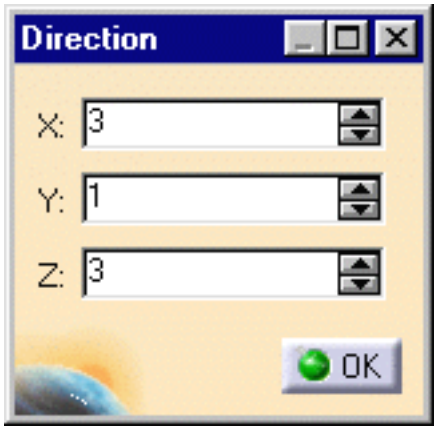
   The Extruded Surface Definition dialog box appears.

2. Select the profile to be extruded and specify the desired extrusion direction.

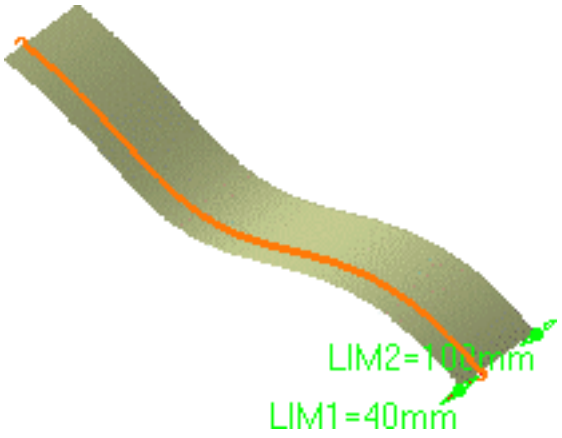
   You can select a line to take its orientation as the extrusion direction or a plane to take its normal as extrusion direction.



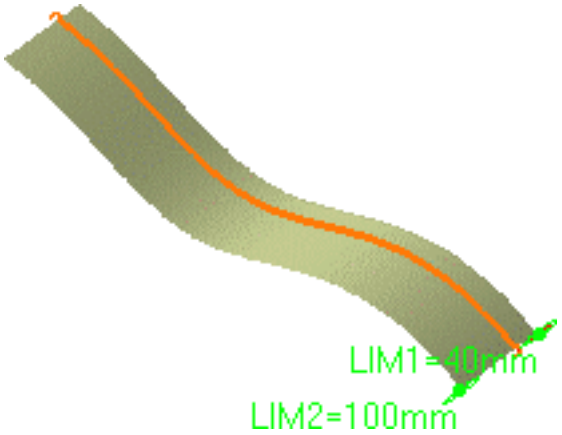
You can also specify the direction by means of X, Y, Z vector components by using the contextual menu on the Direction area.



3. Enter numerical values or use the graphic manipulators to define the start and end limits of the extrusion.

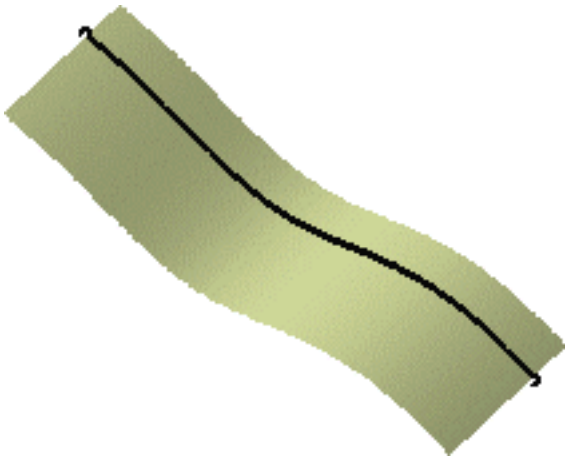


4. You can click the **Reverse Direction** button to display the extrusion on the other side of the selected profile.



5. Click OK to create the surface.

   The surface (identified as Extrude.xxx) is added to the specification tree.



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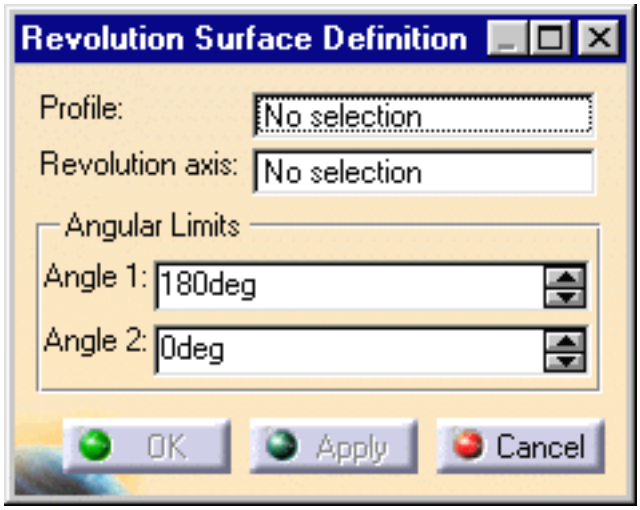
Creating Revolution Surfaces

This task shows how to create a surface by revolving a planar profile about an axis.

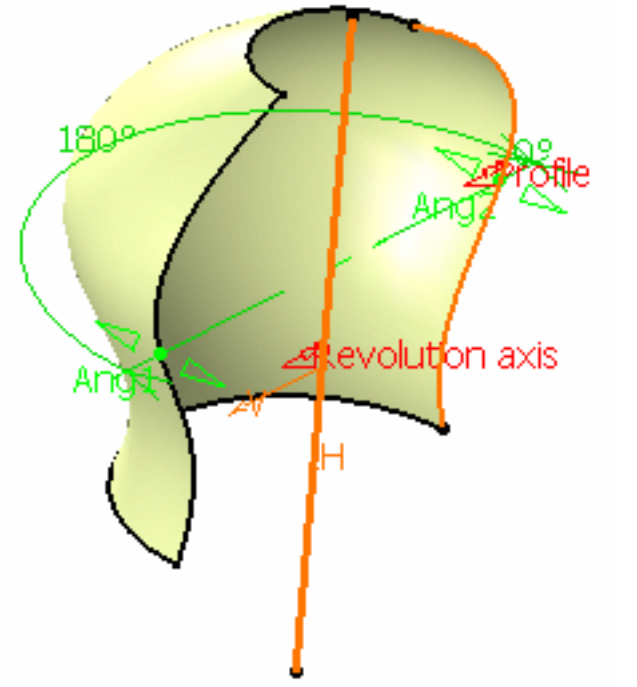
Open the Revolution1.CATPart document.

1. Click the **Revolve** icon .

   The Revolution Surface Definition dialog box appears.

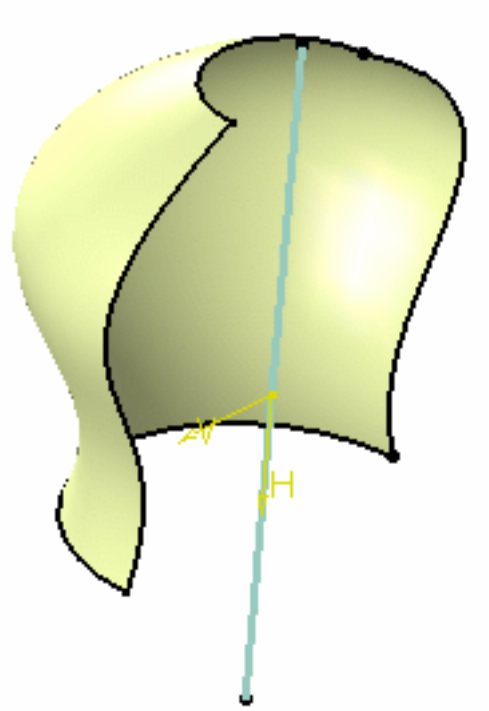


2. Select the **Profile** and a line indicating the desired **Revolution axis**.
3. Enter angle values or use the graphic manipulators to define the angular limits of the revolution surface.



4. Click **OK** to create the surface.

The surface (identified as Revolute.xxx) is added to the specification tree.



- There must be no intersection between the axis and the profile. However, if the result is topologically consistent, the surface will still be created.
- If the profile is a sketch containing an axis, the latter is selected by default as the revolution axis. You can select another revolution axis simply by selecting a new line.

Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.

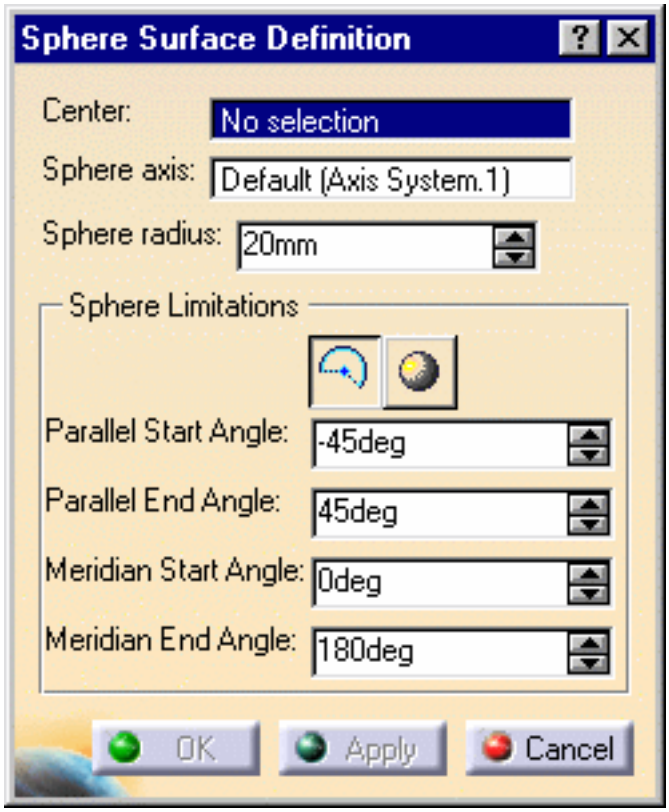
Creating Spherical Surfaces

This task shows how to create surfaces in the shape of a sphere.
The spherical surface is based on a center point, an axis-system defining the meridian & parallel curves orientation, and angular limits.

Open the Sphere1.CATPart document.

1. Click the **Sphere** icon from the Extrude-Revolution toolbar.

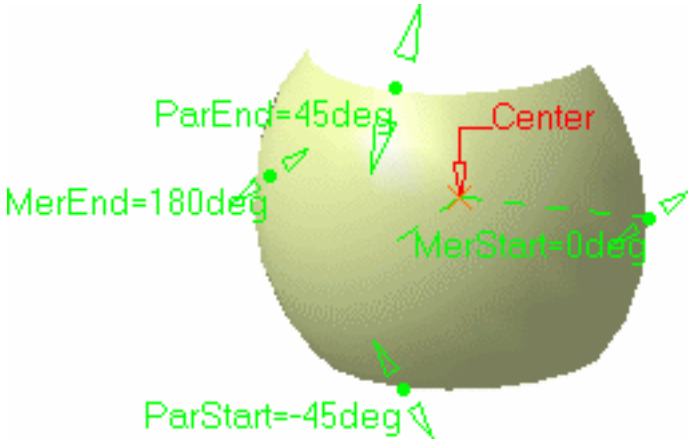
   The Sphere Surface Definition dialog box is displayed.



2. Select the center point of the sphere.
3. Select an axis-system.

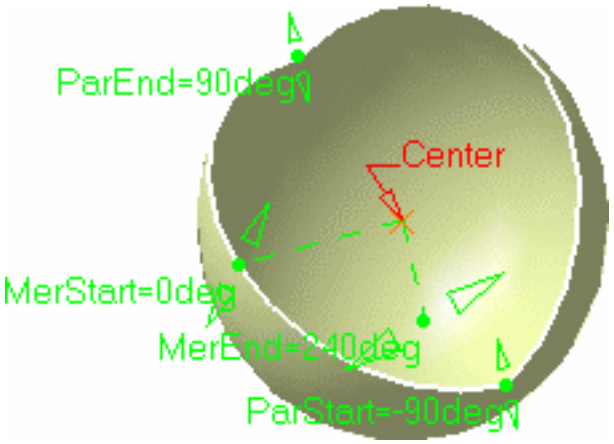
   This axis-system determines the orientation of the meridian and parallel curves, and therefore of the sphere.

   By default, if no axis-system has been previously created in the document, the axis-system is the document xyz axis-system. Otherwise the default axis-system is the current one.



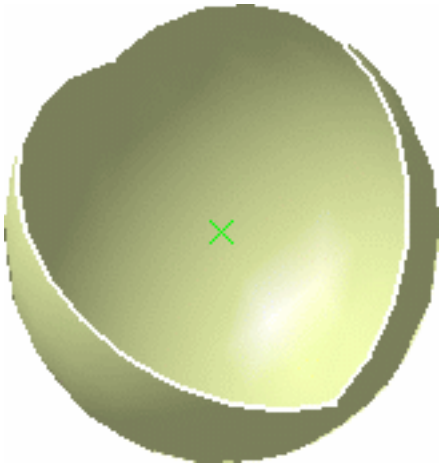
4. Click Apply to preview the surface.

5. Modify the **Sphere radius** and the **Angular Limits** as required.
Here we choose -90° and 90° for the parallel curves, and 240° and 0° for the meridian curves, and left the radius at 20 mm.



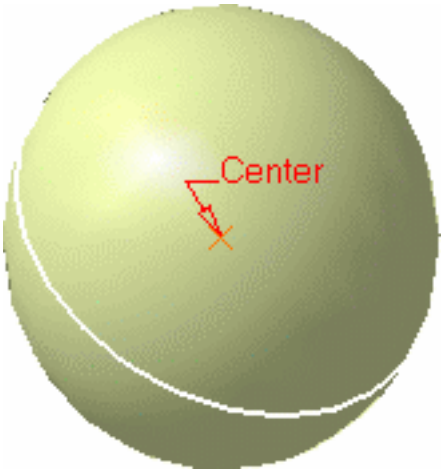
Parallel angular limits are comprised within the -90° and 90° range.
Meridian angular limits are comprised within the -360° and 360° range.

6. Click OK to create the surface.



The spherical surface (identified as Sphere.xxx) is added to the specification tree.

You can also choose to create a whole sphere. In this case, simply click the  icon from the dialog box to generate a complete sphere, based on the center point and the radius. The parallel and meridian angular values are then grayed.



Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.



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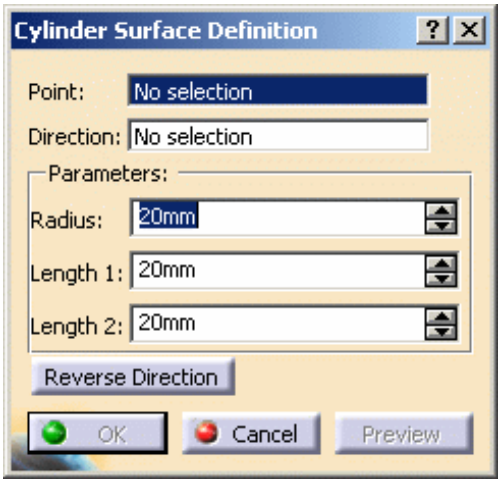
Creating Cylindrical Surfaces

This task shows how to create a cylinder by extruding a circle along a given direction.

Open the Cylinder1.CATPart document.

1. Click the **Cylinder** icon .

   The Cylinder Surface Definition dialog box appears.

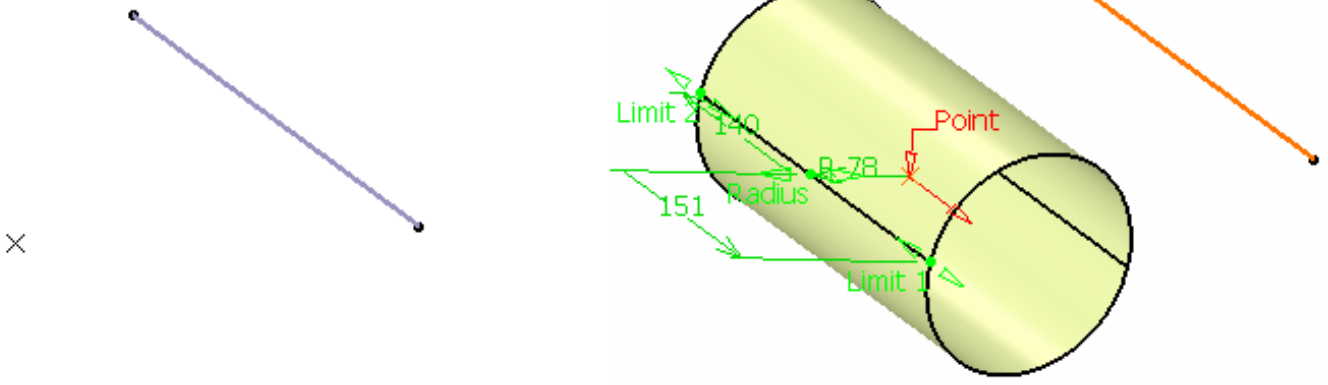


2. Select the **Point** that gives the center of the circle to be extruded and specify the desired **Direction** of the cylinder axis.

   You can select a line to take its orientation as the direction or a plane to take its normal as direction.

   You can also specify the direction by means of X, Y, Z vector components by using the contextual menu on the **Direction** area.

3. Select the **Radius** of the cylinder.
4. Enter values or use the graphic manipulators to define the start and end limits of the extrusion.



5. You can click the **Reverse Direction** button to display the direction of the cylinder on the other side of the selected point or click the arrow in the 3D geometry.
6. Click **OK** to create the surface.

The surface (identified as Cylinder.xxx) is added to the specification tree.

Creating Offset Surfaces

This task shows how to create a surface by offsetting an existing surface.

Open the Offset1.CATPart document.

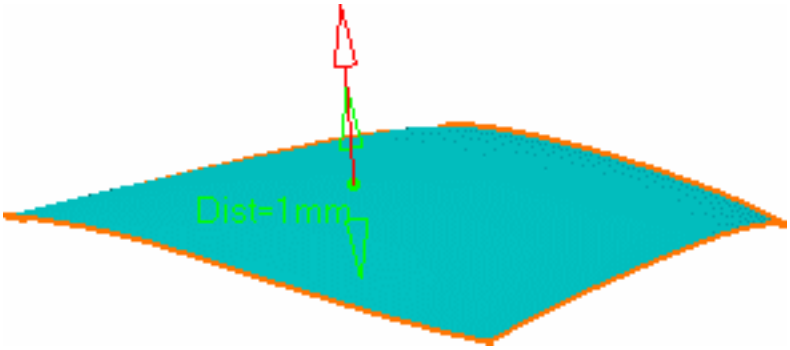
1. Click the **Offset** icon .

   The Offset Surface Definition dialog box appears.



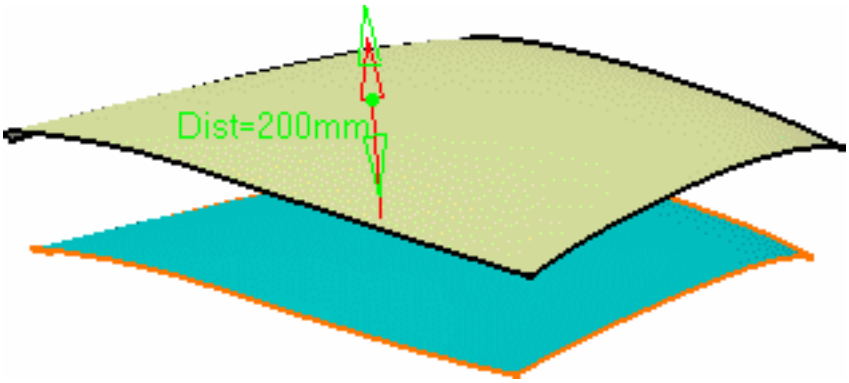
2. Select the surface to be offset.
3. Specify the offset by entering a value or using the graphic manipulator.
4. An arrow indicates the proposed direction for the offset.

   The offset surface is displayed normal to the reference surface.



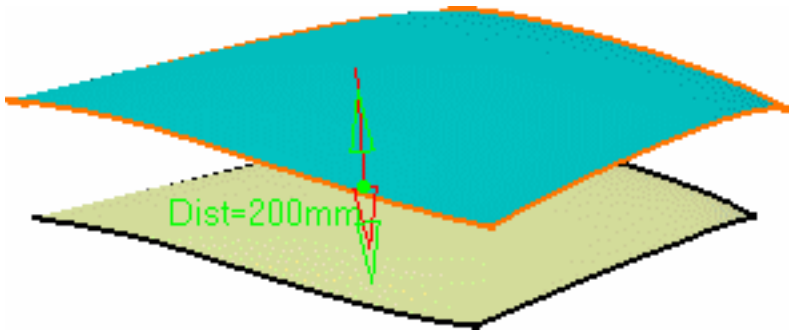
5. Click Apply to previews the offset surface.

   The offset surface is displayed normal to the reference surface.

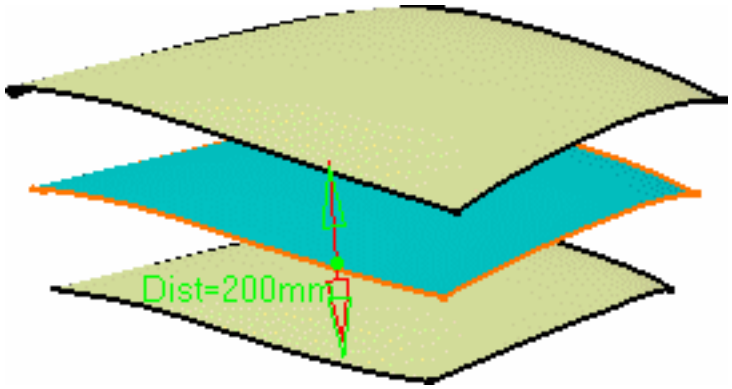


Depending on the geometry configuration and the offset value, an offset may not be allowed as it would result in a debased geometry. In this case, you need to decrease the offset value or modify the initial geometry.

You can display the offset surface on the other side of the reference surface by clicking either the arrow or the **Reverse Direction** button.



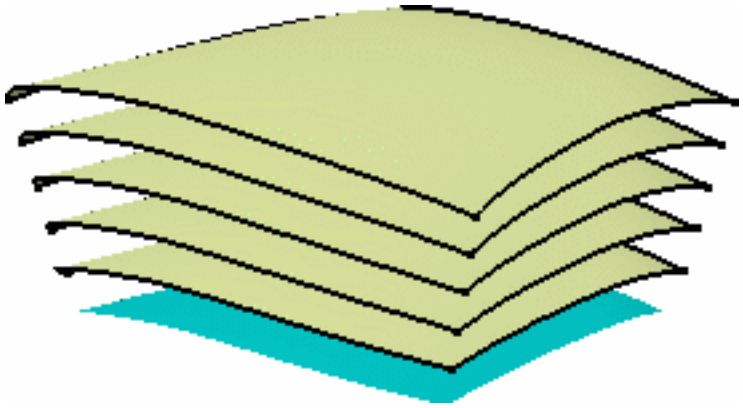
6. Check the **Both sides** button to generate two offset surfaces, one on each side of the reference surface.



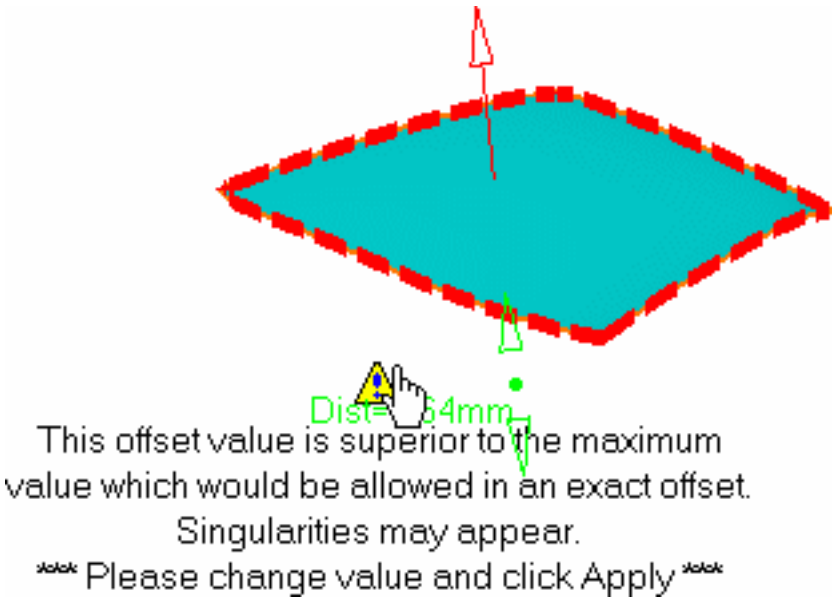
7. Click OK to create the surfaces.

   The surfaces (identified as Offset.xxx) are added to the specification tree.

- Use the **Repeat object after OK** checkbox to create several offset surfaces, each separated from the initial surface by a multiple of the **offset** value.
  Simply indicate in the Object Repetition dialog box the number of instances that should be created and click OK.
  Remember however, that when repeating the offset it may not be allowed to create all the offset surfaces, if it leads to debased geometry.



- Would the value be inconsistent with the selected geometry, a warning message is displayed, along with a warning sign onto the geometry. If you move the pointer over this sign, a longer message is displayed to help you continue with the operation.



Furthermore, the manipulator is locked, and you need to modify the value within the dialog box and click Apply.

The options set in the dialog box are retained when exiting then returning to the Offset function.

Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.

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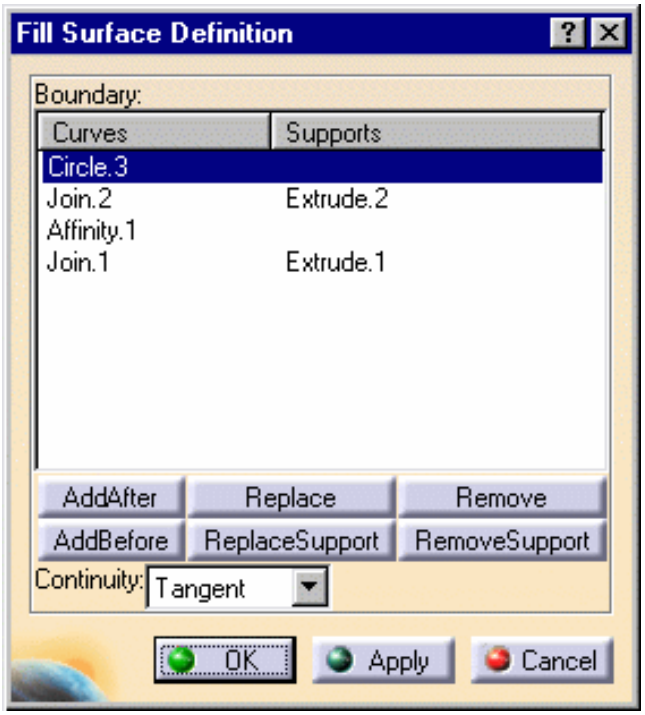
Creating Filling Surfaces

This task shows how to create fill surfaces between a number of boundary segments.

Open the Fill1.CATPart document.

1. Click the **Fill** icon .

   The Fill Surface Definition dialog box appears.

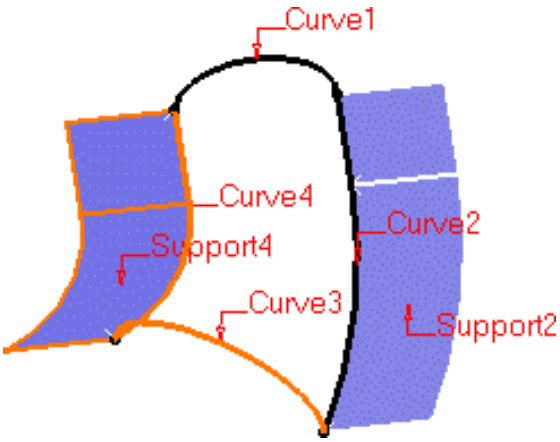


2. Select curves or surface edges to form a closed boundary.

   You can select a support surface for each curve or edge. In this case continuity will be assured between the fill surface and selected support surfaces.

3. Use the combo to specify the desired continuity type between any selected support surfaces and the fill surface: **Point** or **Tangent**.

   The fill surface is displayed within the boundary.



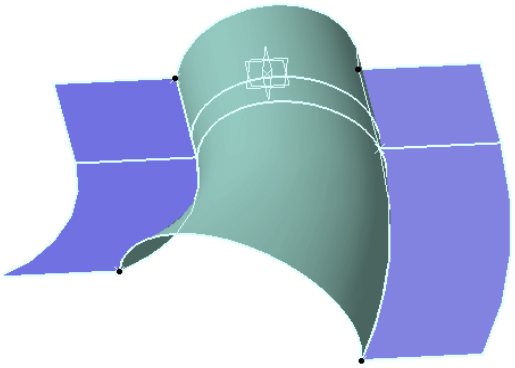
4. You can edit the boundary by first selecting an element in the dialog box list then choosing a button to either:

   - Add a new element after or before the selected one
   - Remove the selected element
   - Replace the selected element by another curve
   - Replace the selected support element by another support surface
   - Remove the selected support element.

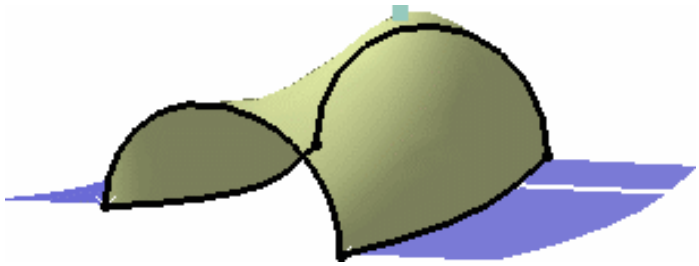
This point should lie within the area delimited by the selected curves. If not, the results may be inconsistent.

6. Click **OK** to create the fill surface.

The surface (identified as Fill.xxx) is added to the specification tree.

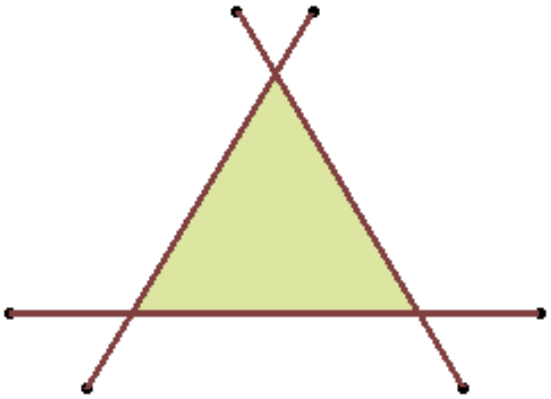


Filling surface with specified supports



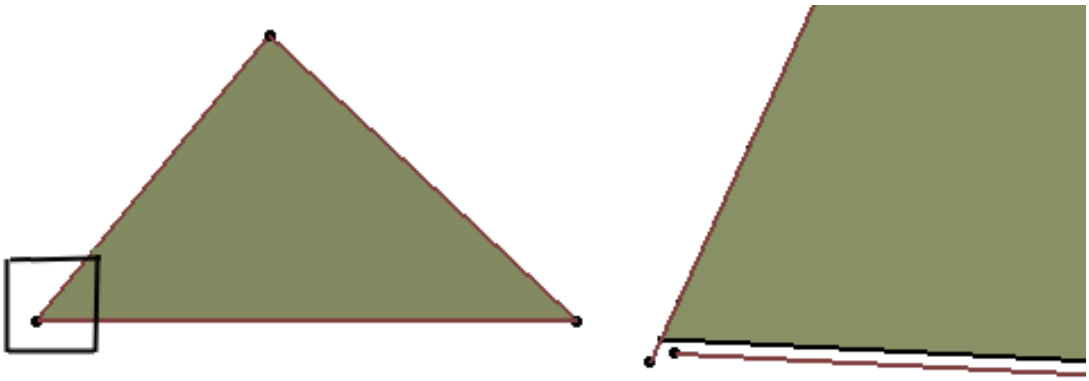
Filling surface with passing point (P2 only)

- The selected curves or surfaces edges can now intersect. Therefore a relimitation of the intersecting boundaries is performed to allow the creation of the fill surface.



Two consecutive boundaries must have only one intersection.

- The selected curves or surfaces edges can have non-coincident boundaries. Therefore, an extrapolation is performed to allow the creation of the fill surface.



The distance between non-coincident boundaries must be smaller than 0.1mm.

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Creating Swept Surfaces

This task shows how to create a swept surface that uses an explicit profile.

You can create a swept surface by sweeping out a profile in planes normal to a spine curve while taking other user-defined parameters (such as guide curves and reference elements) into account.

You can sweep an explicit profile:

- along one or two guide curves (in this case the first guide curve is used as the spine)
- along one or two guide curves while respecting a spine.

The profile is swept out in planes normal to the spine.

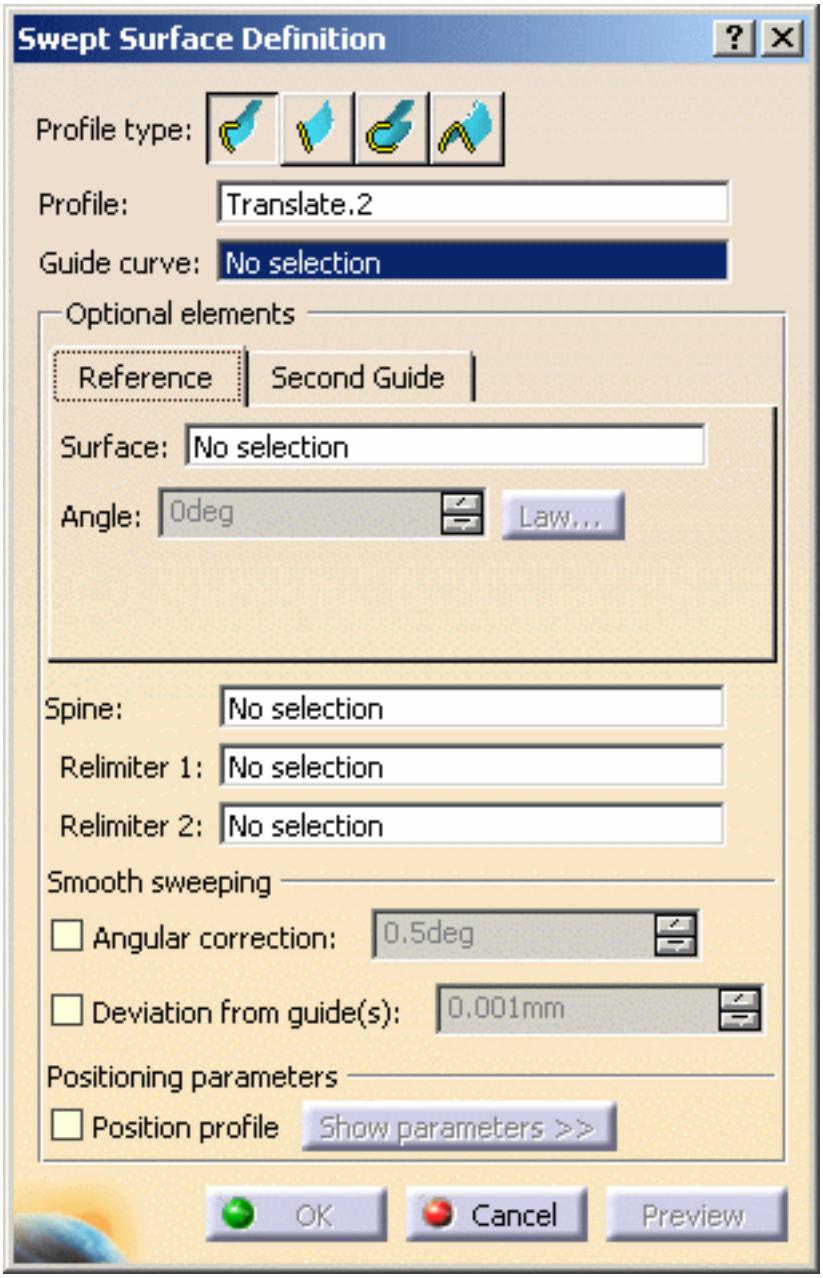
In addition, you can control the positioning of the profile while it is being swept by means of a reference surface.

The profile position may be fixed with respect to the guide curve (positioned profile) or user-defined in the first sweep plane (CATIA P2 only).

Open the Sweep1.CATPart document.

1. Click the **Sweep** icon  .

   The Swept Surface Definition dialog box appears.

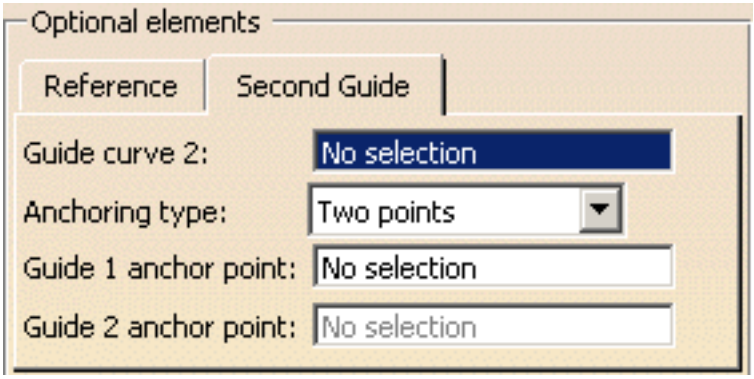


2. Select the planar profile to be swept out, that is the circle.
3. Select a guide curve.



4. If needed, select a spine.
   If no spine is selected, the guide curve is implicitly used as the spine.
5. If needed, select a second guide curve.

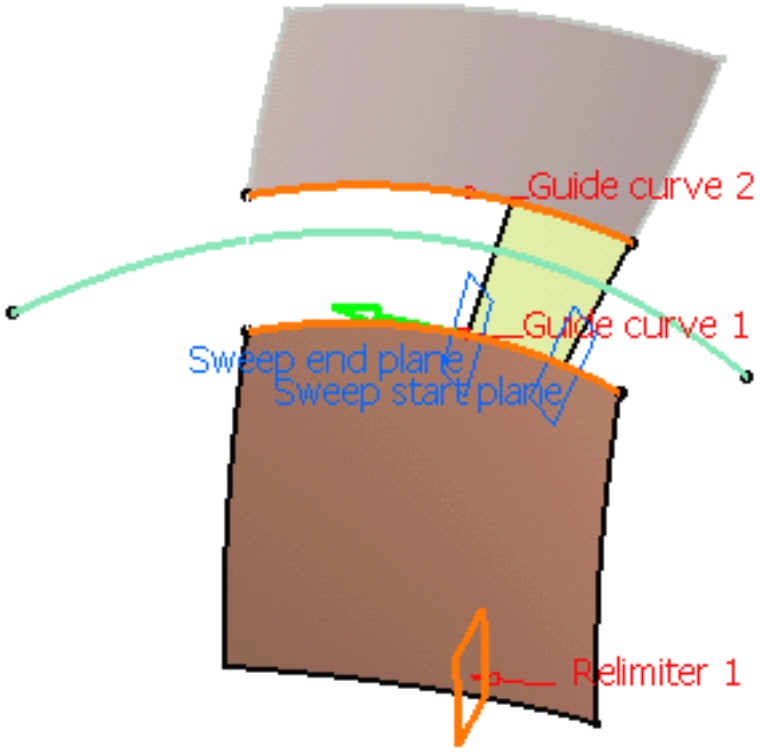
Within the Second guide tab, you can also specify anchor points for each guide. These anchor points are intersection points between the guides and the profile's plane or the profile itself, through which the guiding curves will pass.



There are two anchoring types:

- Two points: select an anchor point on each guide curve
  If the profile is open, these points are optional and the extremities of the profile are used.
- Point and direction: select an anchor point on Guide Curve 1 and an anchor direction.
  In each sweeping plane, the default x-axis is aligned with the two guide curves.

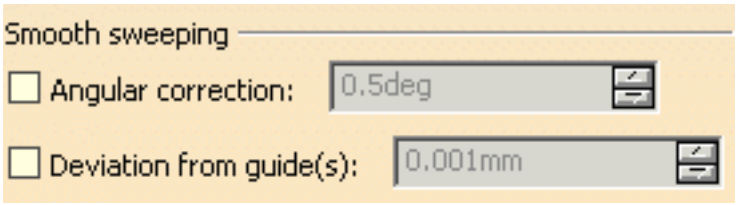
You can define relimiters (points or planes) in order to longitudinally reduce the domain of the sweep, if the swept surface is longer than necessary for example.



Besides is an example with a plane as Relimiter 1.
When there is only one relimiter, you are able to choose the direction of the sweep by clicking the green arrow.

- Relimiters can be selected on a closed curve (curve, spine, or default spine). In that case, you are advised to define points as relimiters, as plane selection may lead to unexpected results due to multi-intersection.
- You can stack the creation of the elements by using the contextual menu available in either field.

- In the Smooth sweeping section, you can check:
  - the **Angular correction** option to smooth the sweeping motion along the reference surface. This may be necessary when small discontinuities are detected with regards to the spine tangency or the reference surface's normal. The smoothing is done for any discontinuity which angular deviation is smaller than 0.5 degree, and therefore helps generating better quality for the resulting swept surface.
  - the **Deviation from guide(s)** option to smooth the sweeping motion by deviating from the guide curve(s).



6. If you want to control the position of the profile during the sweep, you can select a reference surface. You can impose a reference angle on this surface.

7. Click OK to create the swept surface.

   The surface (identified as Sweep.xxx) is added to the specification tree.



Generally speaking, the sweep operation has a derivative effect, meaning that there may be a continuity loss when sweeping a profile along a spine. If the spine presents a curvature continuity, the surface presents at least a tangency continuity. If the spine presents a tangency continuity, the surface presents at least a point continuity.

Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.

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Creating Multi-section Surfaces

This task shows how to create a multi-section surface.

You can generate a multi-section surface by sweeping one or two planar section curves along a computed or user-defined spine. The surface can be made to respect one or more guide curves.

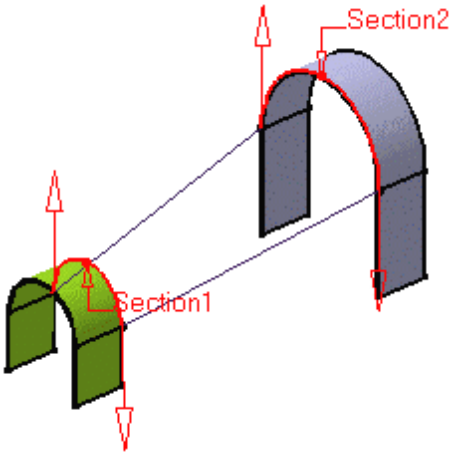
Open the Loft1.CATPart document.

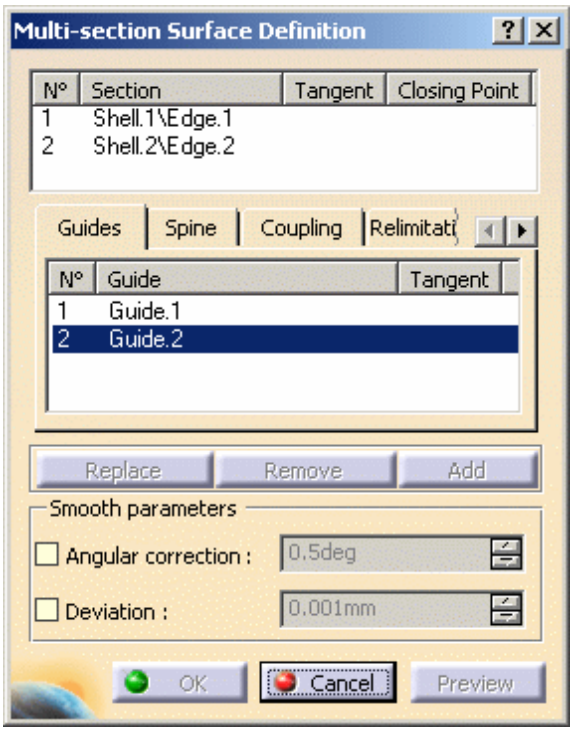
1. Click the **Multi-sections surface** icon .

   The Multi-section Surface Definition dialog box appears.

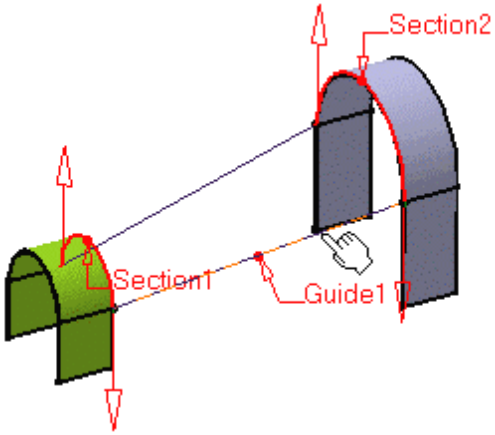
2. Select one or two planar section curves.

   These sections (two at maximum) may be tangent to support surfaces, provided they are not parallel. Closed section curves can have point continuity at each closing point.





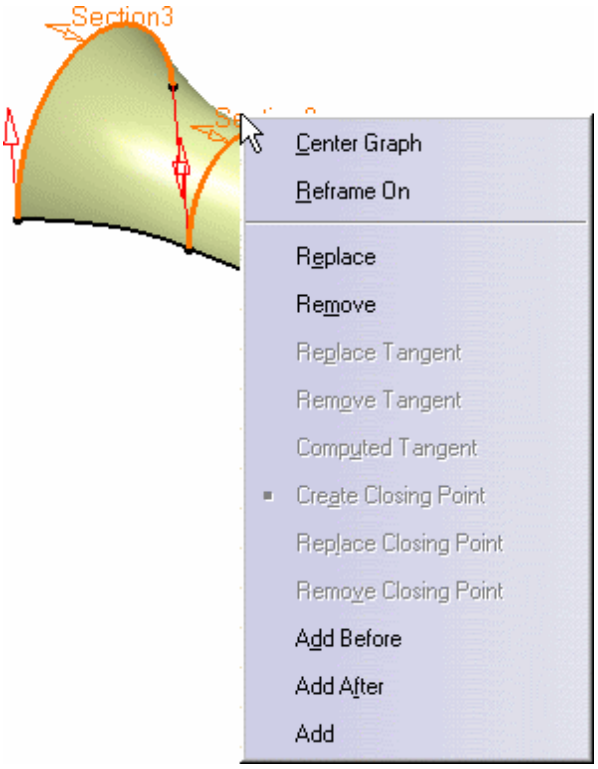
3. If needed, select one or more guide curves.



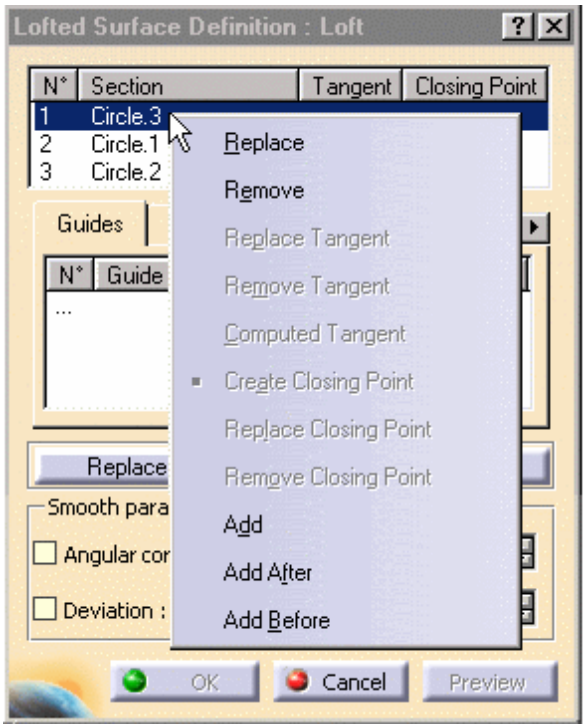
4. In the Spine tab page, select the **Spine** check box to use an automatically computed spine or select a curve to impose that curve as the spine.

5. It is possible to edit the multi-section surface reference elements by first selecting a curve in the dialog box list then choosing a button to either:

- remove the selected curve
- replace the selected curve by another curve
- add another curve.

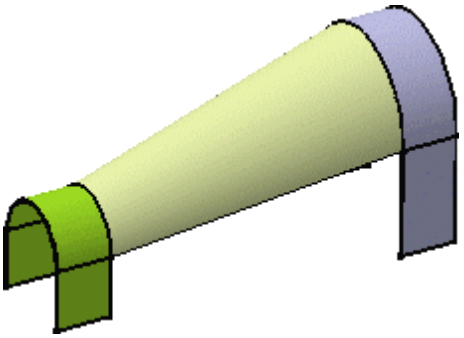


More possibilities are available with the contextual menu and by right-clicking on the red text or on the object. For example, it is possible to remove and replace tangent surfaces and closing points.



6. Click OK to create the multi-section surface.

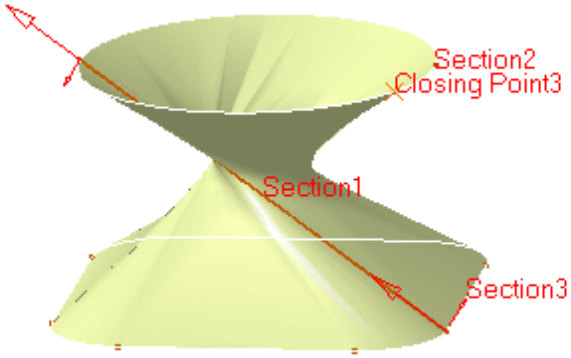
   The surface (identified as Multi-section surface.xxx) is added to the specification tree.



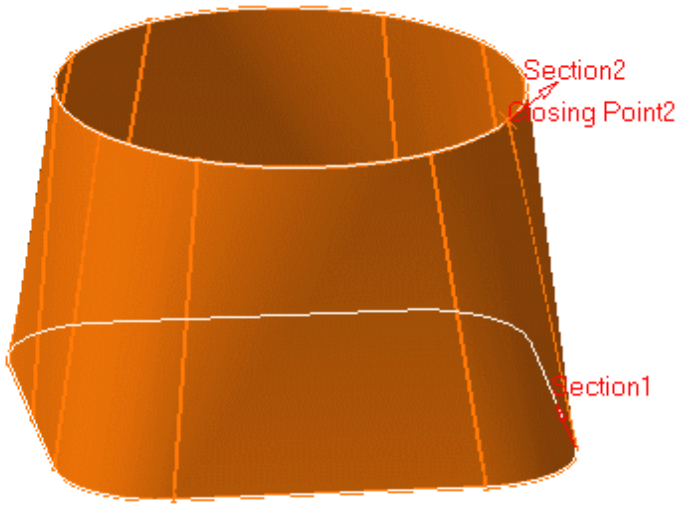
- You can impose tangency conditions onto sections and/or guides, by specifying a direction for the tangent vector (selecting a plane to take its normal, for example). This is useful for creating parts that are symmetrical with respect to a plane. Tangency conditions can be imposed on the two symmetrical halves.
  Similarly, you can impose a tangency onto each guide, by selection of a surface or a plane (the direction is tangent to the plane's normal). In this case, the sections must also be tangent to the surface.

- You can create multi-section surfaces between closed section curves. These curves have point continuity at their closing point.
  This closing point is either a vertex or an extremum point automatically detected and highlighted by the system.
  By default, the closing points of each section are linked to each other.

The red arrows in the figures below represent the closing points of the closed section curves. You can change the closing point by selecting any point on the curve.

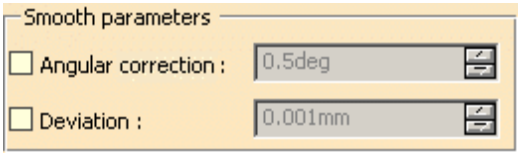


The surface is twisted

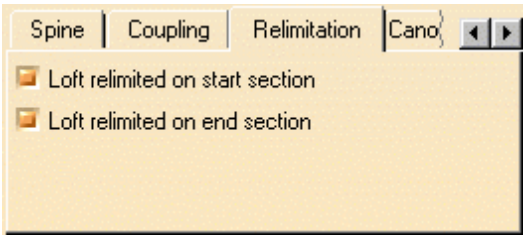


A new closing point has been imposed to get a non-twisted surface

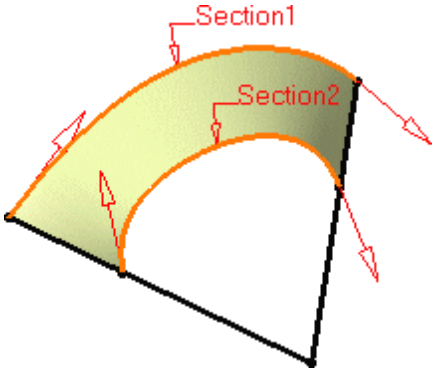
- In the Smooth parameters section, you can check:
  - the **Angular correction** option to smooth the lofting motion along the reference guide curves. This may be necessary when small discontinuities are detected with regards to the spine tangency or the reference guide curves' normal. The smoothing is done for any discontinuity which angular deviation is smaller than 0.5 degree, and therefore helps generating better quality for the resulting multi-section surface.
  - the **Deviation** option to smooth the lofting motion by deviating from the guide curve(s).



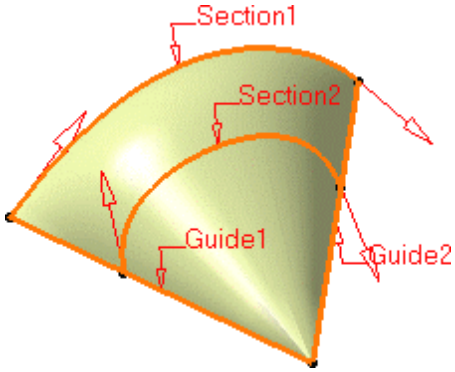
- The Relimitation tab lets you specify the multi-section surface relimitation type.
  You can choose to limit the multi-section surface only on the Start section, only on the End section, on both, or on none.



- when one or both are checked: the multi-section surface is limited to corresponding section
- when one or both are when unchecked: the multi-section surface is swept along the spine:
  - if the spine is a user spine, the multi-section surface is limited by the spine extremities
  - if the spine is an automatically computed spine, and no guide is selected:
    the multi-section surface is limited by the start and end sections
  - if the spine is an automatically computed spine, and guides are selected:
    the multi-section surface is limited by the guides extremities.



Multi-section surface relimitation option checked on both Start and End section



Multi-section surface relimitation option unchecked on End section only

- Use the **Planar surface detection** check button (Canonical Surfaces tab) to automatically convert planar surfaces into planes.

Coupling

This task presents the two kinds of coupling during the creation of the multi-section surface:

- coupling between two consecutive sections
- coupling between guides

Open the Loft2.CATPart document. To perform the following scenario you will need to get some geometry locate

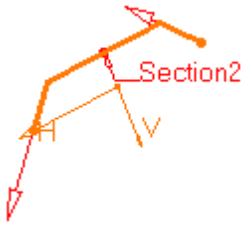
Coupling between two consecutive sections

This coupling is based on the curvilinear abscissa.

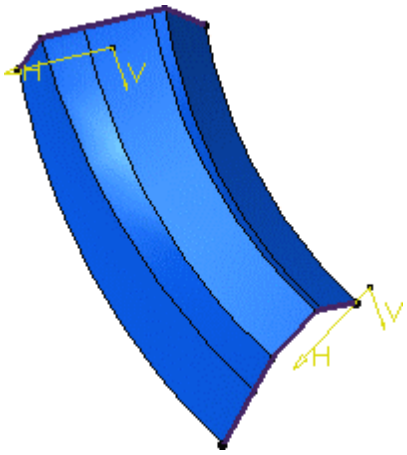
1. Click the **Multi-section surface** icon .

   The Multi-section Surface Definition dialog box appears.

2. Select the two consecutive sections.

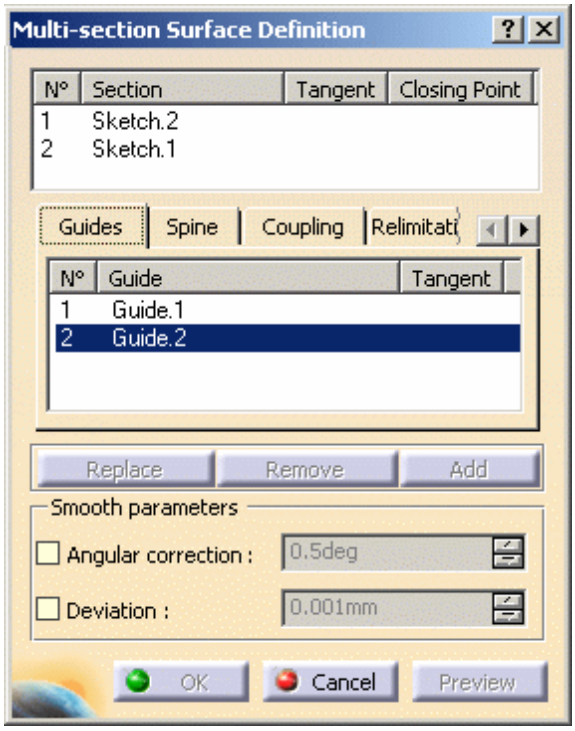




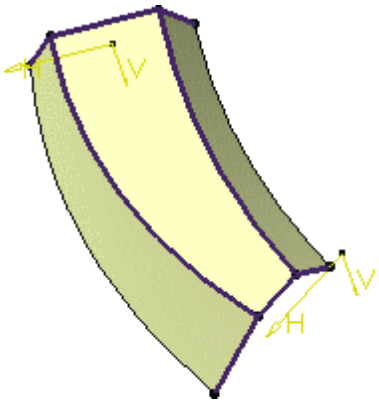


3. Click **OK** to create the multi-section surface.

To create a coupling between particular points, you can add guides or define the coupling type.

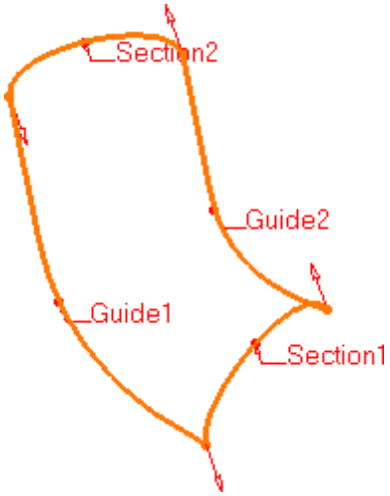


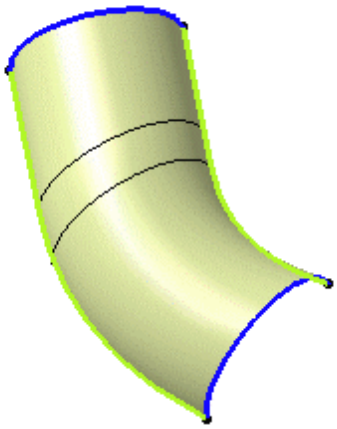




Coupling between guides

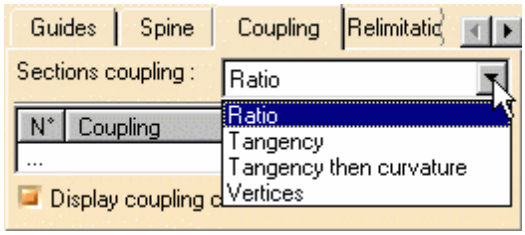
This coupling is performed by the spine.
If a guide is the concatenation of several curves, the resulting multi-section surface will contain as many surfaces as curves within the guide.



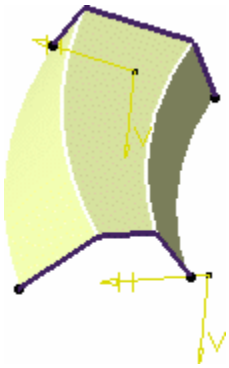


Several coupling types are available, depending on the section configuration:

- **Ratio**: the curves are coupled according to the curvilinear abscissa ratio.



- **Tangency**: the curves are coupled according to their tangency discontinuity points. If they do not have the same number of points, they cannot be coupled using this option.



- **Tangency then curvature**: the curves are coupled according to their tangency continuity first then curvature discontinuity points. If they do not have the same number of points, they cannot be coupled using this option.
- **Vertices**: the curves are coupled according to their vertices. If they do not have the same number of vertices, they cannot be coupled using this option.

Back Up Next

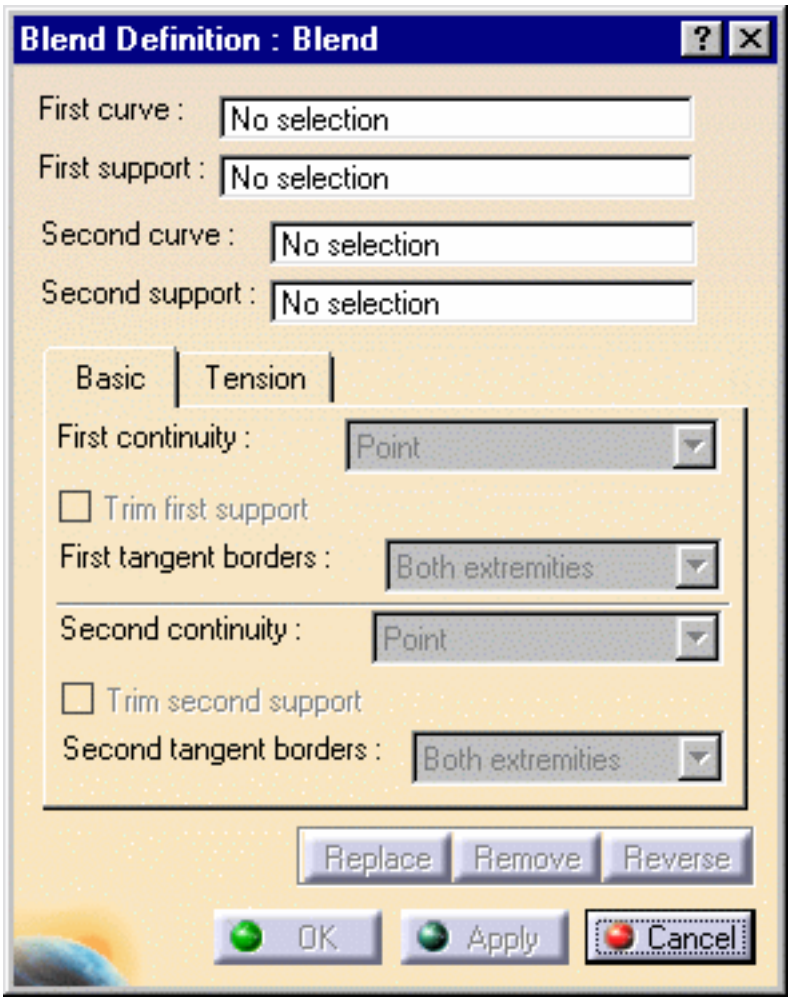
Creating Blended Surfaces

This task shows how to create a blended surface, that is a surface between two wireframe elements, taking a number of constraints into account, such as tension, continuity, and so forth.
Note that curves with one edge only can be used to create blend surfaces.

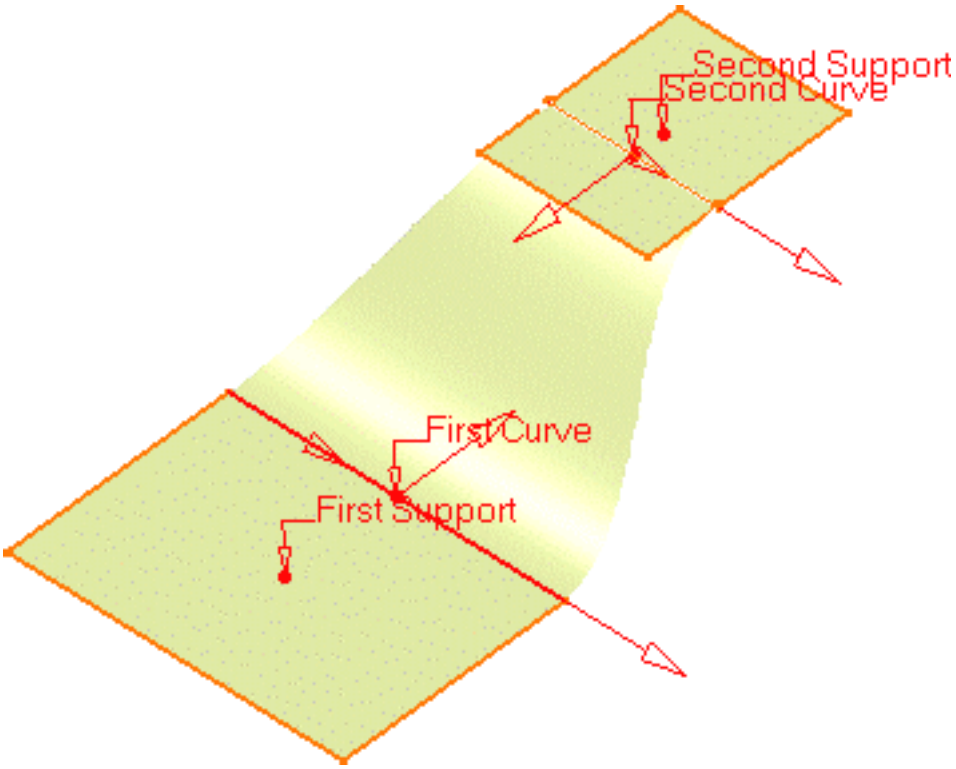
Open the Blend1.CATPart document.

1. Click the **Blend** icon .

   The Blend Definition : Blend dialog box appears.

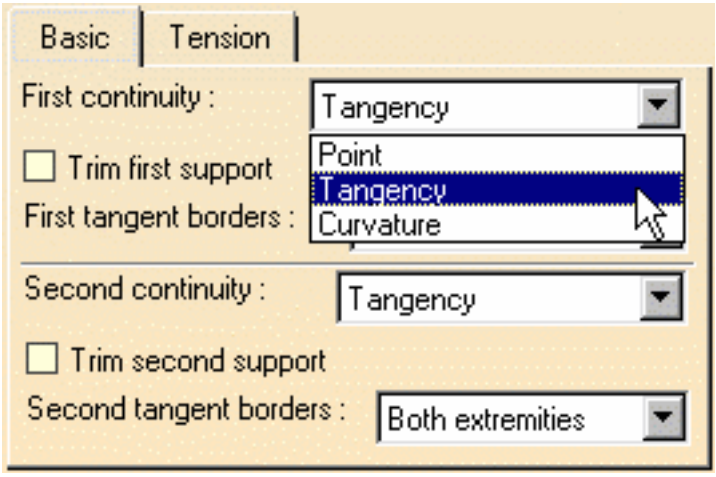


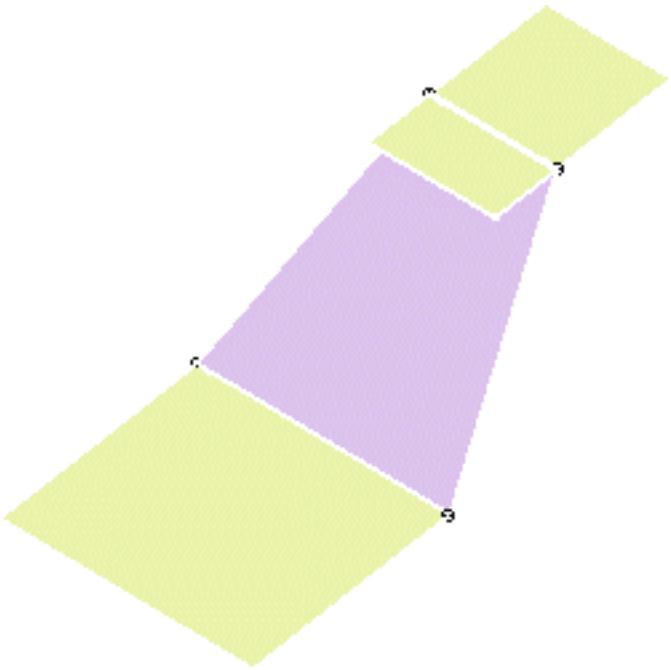
2. Successively select the first curve and its support, then the second curve and its support.
   These can be surface edges, or any curve.



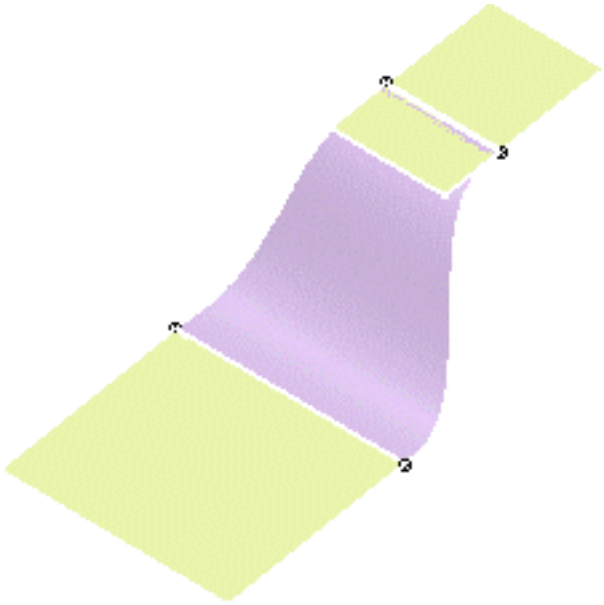
3. Set the continuity type using the **Basic** tab.
   It defines the continuity connection between the newly created surface and the curves on which it lies.

   The illustration above, shows the **Tangency** continuity, and the following illustrations show the **Point** and **Curvature** continuity types:





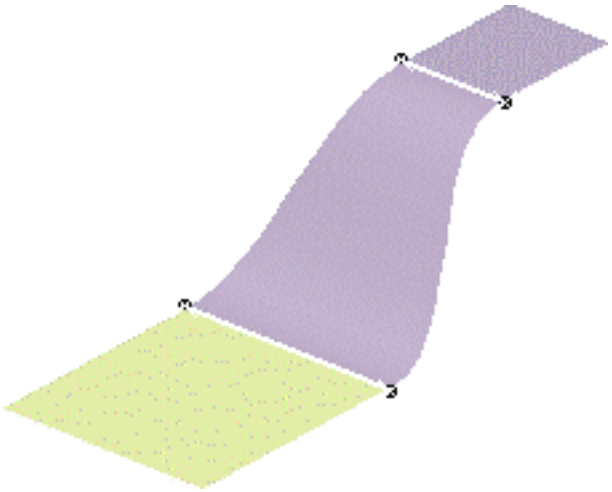
Point continuity on both limits



Curvature

4. Activate the **Trim first/second support** option, on one or both support surfaces to trim them by the curve and assemble them to the blend surface:

   By default the blend surface borders are tangent to the support surface borders.



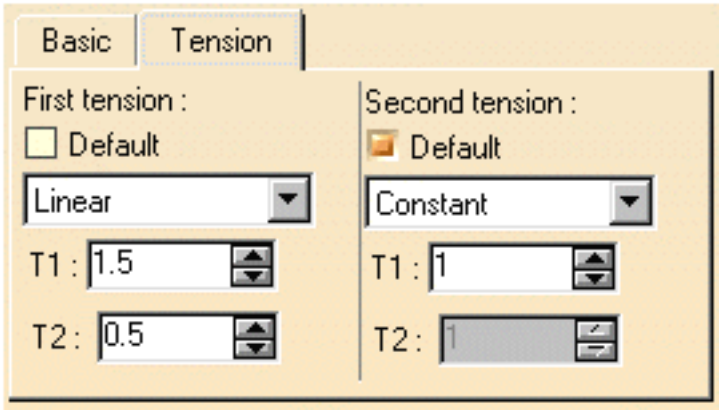
You can also specify whether and where the blend boundaries must be tangent to the supports boundaries:

- **Both extremities**: the tangency constraint applies at both ends of the curve
- **None**: the tangency constraint is disregarded
- **Start extremity**: the tangency constraint applies at the start endpoint of the curve only
- **End extremity**: the tangency constraint applies at the end endpoint of the curve only

The Start and End extremities are defined according to the arrows in the blended surface's preview.

5. Set the tension type using the **Tension** tab.
   It defines the tension of the blend at its limits.

   It can be constant or linear, and can be set for each limit independently.



6. Click OK.

The surface (identified as Blend.xxx) is added to the specification tree.

- Selecting a support is not compulsory.
- You can create closing points using the contextual menu on the **First** or **Second closing point** fields in the dialog box, or using the contextual menu directly on one of the selected curves.
- Use the Replace, Remove, or Reverse buttons, to manage the selected elements (curves, support, closing and coupling points).
- You can also use the contextual menu on the texts displayed on the geometry to set the continuities, trim the supports or manage the curves and support in general.

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Performing Operations on Shape Geometry

Wireframe and Surface allows you to modify your design using techniques such as trimming, translating and rotating.

 Join geometry: select at least two curves or surfaces to be joined.

 Heal geometry: select at least two surfaces presenting a gap to be healed.

 Untrim an element: select a split element, and click the icon.

 Disassemble elements: select a multi-cell element, and choose the disassembling mode.

 Split geometry: select the element to be split and a cutting element.

 Trim geometry: select two elements to be trimmed and specify which side of element

 Create boundary Curves: select a surface's edge, set the propagation type, and re-define the curve limits if needed.

 Extract geometry: select an element's edge or face and click the icon

 Translate geometry: select an element, a translation direction (line, plane or vector), specify the translation distance

 Rotate geometry: select an element, a line as the rotation axis, and specify the rotation angle

 Perform a symmetry: select an element, then a point, line, or plane as reference element

 Transform geometry by scaling: select an element, then a point, plane, or planar surface as reference element, and specify the scaling ratio

 Transform geometry by affinity: select an element to be transformed, specify the axis system characteristics, and the enter the affinity ratio values

 Transform geometry into a new axis-system: select an element to be transformed, specify the axis system characteristics, and the enter the affinity ratio values

 Create the nearest sub-element: select the **Insert -> Operations -> Near** menu item, the element made of several sub-elements, then a reference element whose position is close to the sub-element to be created


Extrapolate curves: select a curve endpoint then the curve itself, specify the extrapolation limit (length value or limiting surface/plane), and specify the continuity constraints (tangent/curvature)


Extrapolate surfaces: select a surface boundary then the surface itself, specify the extrapolation limit (value or limiting surface/plane), and specify the extremities constraints (tangent/normal)


Invert geometry orientation: select the **Insert -> Operations -> Invert Orientation** menu item, then the surface or curve whose orientation is to be inverted, click the orientation arrow, and click Invert Orientation again to accept the inverted element.

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Joining Surfaces or Curves

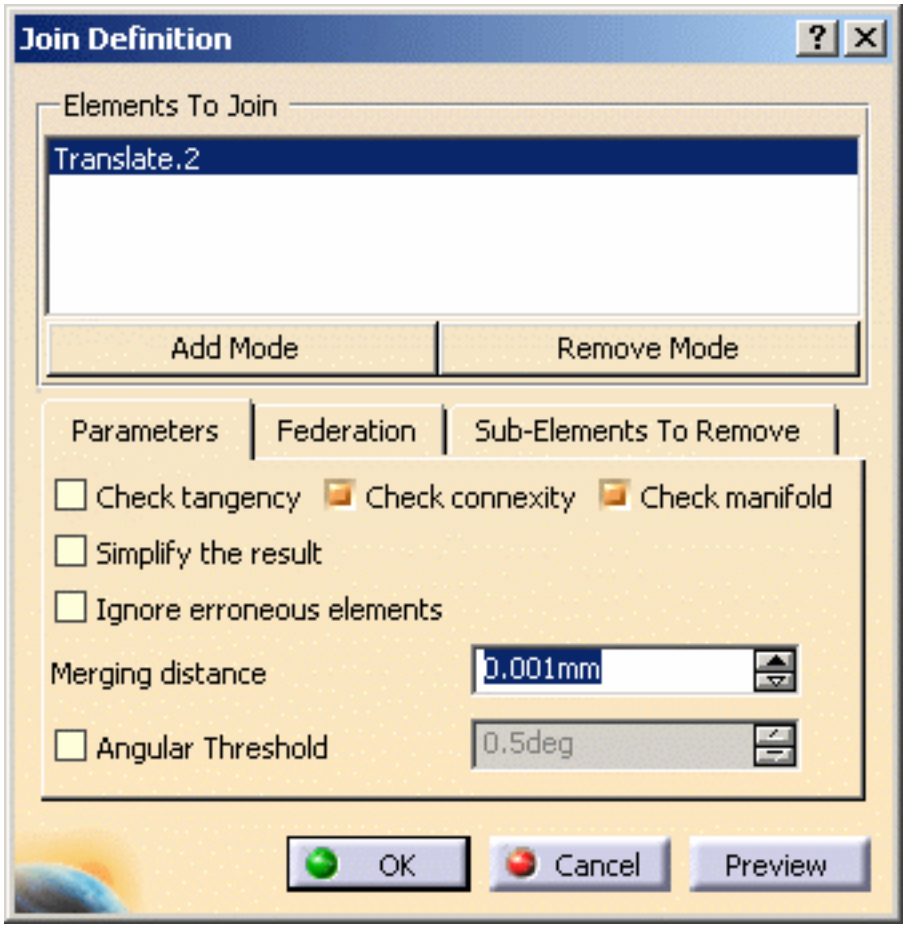
This task shows how to join at least two surfaces or two curves.

The surfaces or curves to be joined must be adjacent.

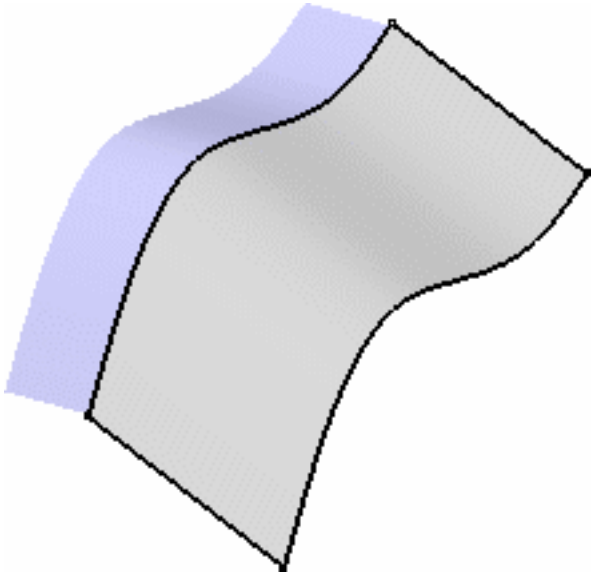
Open the Join1.CATPart document.

1. Click the **Join** icon.

   The Join Definition dialog box appears.



2. Select the surfaces or curves to be joined.
3. You can edit the list of elements to be joined:



- by selecting elements in the geometry:
  - **Standard selection** (no button clicked):
    when you click an unlisted element, it is added to the list

when you click a listed element, it is removed from the list

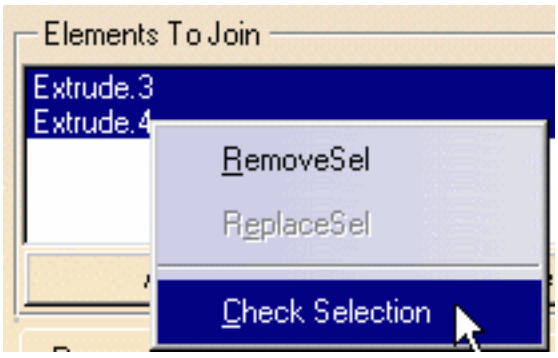
- **Add Mode**:
  when you click an unlisted element, it is added to the list
  when you click a listed element, it remains in the list
- **Remove Mode**:
  when you click an unlisted element, the list is unchanged
  when you click a listed element, it is removed from the list

- by selecting an element in the list then using the **Remove \Replace** contextual menu items.

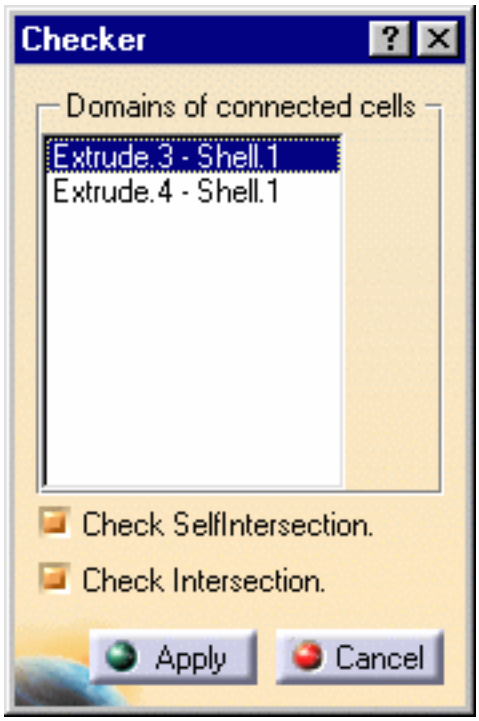
If you double-click the **Add Mode** or **Remove Mode** button, the chosen mode is permanent, i.e. successively selecting elements will add/remove them. However if you click only once, only the next selected element is added or removed.
You only have to click the button again, or click another one, to deactivate the mode.

4. Right-click the elements from the list and choose the **Check Solution** command.

   This let's you check whether any element to be joined presents any intersection (i.e. at least one common point) with other elements prior to creating the joined surface:

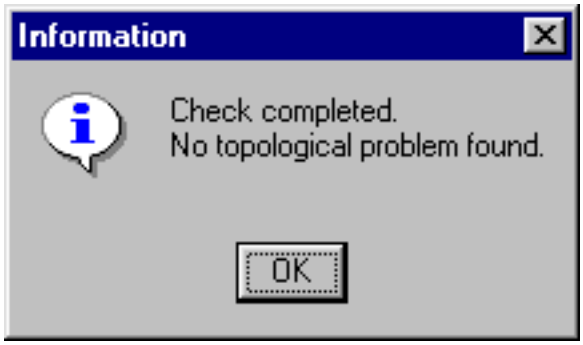


The Checker dialog box is displayed, containing the list of domains (i.e. sets of connected cells) belonging to the selected elements from the **Elements To Join** list.

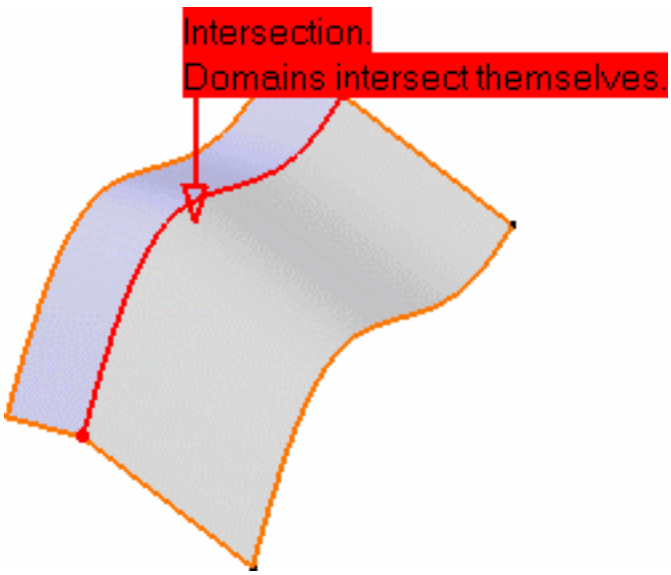


5. Click Apply.

- An Information message is issued when no intersection is found.



- When an element is self-intersecting, or when several elements intersect, a text is displayed on the geometry, where the intersection is detected.



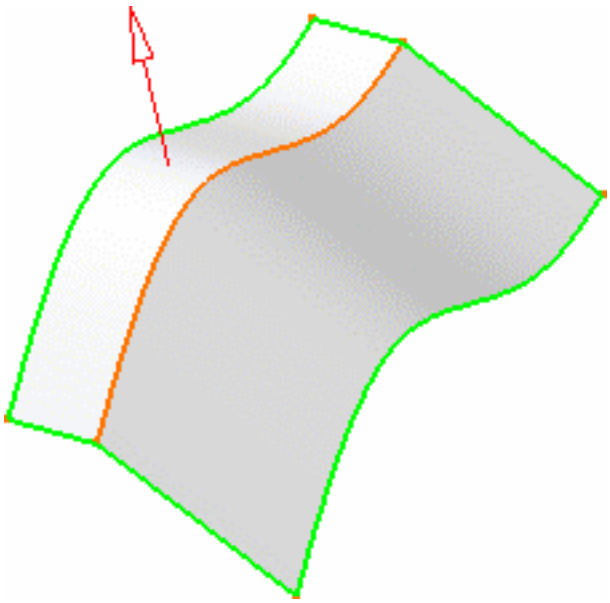
6. Click Cancel to return to the Join Definition dialog box.

7. Right-click the elements from the list and choose the **Propagate** command. It allows the selection of elements of same dimension to be added to the **Elements To Join** list.

- The initial element to propagate cannot be a sub-element
- Forks stop the propagation
- Intersections are not detected

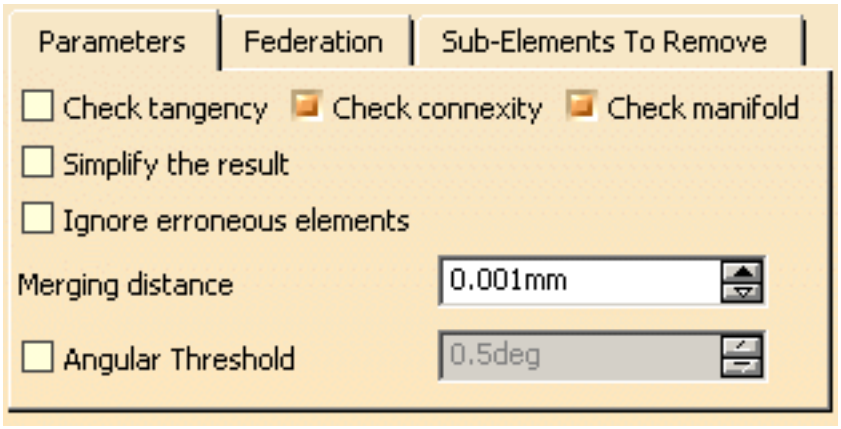
8. Click Apply in the Join Definition dialog box.

   The joined element is previewed, and its orientation displayed. Click the arrow to invert it if needed.



The join is oriented according to the first element in the list. If you change this element, the join's orientation is automatically set to match the orientation of the new topmost element in the list.

9. Check the **Check tangency** button to find out whether the elements to be joined are tangent. If they are not, and the button is checked, an error message is issued.



10. Check the **Check connexity** button to find out whether the elements to be joined are connex. If they are not, and the button is checked, an error message is issued indicating the number of connex domains in the resulting join.
When clicking Apply, the free boundaries are highlighted, and help you detect where the joined element is not connex.

11. Check the **Check manifold** button to find out whether the resulting join is manifold.

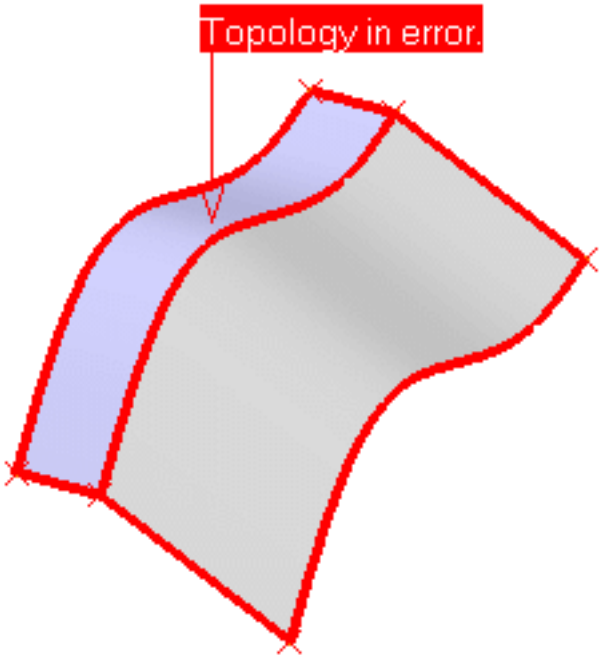
The **Check manifold** button is only available with curves. Checking it automatically checks the **Check connexity** button.

The **Simplify the result** check button allows the system to automatically reduce the number of elements (faces or edges) in the resulting join whenever possible.

The **Ignore erroneous elements** check button lets the system ignore elements that would not allow the join to be created.

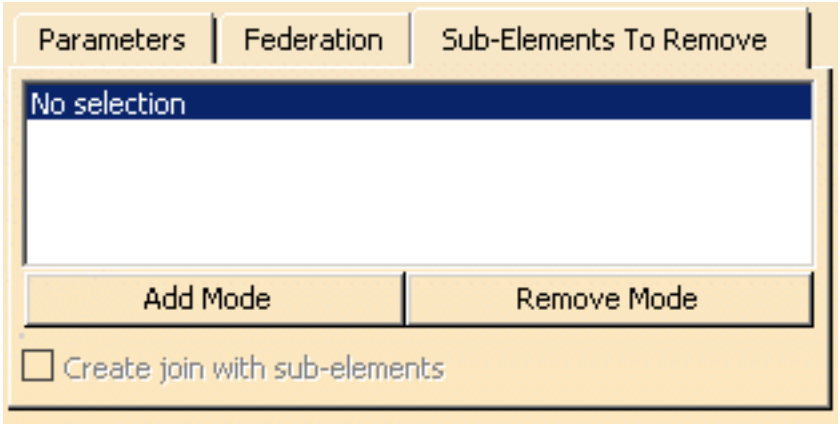
9. You can also set the tolerance at which two elements are considered as being only one using the **Merging distance**.

10. Check the **Angle Tolerance** button to specify the angle value below which the elements are to be joined.
If the angle value on the edge between two elements is greater than the **Angle Tolerance** value, the elements are not joined. This is particularly useful to avoid joining overlapping elements.



If the edges or the faces have a angular threshold higher than the predefined value, a text is displayed on the geometry indicating the error type.
You can either deactivate the check box or increase the value of the angular threshold, or remove all the elements or sub-elements that are in error.

11. Click the **Sub-Elements To Remove** tab to display the list of sub-elements in the join.



These sub-elements are elements making up the elements selected to create the join, such as separate faces of a surface for example, that are to be removed from the join currently being created.

You can edit the sub-elements list as described above for

the list of elements
to be joined.

12. Check the **Create join with sub-elements** option to create a second join, made of all the sub-elements displayed in the list, i.e. those that are not to be joined in the first join.
This option is active only when creating the first join, not when editing it.

13. Click **OK** to create the joined surface or curve.

The surface or curve (identified as Join.xxx) is added to the specification tree.

- Sometimes elements are so close that it is not easy to see if they present a gap or not, even though they are joined. Check the **Surfaces' boundaries** option from the **Tools -> Options** menu item, **General, Display**, **Visualization** tab.
- Once the Join.xxx element has been created, you can use the **Check** contextual menu from the specification tree. In this case however it verifies the connexity of all the sub-elements making up the joined surface. This is particularly useful when many elements have been joined, so has to highlight in the geometry which sub-element is not connected to the other ones, thus allowing you to rework the geometry if needed.

Up Next

Healing Geometry

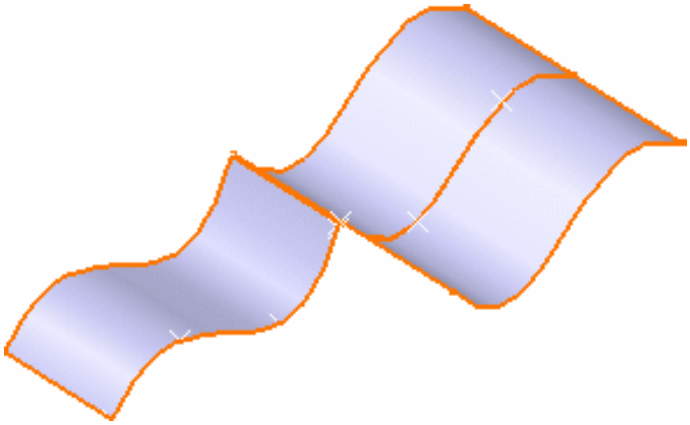
This task shows how to heal surfaces, that is how to fill any gap that may be appearing between two surfaces.
This command can be used after having checked the connections between elements for example, or to fill slight gaps between joined surfaces.

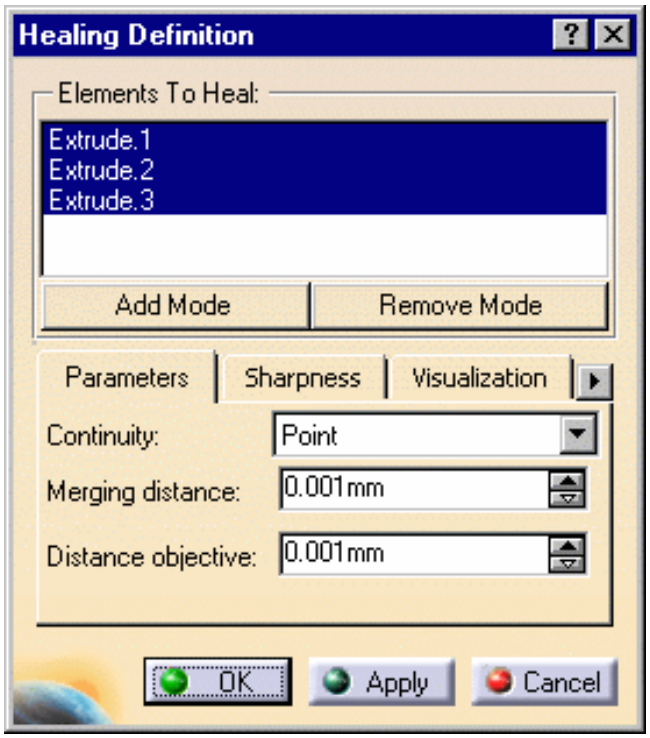
Open the Healing1.CATPart document.
Display the Join-Healing toolbar by clicking and holding the arrow from the **Join** icon.

1. Click the **Healing** icon.

   The Healing Definition dialog box appears.

2. Select the surfaces to be healed.





3. You can edit the list of elements in the definition list:

- by selecting elements in the geometry:
  - **Standard selection** (no button clicked):
    when you click an unlisted element, it is added to the list
    when you click a listed element, it is removed from the list
  - **Add Mode**:
    when you click an unlisted element, it is added to the list
    when you click a listed element, it remains in the list
  - **Remove Mode**:
    when you click an unlisted element, the list is unchanged
    when you click a listed element, it is removed from the list
- by selecting an element in the list then using the **Remove\Replace** contextual menu items.

If you double-click the **Add Mode** or **Remove Mode** button, the chosen mode is permanent, i.e. successively selecting elements will add/remove them. However, if you click only once, only the next selected element is added or removed.
You only have to click the button again, or click another one, to deactivate the mode.

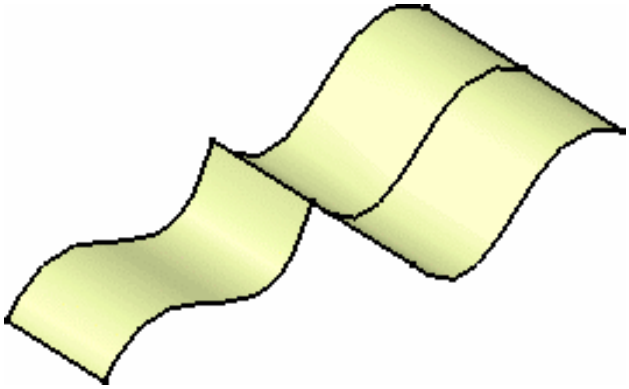
4. From the **Parameters** tab, define the distance below which elements are to be healed, that is deformed so that there is no more gap, using the **Merging distance** as described in Joining Geometry.
   Elements between which the gap is larger than the indicated value are not processed.

   In our example, we increase it to 1mm.

You can also set the **Distance objective**, i.e. the maximum gap allowed between two healed elements. By default it is set to 0.001 mm, and can be increased to 0.1 mm.

5. Click **OK** to create the healed surfaces.

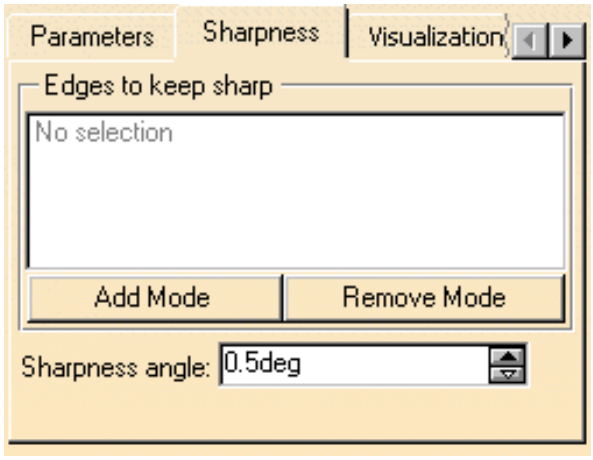
   The surface (identified as Heal. xxx) is added to the specification tree.

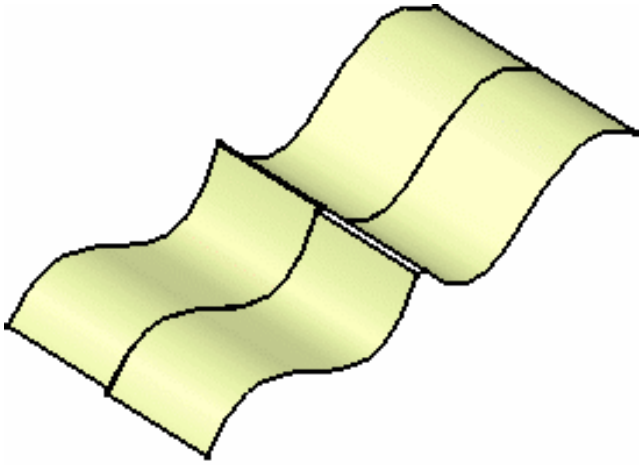


- Check the **Surfaces' boundaries** option from the **Tools -> Options** menu item, **General -> Display -> Visualization** tab to display the boundaries. This may be especially useful when selecting, and also to identify gaps.

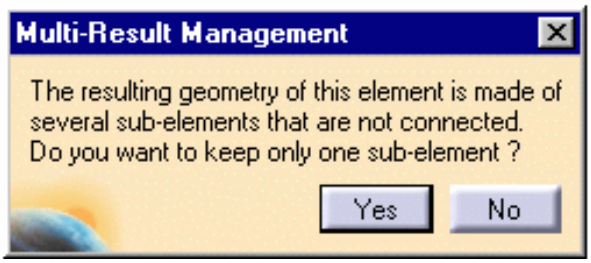
- Provided the **Tangent** mode is active, you can retain sharp edges, by clicking the **Sharpness** tab, and selecting one or more edges.
  You can edit the list of edges as described above for the list of elements to be healed.

- The **Sharpness angle** allows to redefine the limit between a sharp angle and a flat angle. This can be useful when offsetting the resulting healed geometry for example. By default this angle value is set to 0.5 degree.

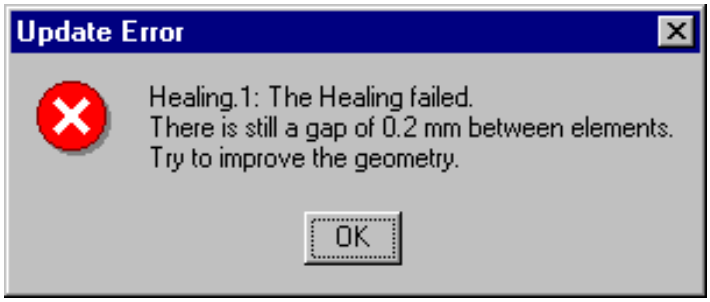




- In some cases, depending on the geometry configuration and the set parameters, the Multi-Result Management dialog box is displayed.
  Click **No** or refer to Creating the Nearest Entity of a Multiple Element.



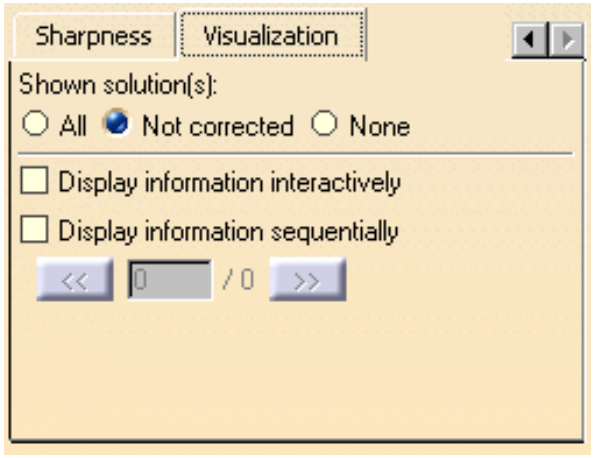
When the healing failed, the update error dialog box appears.
Click OK to improve the geometry.



The erroneous elements are displayed on the geometry.

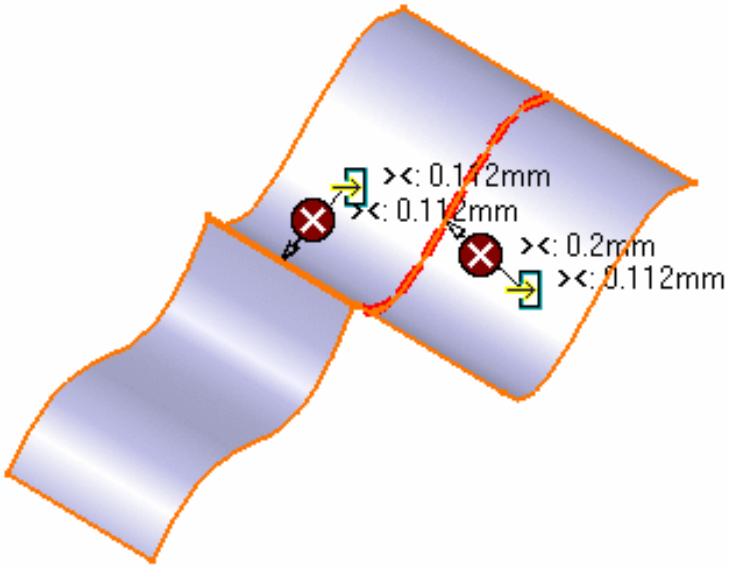
The **Visualization** tab enables you to better understand the discontinuities in the model and the results of the healing action.

It lets you define the way the messages are displayed on the smoothed element.

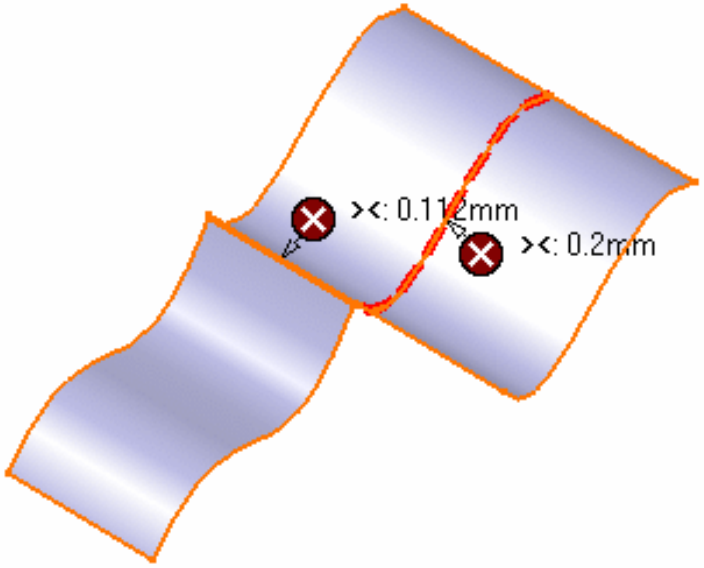


You can choose to see:

- **All** the messages, that is to say the messages indicating where the discontinuity remains as well as those indicating where the discontinuity type has changed (in point (**><**) and tangency (**^**))



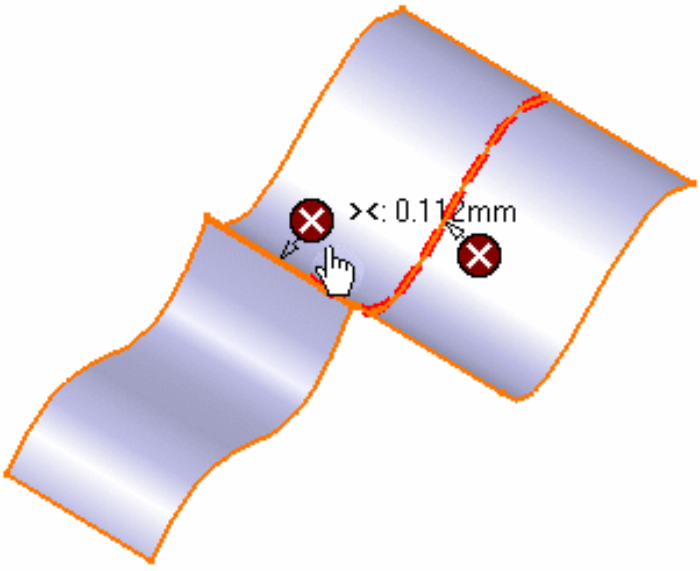
- only the messages indicating where the discontinuity is **Not corrected** and still remains



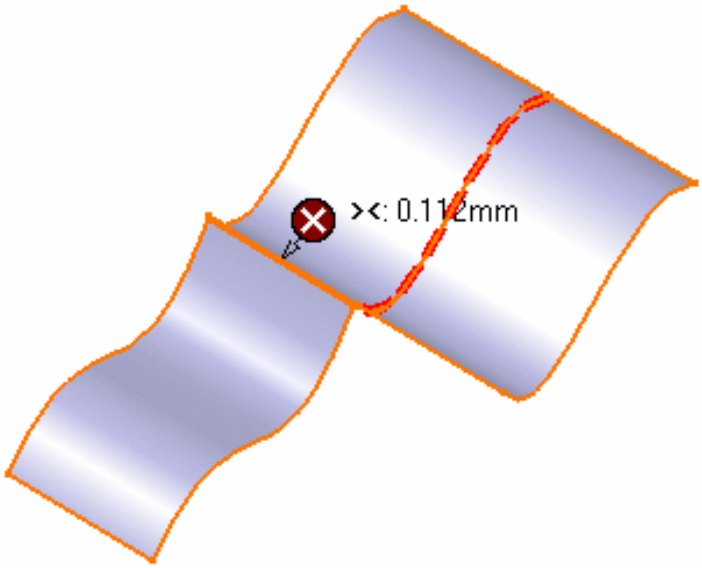
- **None** of the messages

You can also choose to see:

- **Display information interactively**: only the pointers in the geometry are displayed, above which the text appears when passing the pointer



- **Display information sequentially**: only one pointer and text are displayed in the geometry, and you can sequentially move from one pointer to another using the backward/forward buttons





Back Up Next

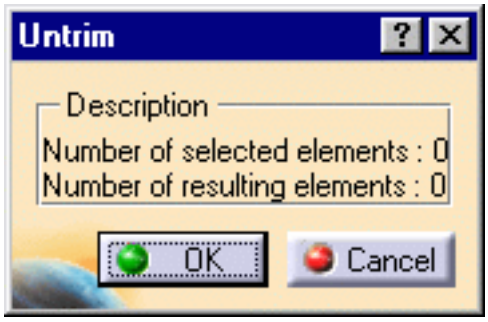
Restoring a Surface

In this task you will learn how to restore the limits of a surface when it has been split using the **Break** icon (see Splitting Geometry).

Open the Untrim1.CATPart document.

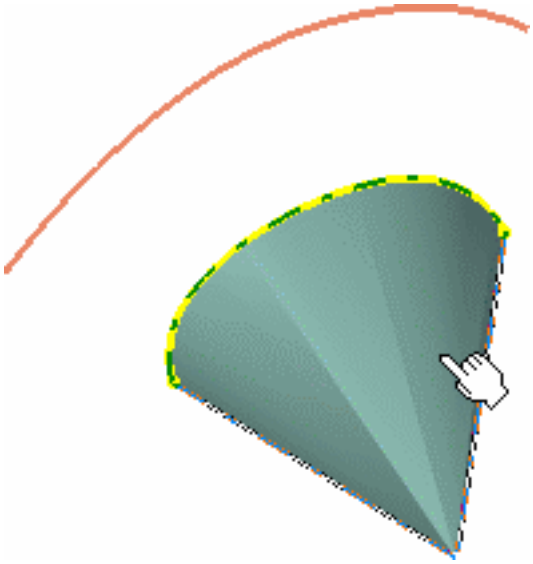
1. Click the **Untrim** icon in the **Join-Healing** Modification toolbar.

   The Untrim dialog box is displayed.



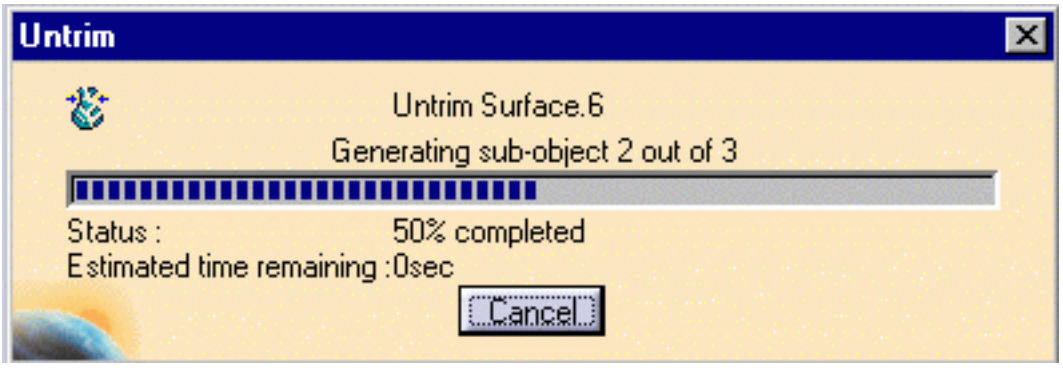
2. Select the surface which limits should be restored.

   The dialog box is updated accordingly.

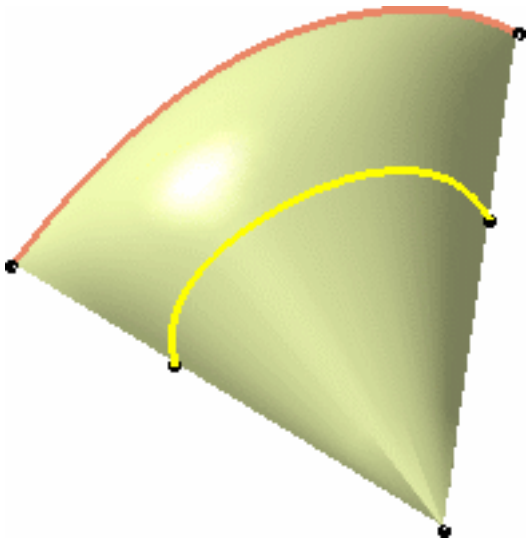


3. Click OK in the dialog box.

   A progression bar is displayed, while the surface is restored.
   It automatically disappears once the operation is complete (progression at 100%).



The initial surface is automatically restored.



The restored surface or curve is identified as Surface Untrim.xxx or Curve Untrim.xxx.

If the surface has been trimmed several times, it is the initial surface which is restored. To partially untrim the surface, you need to use the **Undo** command right after the trim.

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Disassembling Elements

In this task you will learn how to disassemble multi-cell bodies into mono-cell bodies.

Open the Disassembling1.CATPart document, or any document containing a multi-cell element.

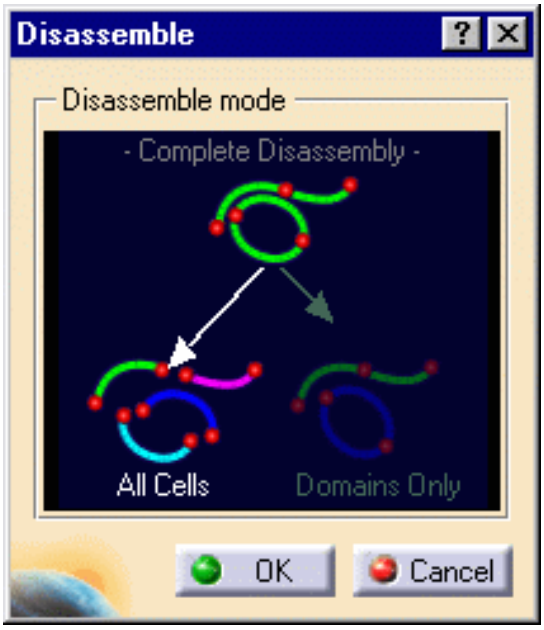
1. Select the element to be disassembled.
   You can select only an edge of a surface, the system recognizes the whole element to be disassembled.

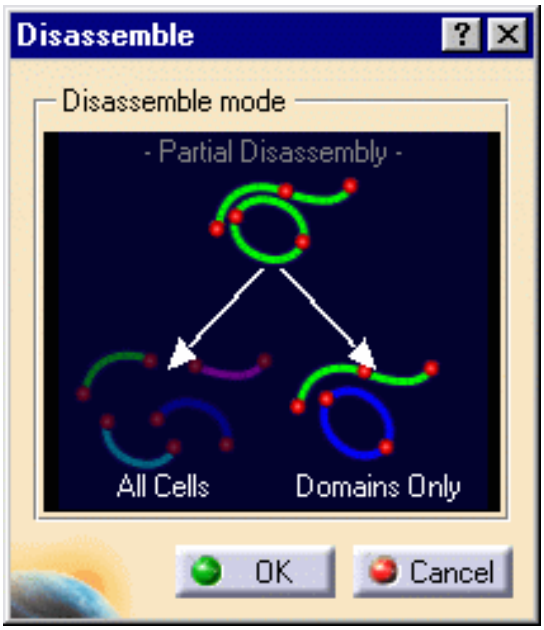
   Here we selected the join made of three elements, each made of several cells.



2. Click the **Disassemble** icon in the **Join-Healing** toolbar.

   The Disassemble dialog box is displayed.

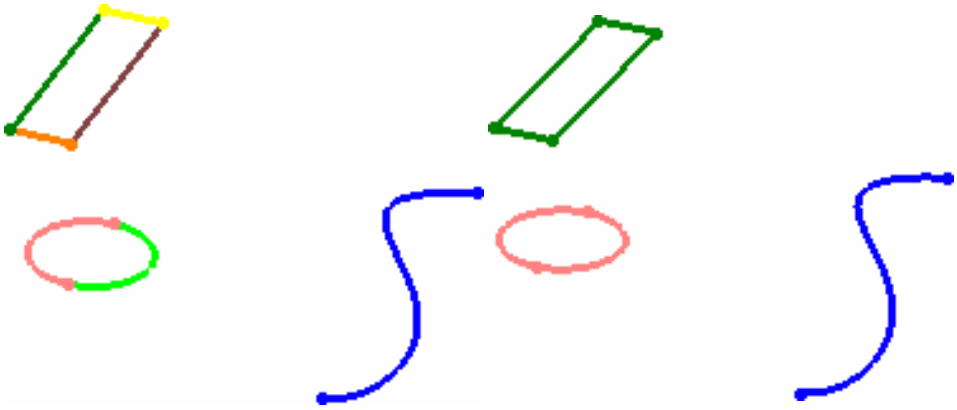




3. Choose the disassembling mode:

- **All Cells**: all cells are disassembled, i.e. for all the selected element, a separate curve is created for each cell.
- **Domains Only**: elements are partially disassembled, i. e. each element is kept as a whole if its cells are connex, but is not decomposed in separate cells. A resulting element can be made of several cells.

  In the illustrations, we have colored the resulting curves for better identification.

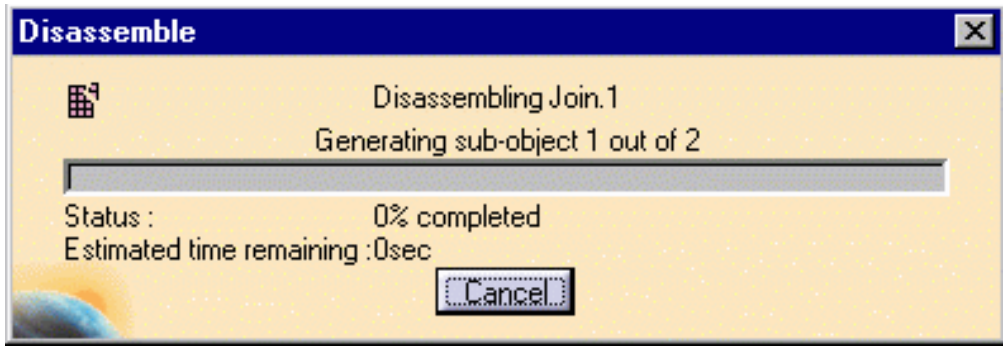


Results when disassembling all cells (seven curves are created)

Results when disassembling domains only (three curves are created)

4. Click OK in the dialog box.

A progression bar is displayed, while the surface is being disassembled. It automatically disappears once the operation is complete (progression at 100%).



The selected element is disassembled, that is to say independent elements are created, that can be manipulated independently.

Multi-selection is available.



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Splitting Geometry

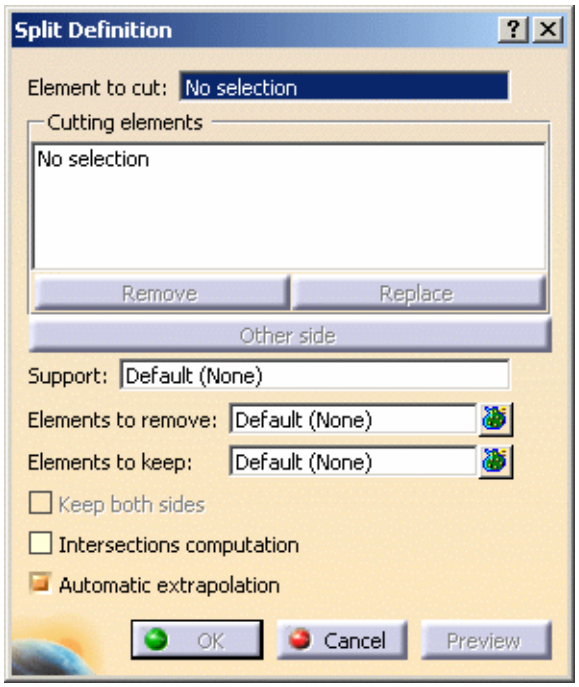
This task shows how to split a surface or wireframe element by means of a cutting element.
You can split:

- a wireframe element by a point, another wireframe element or a surface
- a surface by a wireframe element or another surface.

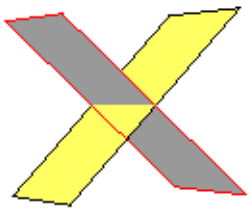
Open the Split1.CATPart document.

1. Click the **Split** icon .

   The Split Definition dialog box appears.



2. Select the element to be split.

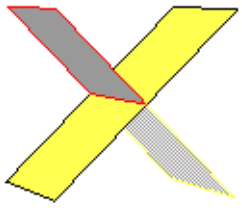


You should make your selection by clicking on the portion that you want to keep after the split.

3. Select the cutting element.

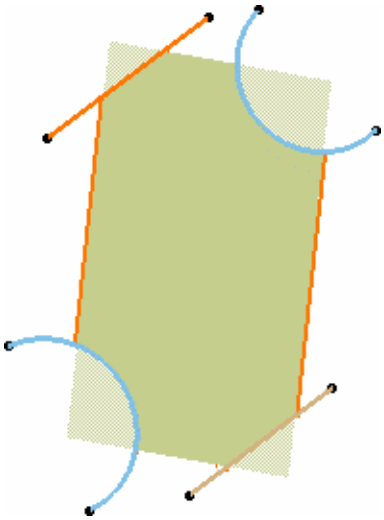
   A preview of the split appears. You can change the portion to be kept by selecting that portion.

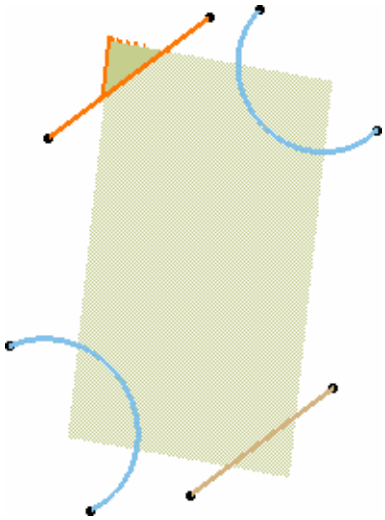
   You can also select the portion to be kept by clicking the **Other side** button.



You can select several cutting elements. In that case, note that the selection order is important as the area to be split is defined according to the side to be kept in relation to current splitting element.

In the illustrations below, the top-left line is the first splitting element. In the left illustration it defines an area that intersects with the other three splitting curves, and in the illustration to the right, these three elements are useless to split the area defined by the first splitting element.

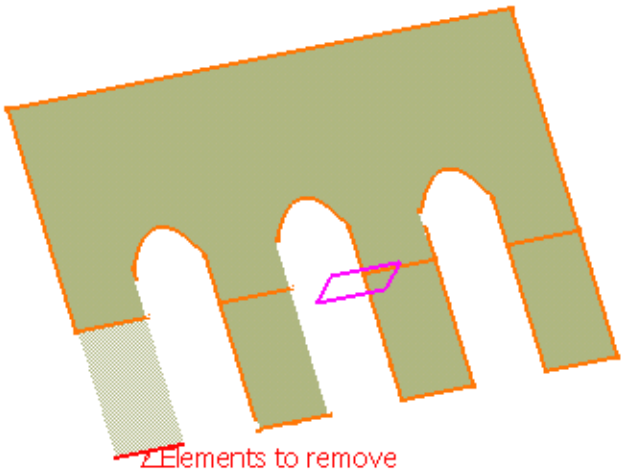




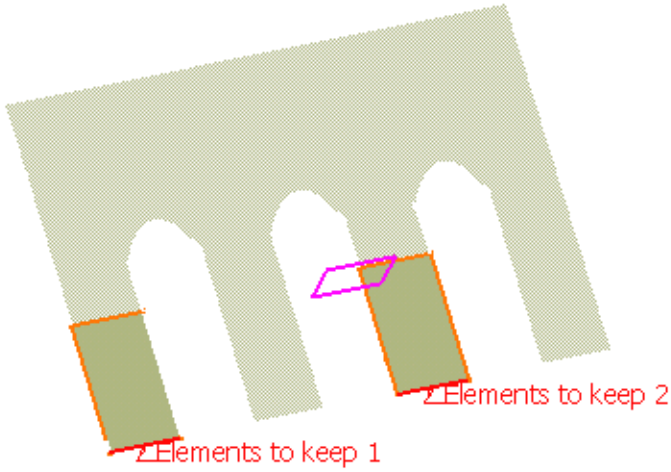
Would you need to remove, or replace, one of these cutting elements, select it from the list and click the **Remove** or **Replace** button.

The **Elements to remove** and **Elements to keep** options allows to define the portions to be removed or kept when performing the split operation.

- Click in the field of your choice to be able to select the elements in the 3D geometry.
- Right-click in the field either to clear the selection or display the list of selected elements.

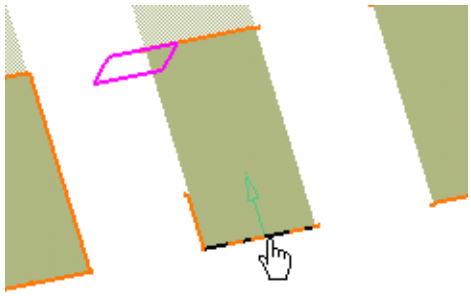


Only the selected element is removed.
All other elements are kept.



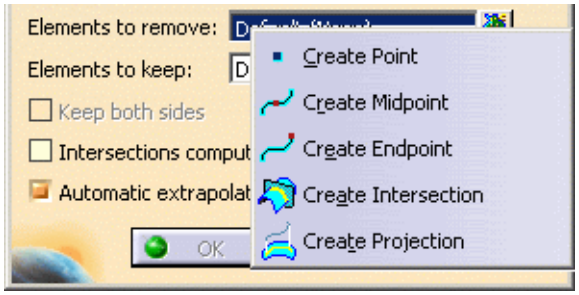
The selected elements are kept.
All other elements are removed.

- You must select sub-elements as elements to keep or to remove; otherwise, a warning message is issued.



- You can also select a point to define the portion to keep or to remove.

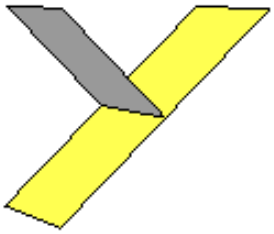
  A contextual menu is available on the **Elements to remove** and **Elements to keep** fields.



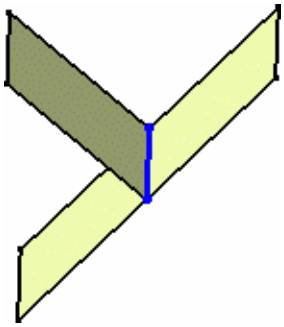
You do not need to select elements to keep if you already selected elements to remove and vice-versa.

4. Click OK to split the element.

   The created element (identified as Split.xxx) is added to the specification tree.

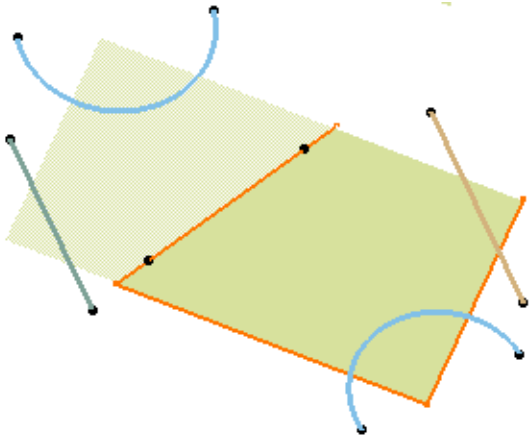


- Check the **Keep both sides** option to retain the other side of the split element after the operation. In that case it appears as aggregated under the first element.



- Check the **Intersection computation** button to create an aggregated intersection when performing the splitting operation. This element will be added to the specification tree as Intersect.x

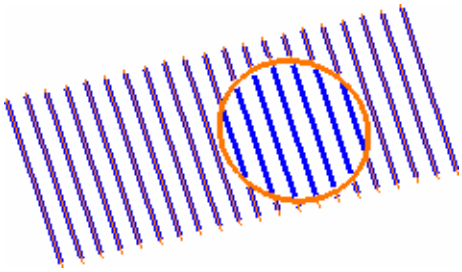
- Uncheck the **Automatic extrapolation** button if do not you want the automatic extrapolation of the cutting curve.
  If the **Automatic extrapolation** button is unchecked, an error message is issued when the cutting element needs to be extrapolated, and the latter is highlighted in red in the 3D geometry.
  To be able to split the surface, check the **Automatic extrapolation** button.



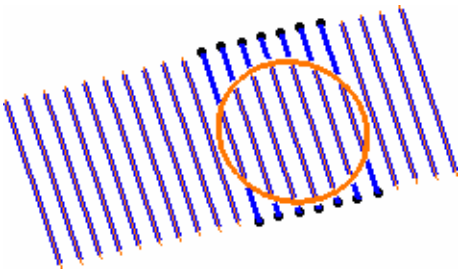
This option is available in the case of a split surface/curve or surface/surface.

- When splitting a wire (curve, line, sketch and so forth) by another wire, you can select a support to define the area that will be kept after splitting the element. It is defined by the vectorial product of the normal to the support and the tangent to the splitting element.
  This is especially recommended when splitting a closed wire.

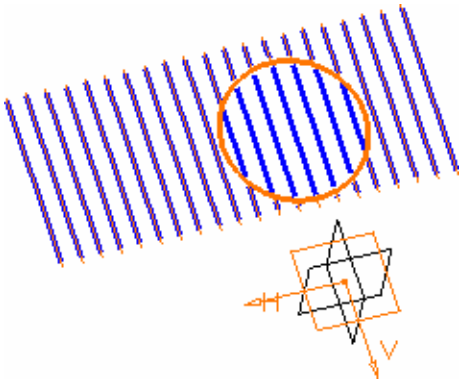
The non disconnected elements of the element to cut are kept in the result of the split.



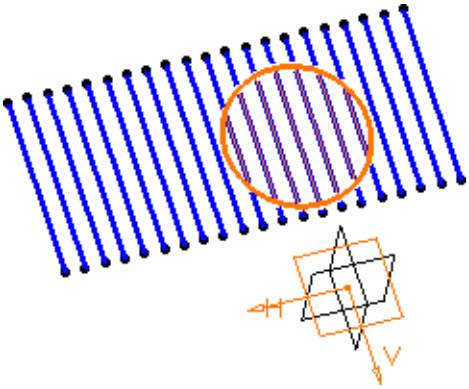
Splitting with no support selected: first solution



Splitting with no support selected: second solution



Splitting with a selected support (xy plane): first solution

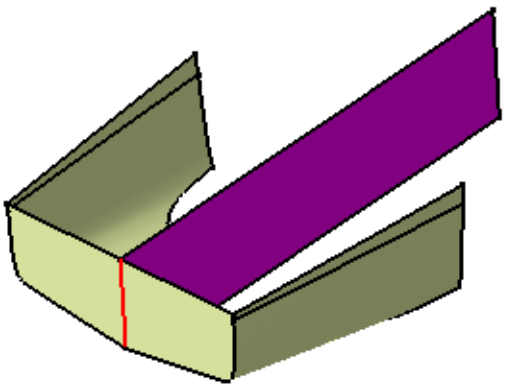


Splitting with a selected support (xy plane): second solution

- You can create a Join as the splitting element, by right-clicking in the **Cutting Elements** field and choosing the **Create Join** item.

If you split a surface and you keep both sides by joining the resulting splits, you cannot access the internal sub-elements of the join: indeed, splits result from the same surface and the cutting elements are common.

Avoid splitting geometry when the intersection between the element to cut and the cutting element is merged with an edge of the element to cut.
In that case, you can use the **Elements to remove** and **Elements to keep** options to remove the positioning ambiguity.

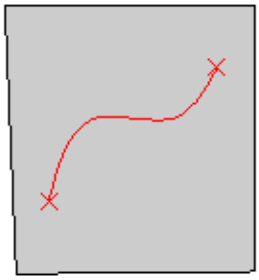


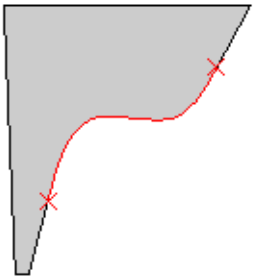
Splitting surface/curve or surface/surface

The following steps explain how split a surface by a curve or another surface.

Split surface/curve

1. First, the cutting element (the curve) is laid down the surface.
2. Then, the result of step 1 is tangentially extrapolated in order to split the surface correctly (as shown in following figure). However, when this extrapolation leads to the intersection of the cutting element with itself prior to fully splitting the initial element, an error message is issued as there is an ambiguity about the area to be split.



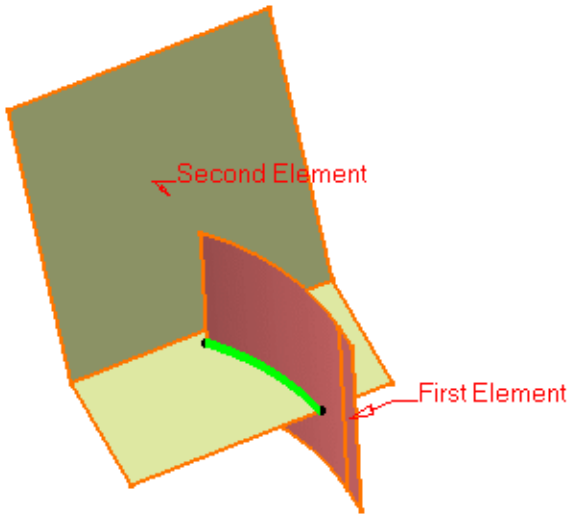


If the cutting element does not reach the free edges of the element to cut, an extrapolation in tangency is performed using the part of the cutting element that lays down the surface.

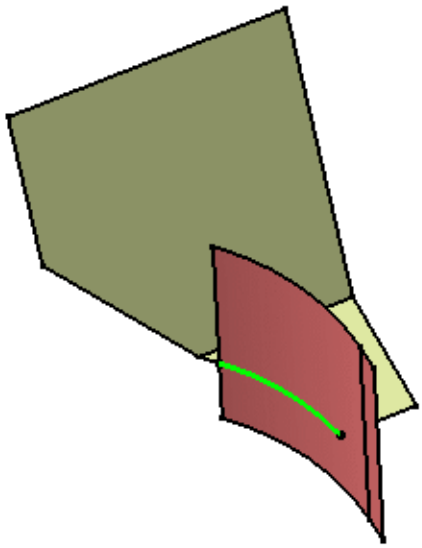
Split surface/surface

Open the Split2.CATPart document.

1. First, an intersection (the green wire) is created between the two elements (the surfaces).



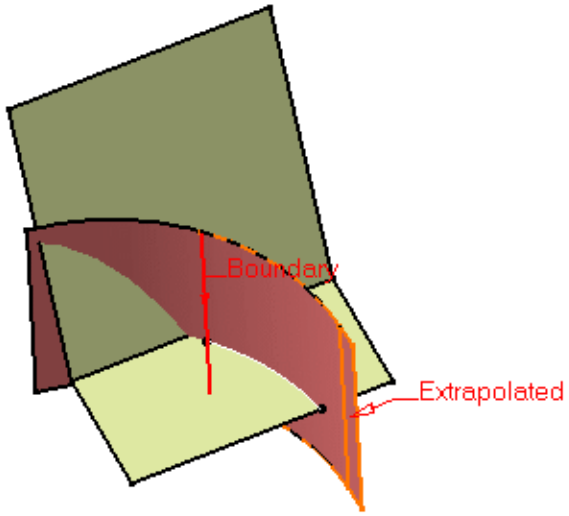
2. Then, the result of the intersection is automatically extrapolated in tangency up to the closest free edges of the element to cut.
   The result of the extrapolation is used as the cutting element and the split is created.



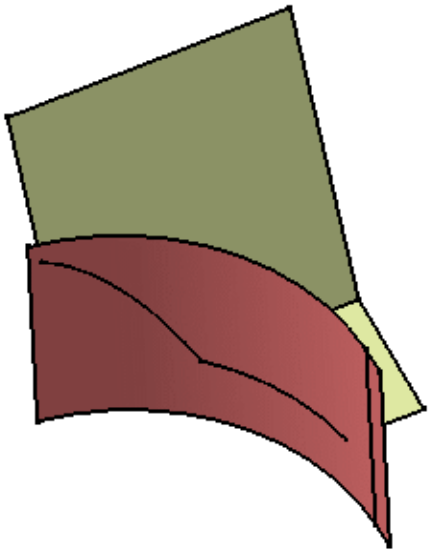
Please note that it is **not** the cutting element which is extrapolated but the result of the intersection.

If the result of the split is not what was expected, it is also possible to manually extrapolate the cutting element with the extrapolate feature before creating the split.

1. Extrapolate the cutting element (the red surface) in order to fully intersect the element to cut.



2. Then, use the extrapolated surface as the cutting element to split the surface.

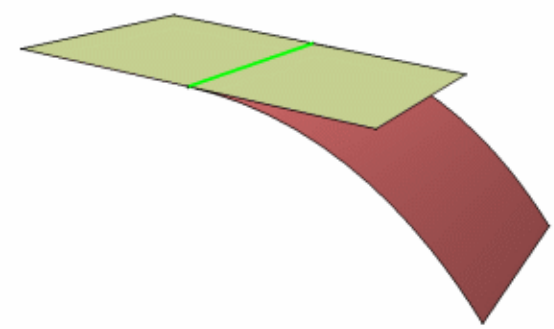


Avoid using input elements which are tangent to each other since this may result in geometric instabilities in the tangency zone.

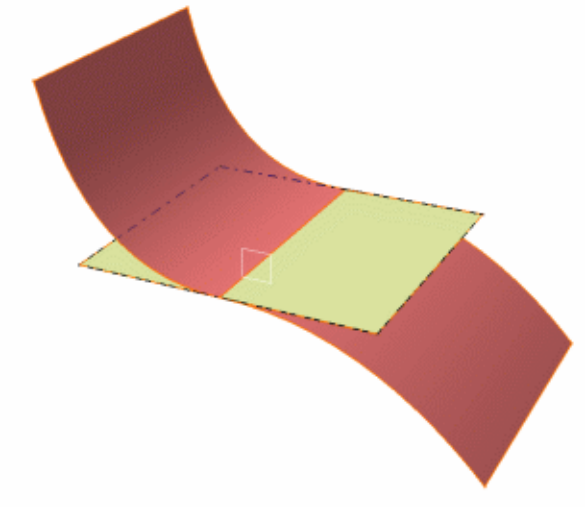
In case surfaces are tangent or intersect face edges, please process as follow in order to avoid indeterminate positioning.

Use the border edge of the cutting surface to split the element to cut:

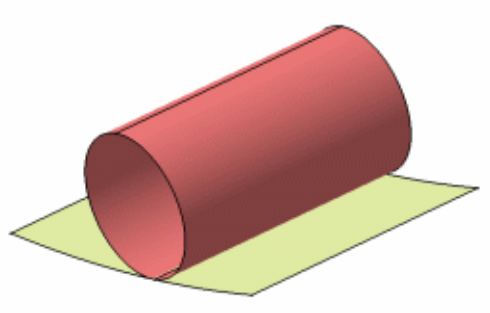
1. Delimit the boundary of the cutting surface
2. Project this boundary onto the surface to split
3. Use this projection as the cutting element



Steps 2 and 3 may be optional if the tangency constraint between the two surfaces has been clearly defined by the user during the surface creation.



The following cases should be avoided when possible (especially when the tangency constraint between the two surfaces has not been clearly defined by the user during the surface creation), as the result of the positioning is likely to be indeterminate and the result of the intersection to be unstable.



When these cases cannot be avoided, it is recommended, first to create the intersection between the two surfaces, then to split the element to cut with the resulting intersection. Doing so, the position can be properly defined but the instability of the result relating to the intersection remains.

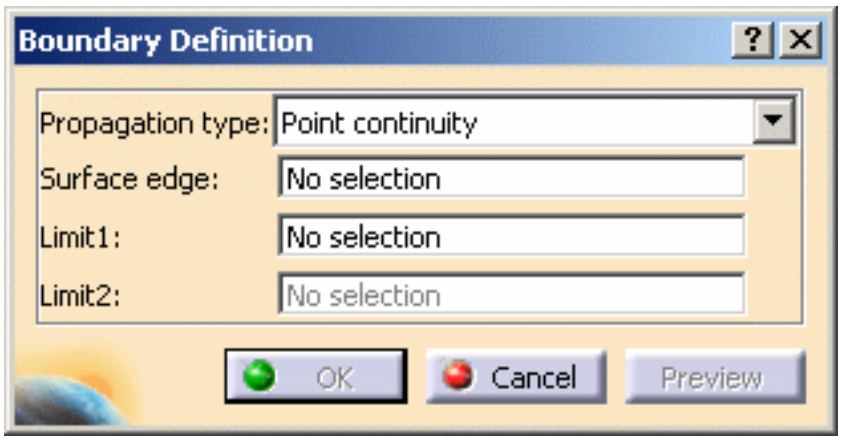
Creating Boundary Curves

This task shows how to create the boundary curve of a surface.

Open the Boundaries1.CATPart document.

1. Click the **Boundary** icon .

   The Boundary Definition dialog box appears.



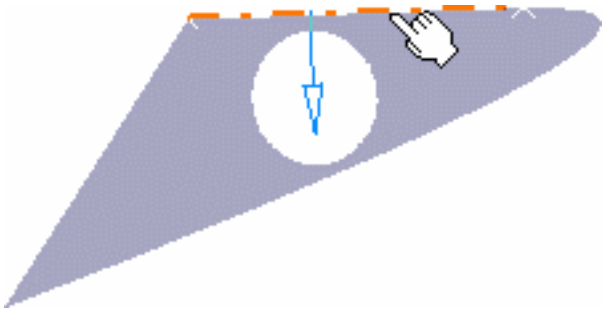
2. Use the combo to choose the **Propagation type**:

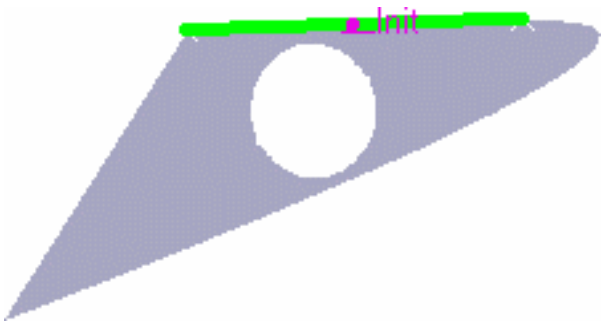
- **Complete boundary:** the selected edge is propagated around the entire surface boundary.
- **Point continuity:** the selected edge is propagated around the surface boundary until a point discontinuity is met.
- **Tangent continuity:** the selected edge is propagated around the surface boundary until a tangent discontinuity is met.
- **No propagation**: no propagation or continuity condition is imposed, only the selected edge is kept.

You can now select the propagation type before selecting an edge.

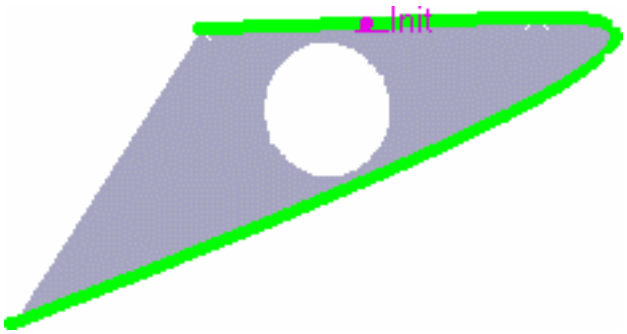
3. Select a Surface edge.

The boundary curve is displayed according to the selected propagation type.

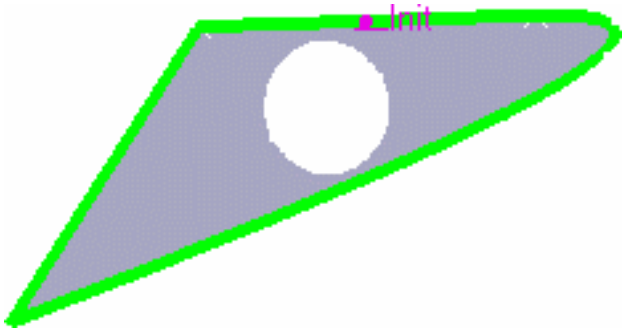




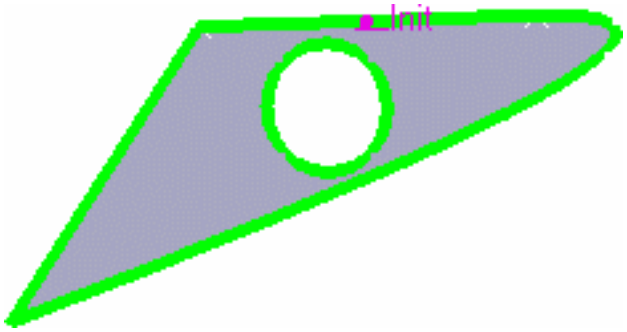
No propagation



Tangent continuity



Point continuity



Complete boundary

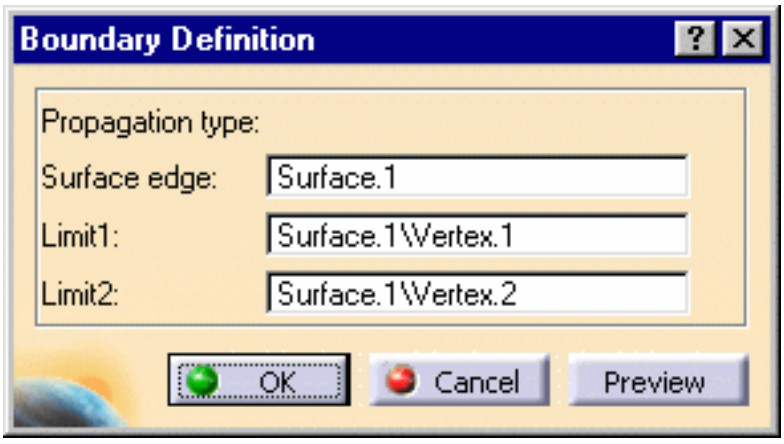
4. You can relimit the boundary curve by means of two elements.

If you relimit a closed curve by means of only one element, a point on curve curve for example, the closure vertex will be moved to the relimitation point, allowing this point to be used by other features.

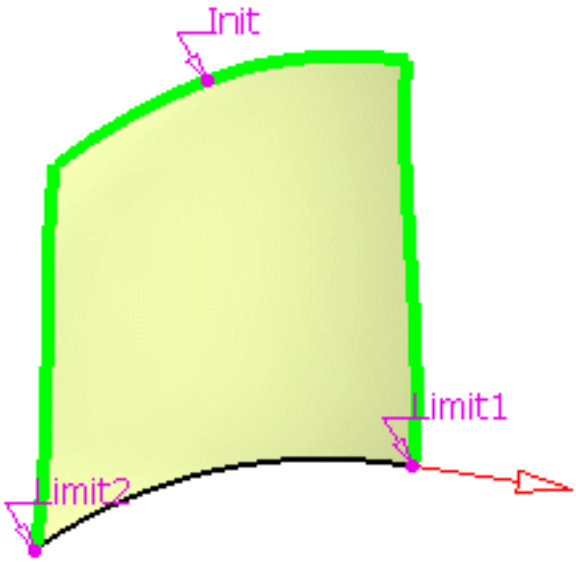
5. Click **OK** to create the boundary curve.

The curve (identified as Boundary.xxx) is added to the specification tree.

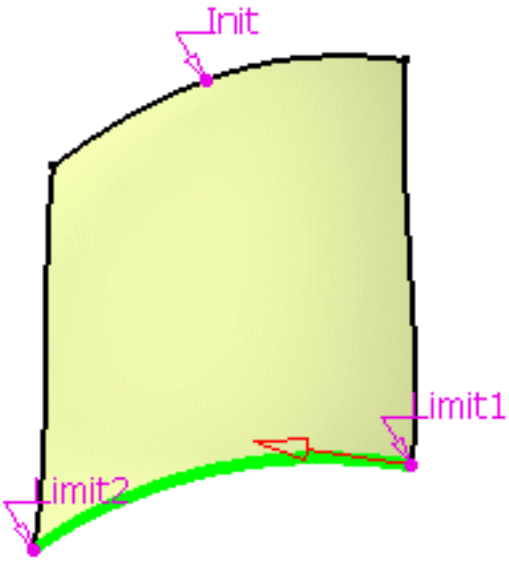
- If you select the surface directly, the Propagation type no longer is available, as the complete boundary is automatically generated.



Provided the generated boundary curve is continuous, you can still select limiting point to limit the boundary.



Using the arrows you can then invert the limited boundary.



If you select a curve which has an open contour, the Propagation type becomes available: choose the **No Propagation** type and select the curve again. The extremum points will define the boundary curve.

- You cannot copy/paste a boundary from a document to another. If you wish to do so, you need to copy/paste the surface first into the second document then create the boundary.

Extracting Geometry

This task shows how to perform an extract from elements (curves, points, solids, and so forth).

This may be especially useful when a generated element is composed of several non-connex sub-elements. Using the extract capability you can generate separate elements from these sub-elements, without deleting the initial element.

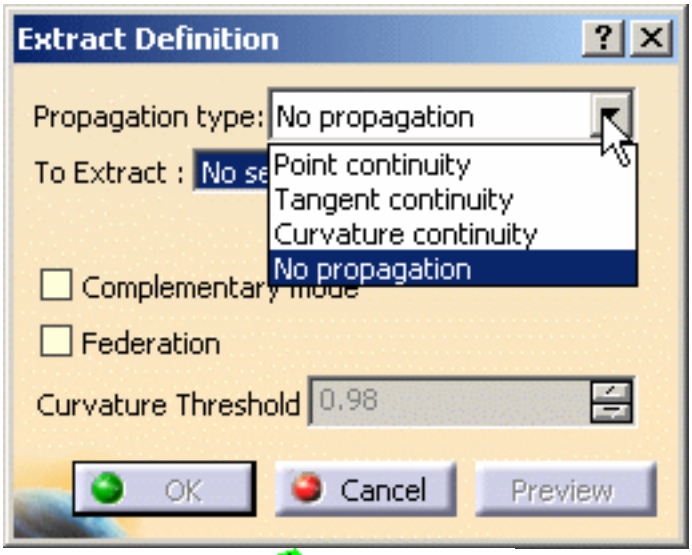
Open the Extract1.CATPart document.

1. Select an edge or the face of an element.

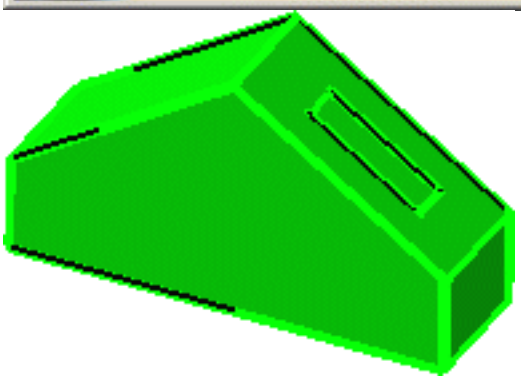
   The selected element is highlighted.

2. Click the **Extract** icon .

   The Extract Definition dialog box is displayed.

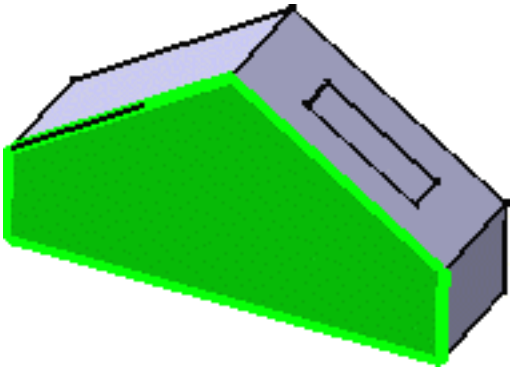


3. Choose the **Propagation** type:



- **Point continuity**: the extracted element will not have a hole.

- **Tangent continuity**: the extracted element will be created according to tangency conditions.



- **Curvature continuity**: the extracted element will be created according to curvature conditions.

  Define a **Curvature Threshold** value. For a curvature discontinuity: the value is a ratio between 0 and 1 which is defined as follows:

  if ||Rho1-Rho2|| / ||Rho2|| < (1-r)/r
  where Rho1 is the curvature vector on one side of the discontinuity, Rho2 the curvature vector on the other side, and r the ratio specified by the user;
  then the discontinuity is smoothed.

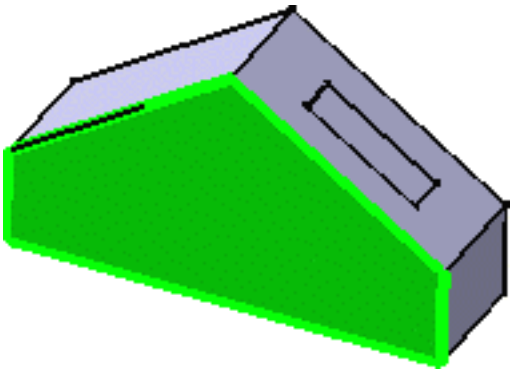
  For example, r=1 corresponds to a continuous curvature and r=0.98 to the model tolerance (default value). A great discontinuity will require a low r to be taken into account.



The extracted element must be a wire.

This option is only available with Generative Shape Design 2, it is not available with Wireframe and Surface.

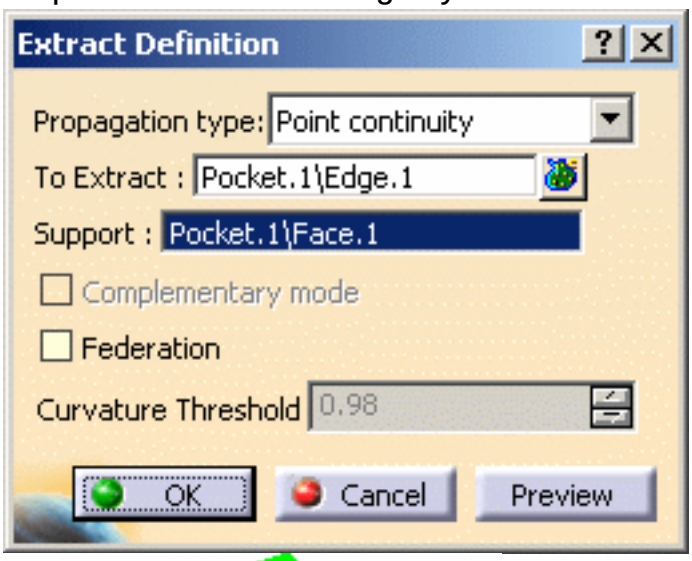
- **No propagation**: only the selected element will be created.



4. Click OK to extract the element.

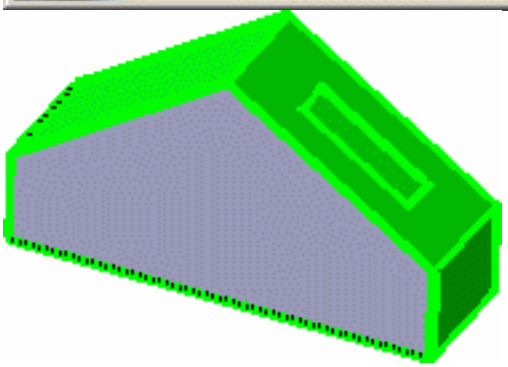
The extracted element (identified as Extract.xxx) is added to the specification tree.

When the result of an extract is not connex (during creation or edition) due to naming ambiguity, you can now select the part to keep to solve the ambiguity.



- If you extract an edge that you want to propagate, and there is an ambiguity about the propagation side, a warning is issued and you are prompted to select a support face. In this case, the dialog box dynamically updates and the **Support** field is added.

- The **Complementary mode** option, once checked, highlights, and therefore selects, the elements that were not previously selected, while deselecting the elements that were explicitly selected.



- In the current model, if you select an element using the Tangent or Point continuity as the Propagation type, a warning is issued and you have to select No propagation instead.
- If the selected element has a support face and is not a surface, even though the Complementary mode option is checked, the Complementary mode will not be taken into account for the extraction and the option will therefore be inactive. After the extraction, the option will be available again.
- Check the **Federation** button to generate groups of elements belonging to the resulting extracted element that will be detected together with the pointer when selecting one of its sub-elements. For further information, see Using the Federation Capability.
- Multi-selection is available. Refer to Selecting Using Multi-Output.
- You cannot copy/paste an extracted element from a document to another. If you wish to do so, you need to copy/paste the initial element first into the second document then perform the extraction.



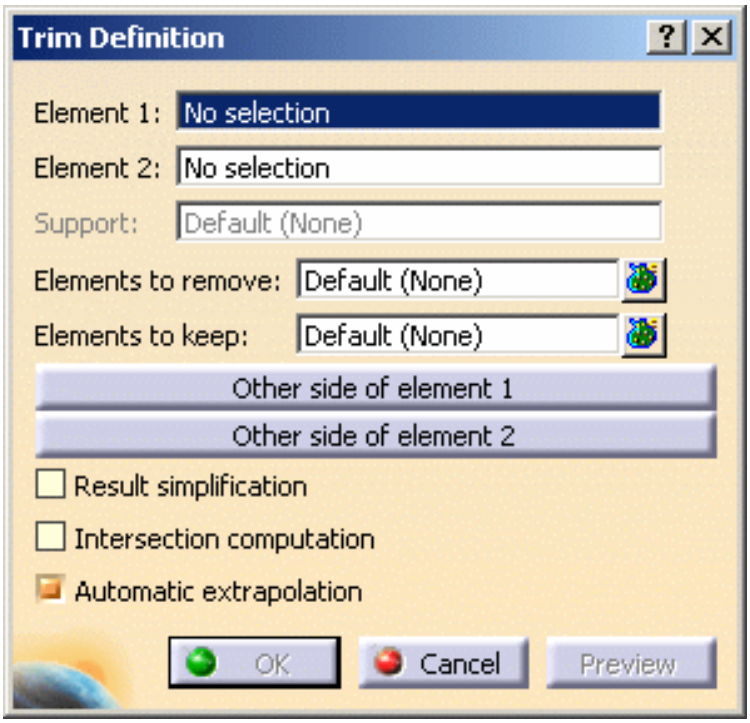
Trimming Geometry

This task shows how to trim two surfaces or two wireframe elements.

Open the Trim1.CATPart document.

1. Click the **Trim** icon .

   The Trim Definition dialog box appears.



2. Select the two surfaces or two wireframe elements to be trimmed.

   A preview of the trimmed element appears. You can change the portion to be kept by selecting that portion.

   You can also select the portions to be kept by clicking the **Other side of element 1** and **Other side of element 2** buttons.



3. Click **OK** to trim the surfaces or wireframe elements.

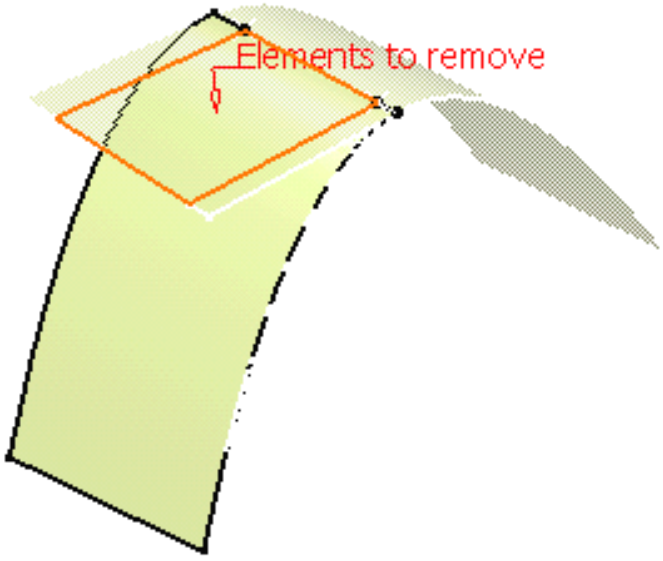
   The trimmed element (identified as Trim.xxx) is added to the specification tree.



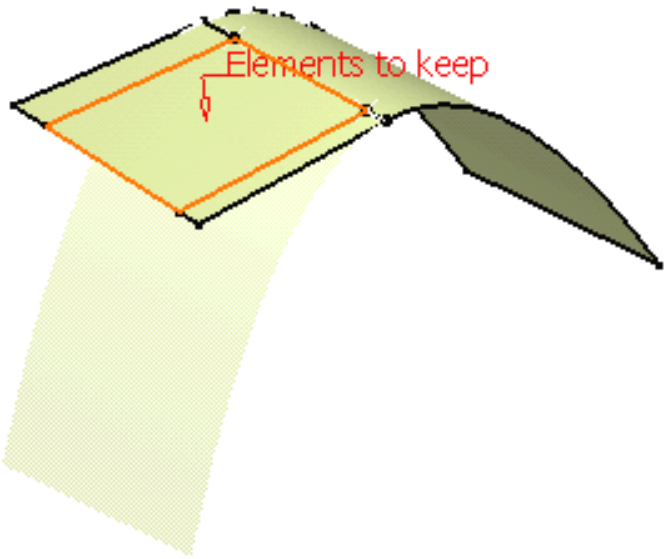
- You should make your selections by clicking on the portions that you want to keep after the trim.
- Please refer to the Splitting Geometry chapter in the case surfaces intersect face edges.

In case the elements to be trimmed are tangent, you are advised to use the **Elements to remove** and **Elements to keep** options to define the portions to be kept or removed.

- Click in the field of your choice to be able to select the elements in the 3D geometry.
- Right-click in the field either to clear the selection or display the list of selected elements.



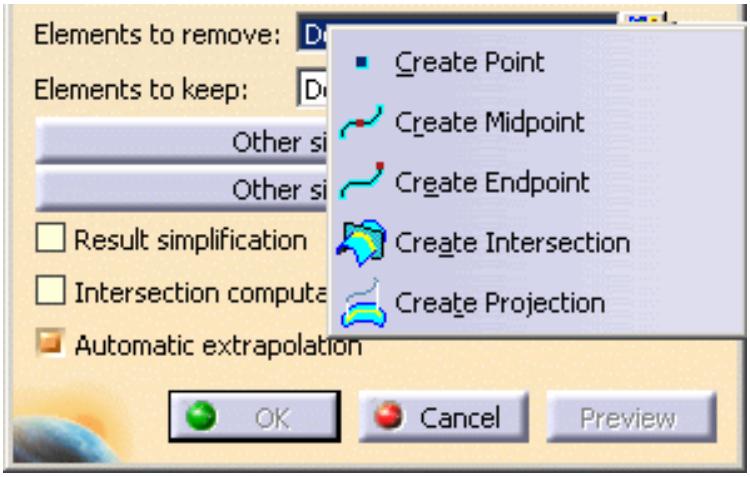
Only the selected portion is removed.
All other elements are kept.



Only the selected portions is kept.
All other elements are removed.

- You can also select a point to define the portion to keep or to remove.

  A contextual menu is available on the **Elements to remove** and **Elements to keep** fields.

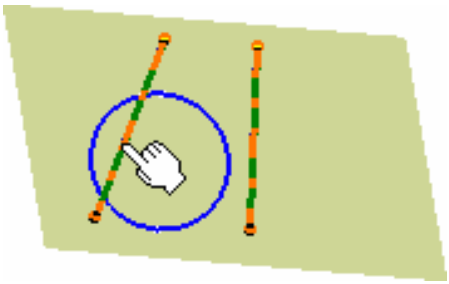


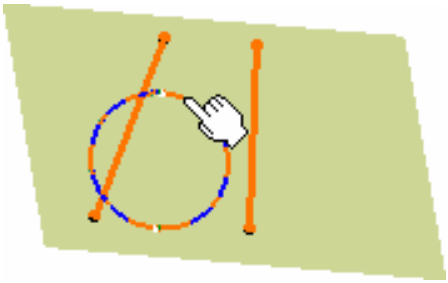
- You do not need to select elements to keep if you already selected elements to remove and vice-versa.

- When trimming wires (curve, line, sketch and so forth) by another wire, you can select a support to define the area that will be kept after trimming the element. It is defined by the vectorial product of the normal to the support and the tangent to the trimming element.

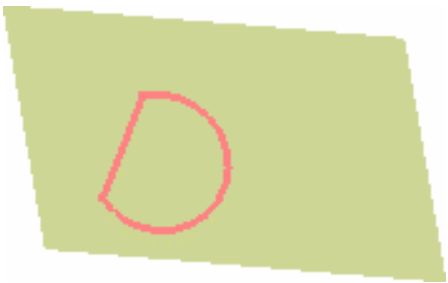
This is especially recommended when trimming a closed wire.

In our example, the Sketch composed of two lines (Sketch.11) is trimmed by the circle (Sketch.10).







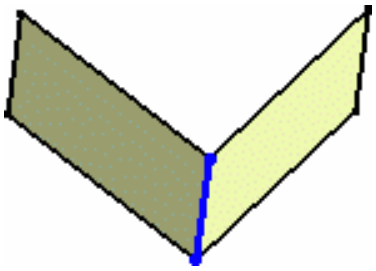


Resulting trimmed element without support selection

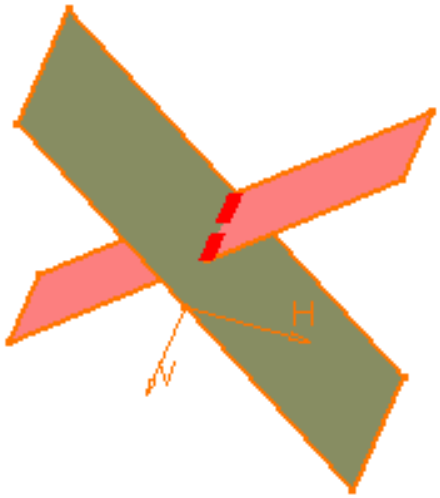
Resulting trimmed element with support selection

- Check the **Result simplification** button to allow the system to automatically reduce the number of faces in the resulting trim whenever possible.

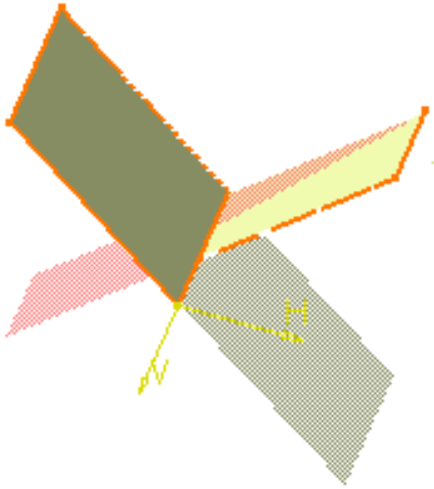
- Check the **Intersection computation** button to create a completely independent element when performing the trimming operation. In that case it appears as a separate Intersect.xxx element in the specification tree.



- Uncheck the **Automatic extrapolation** button if you do not want the automatic extrapolation of the elements to trim. If the **Automatic extrapolation** button is unchecked, an error message is issued when the elements to trim need to be extrapolated, and the latter are highlighted in red in the 3D geometry.



To be able to trim the two surfaces or wireframe elements, check the **Automatic extrapolation** button.



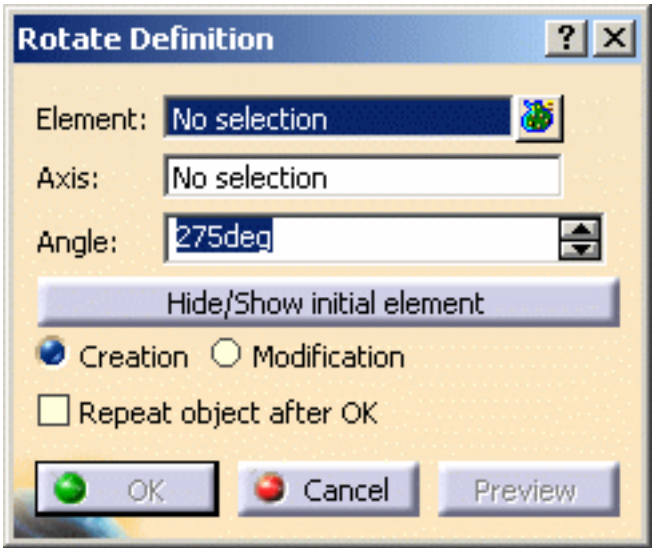
Rotating Geometry

This task shows you how to rotate geometry about an axis.

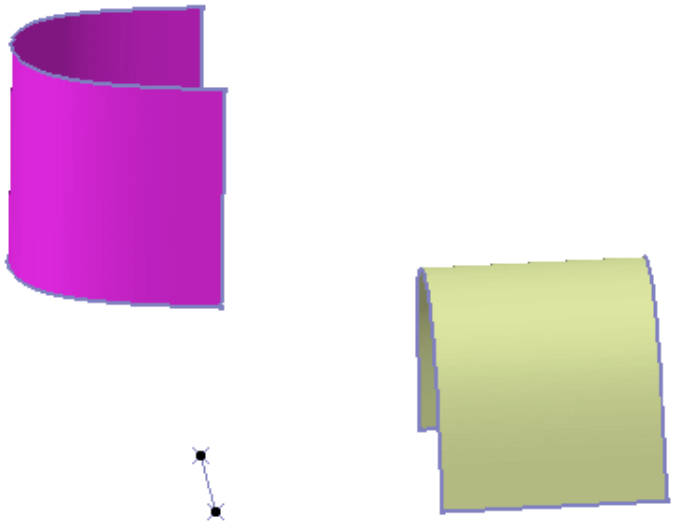
Open the Transform1.CATPart document.

1. Click the **Rotate** icon .

   The Rotate Definition dialog box appears as well as the Multi-Selection dialog box allowing to perform multi-selection.



2. Select the **Element** to be rotated.
3. Select a line as the rotation **Axis**.
4. Enter a value or use the drag manipulator to specify the rotation **Angle**.

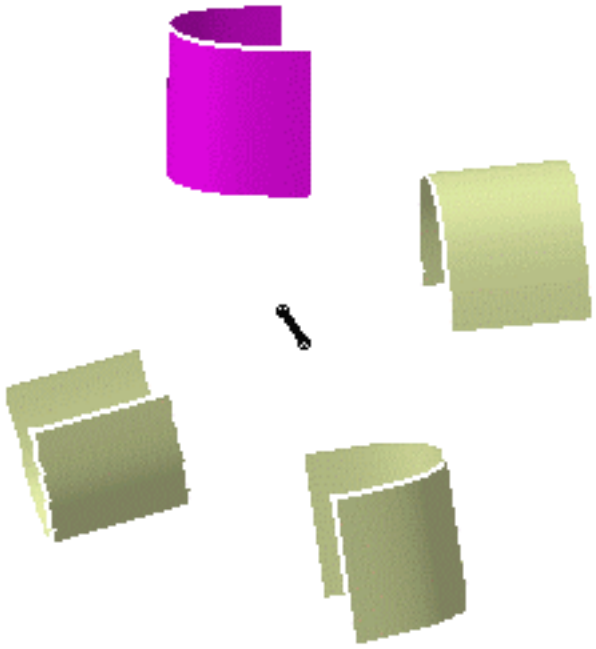


5. Click **OK** to create the rotated element.

The element (identified as Rotate.xxx) is added to the specification tree.

- Use the **Creation** or **Modification** checkboxes to modify the translation behavior:
  - **Creation** is the default behavior when creating a translated element
  - **Modification** enables to modify only the translated element and put the original element in no show.
    Therefore, if this option is checked, the **Hide/Show initial element** button and **Repeat object after OK** checkbox are grayed.

- The original surface is unchanged.
- Use the **Repeat object after OK** checkbox to create several rotated surfaces, each separated from the initial surface by a multiple of the **Angle** value.
  Simply indicate in the Object Repetition dialog box the number of instances that should be created and click OK.



- You can select an axis system as the **Element** to be translated, providing it was previously created.

  The element is identified as Translate.xxx in the specification tree, however the associated icon is the axis system's .

- You can edit the rotated element's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.
- The following capabilities are available: Stacking Commands, Selecting Using Multi-Output, Measure Between and Measure Item.

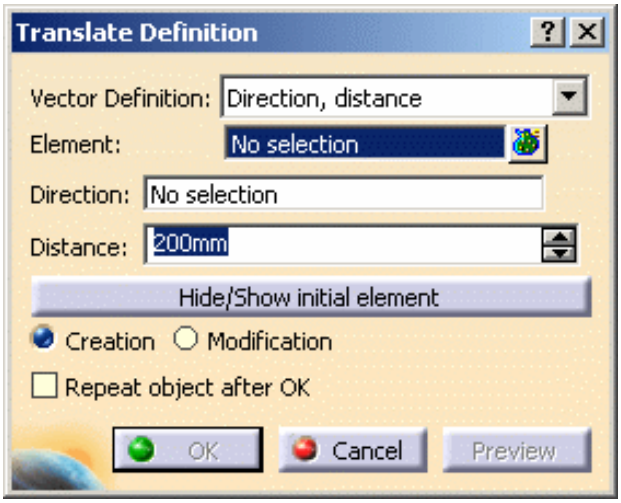
Translating Geometry

This task shows you how to translate one, or more, point, line or surface element.

Open the Translate1.CATPart document.

1. Click the **Translate** icon .

   The Translate Definition dialog box appears as well as the Multi-Selection dialog box allowing to perform multi-selection.



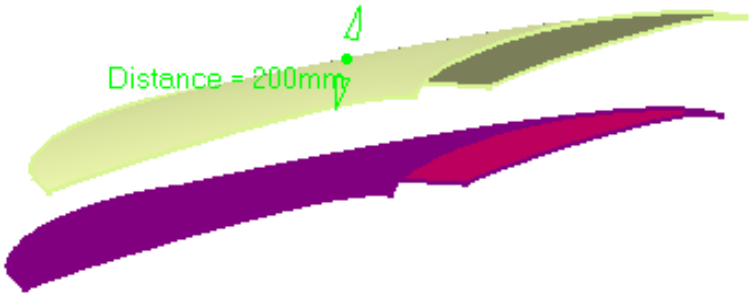
2. Select the **Element** to be translated.

3. Select the **Vector Definition**.

Direction, distance

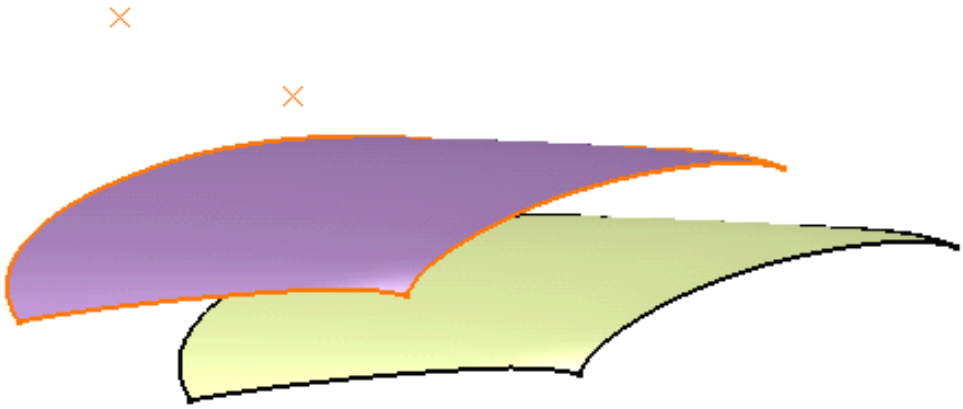
1. Select a line to take its orientation as the translation direction or a plane to take its normal as the translation direction.

   You can also specify the direction by means of X, Y, Z vector components by using the contextual menu on the **Direction** field.



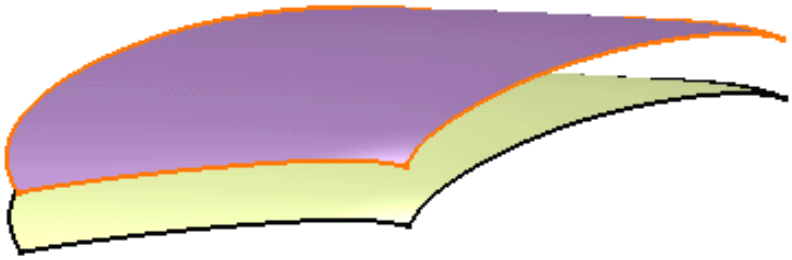
2. Specify the translation **Distance** by entering a value or using the spinners.

Point to Point



1. Select the Start point.
2. Select the End point.

Coordinates



1. Define the X, Y, and Z coordinates. In the example besides, we chose 50mm as X, 0mm as Y, and -100 as Z.

4. Click **OK** to create the translated element.

The element (identified as Translate.xxx) is added to the specification tree.
The original element is unchanged.

- Use the **Hide/Show initial element** button to hide or show the original element for the translation.
- Use the **Repeat object after OK** checkbox to create several translated surfaces, each separated from the initial surface by a multiple of the **Distance** value.
  Simply indicate in the Object Repetition dialog box the number of instances that should be created and click OK.



- Use the **Creation** or **Modification** checkboxes to modify the translation behavior:
  - **Creation** is the default behavior when creating a translated element
  - **Modification** enables to modify only the translated element and put the original element in no show. Therefore, if this option is checked, the **Hide/Show initial element** button and **Repeat object after OK** checkbox are grayed.

- The elements to be translated are kept next time you enter the command and you change the vector definition.
- You can select an axis system as the **Element** to be translated, providing it was previously created. The element is identified as Translate.xxx in the specification tree, however the associated icon is the axis system's .

Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.

The following capabilities are available: Stacking Commands, Selecting Using Multi-Output, Measure Between and Measure Item.

Performing a Symmetry on Geometry

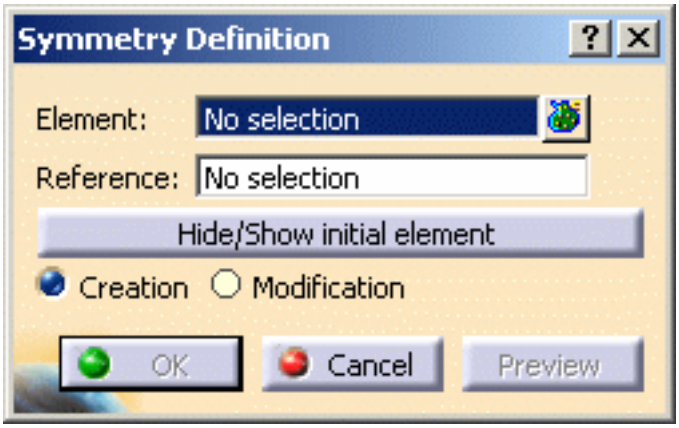
This functionality is P2 for FreeStyle Shaper, Optimizer, and Profiler.

This task shows you how to transform geometry by means of a symmetry operation.

Open the Transform1.CATPart document.

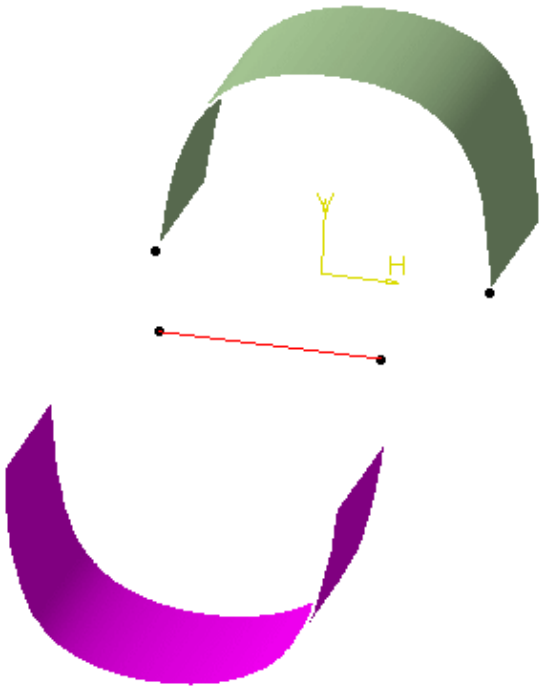
1. Click the **Symmetry** icon .

   The Symmetry Definition dialog box appears as well as the Multi-Selection dialog box allowing to perform multi-selection.

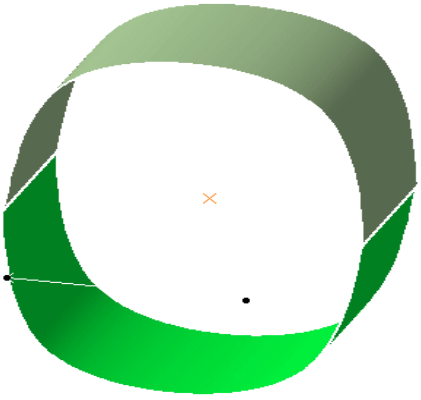


2. Select the **Element** to be transformed by symmetry.
3. Select a point, line or plane as **Reference** element.

The figure below illustrates the resulting symmetry when the line is used as reference element.



The figure below illustrates the resulting symmetry when the point is used as reference element.



4. Click **OK** to create the symmetrical element.

The element (identified as Symmetry.xxx) is added to the specification tree.

- Use the **Creation** or **Modification** checkboxes to modify the translation behavior:
  - **Creation** is the default behavior when creating a translated element
  - **Modification** enables to modify only the translated element and put the original element in no show.
    Therefore, if this option is checked, the **Hide/Show initial element** button and **Repeat object after OK** checkbox are grayed.

- The original element is unchanged.

- The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.



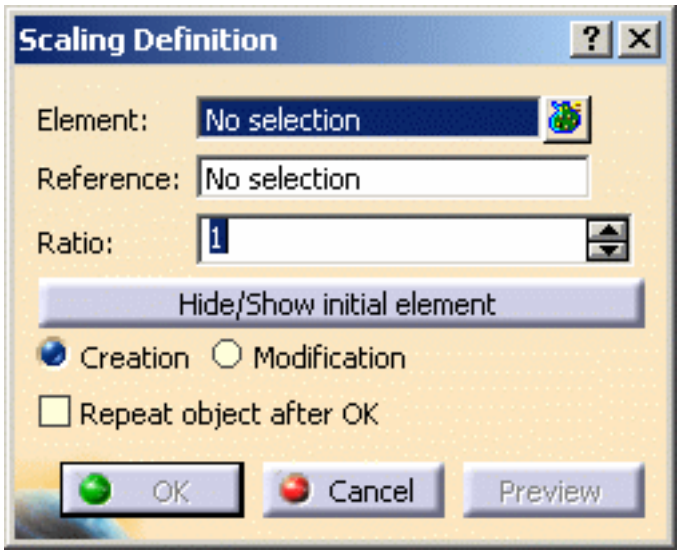
Transforming Geometry by Scaling

This task shows you how to transform geometry by means of a scaling operation.

Open the Transform1.CATPart document.

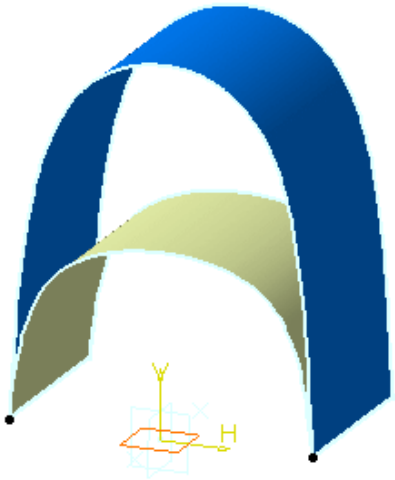
1. Click the **Scaling** icon .

   The Scaling Definition dialog box appears as well as the Multi-Selection dialog box allowing to perform multi-selection.

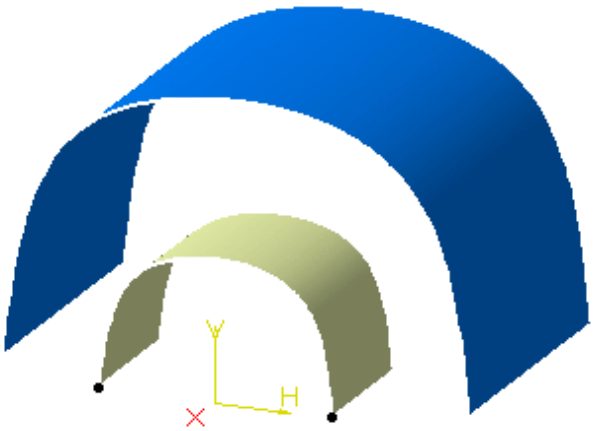


2. Select the **Element** to be transformed by scaling.
3. Select the scaling **Reference** point, plane or planar surface.
4. Specify the scaling **Ratio** by entering a value or using the drag manipulator.

The figure below illustrates the resulting scaled element when the plane is used as reference element (ratio = 2).



The figure below illustrates the resulting scaled element when the point is used as reference element (ratio = 2).



5. Click OK to create the scaled element.

The element (identified as Scaling.xxx) is added to the specification tree.

- Use the **Creation** or **Modification** checkboxes to modify the translation behavior:
  - **Creation** is the default behavior when creating a translated element
  - **Modification** enables to modify only the translated element and put the original element in no show.
    Therefore, if this option is checked, the **Hide/Show initial element** button and **Repeat object after OK** checkbox are grayed.

- The original element is unchanged.

- Use the **Repeat object after OK** checkbox to create several scaled surfaces, each separated from the initial surface by a multiple of the initial **Ratio** value.
  Simply indicate in the Object Repetition dialog box the number of instances that should be created and click OK.



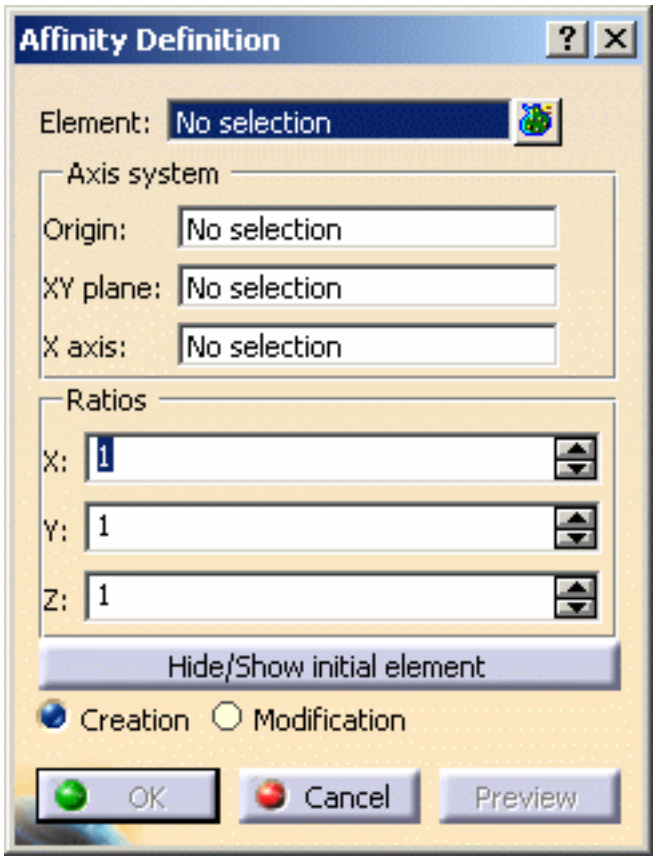
- The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.

Transforming Geometry by Affinity

This task shows you how to transform geometry by means of an affinity operation.

Open the Transform1.CATPart document.

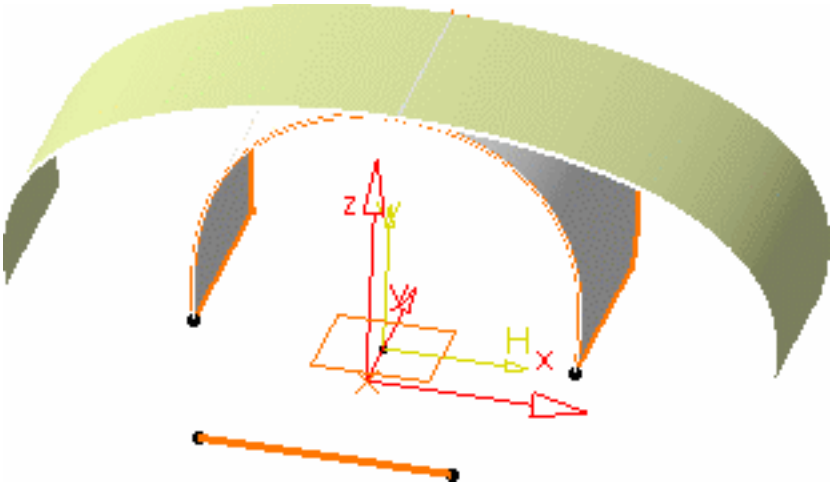
1. Click the **Affinity** icon .



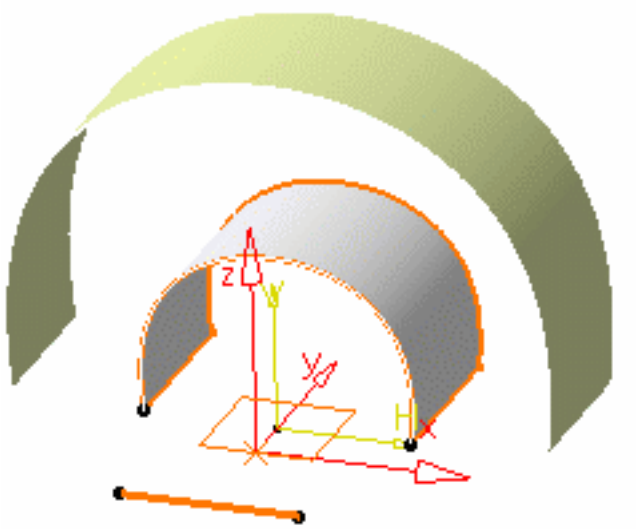
The Affinity Definition dialog box appears as well as the Multi-Selection dialog box allowing to perform multi-selection.

2. Select the **Element** to be transformed by affinity.
3. Specify the characteristics of the **Axis system** to be used for the affinity operation:*
   - the **Origin** (Point.1 in the figures below)
   - the **XY plane** (the XY plane in the figures below)
   - the **X axis** (Line.1 in the figures below).
4. Specify the affinity **Ratios** by entering the desired **X**, **Y**, **Z** values.

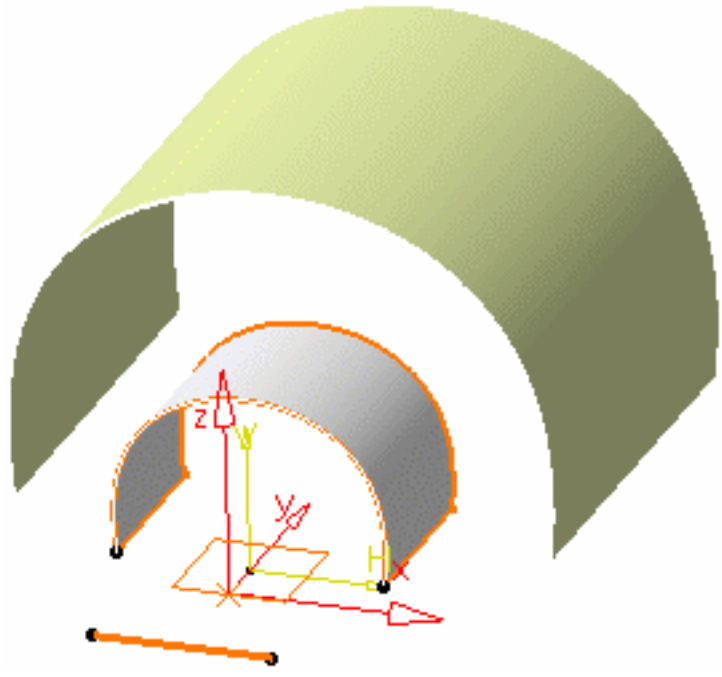
The figure below illustrates the resulting affinity with ratios X = 2, Y =1 and Z=1.



The figure below illustrates the resulting affinity with ratios X = 2, Y =1 and Z=2.



The figure below illustrates the resulting affinity with ratios X = 2, Y =2.5 and Z=2



5. Click **OK** to create the affinity element.

The element (identified as Affinity.xxx) is added to the specification tree.

- Use the **Creation** or **Modification** checkboxes to modify the translation behavior:
  - **Creation** is the default behavior when creating a translated element
  - **Modification** enables to modify only the translated element and put the original element in no show.
    Therefore, if this option is checked, the **Hide/Show initial element** button and **Repeat object after OK** checkbox are grayed.

- The original element is unchanged.
- The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.



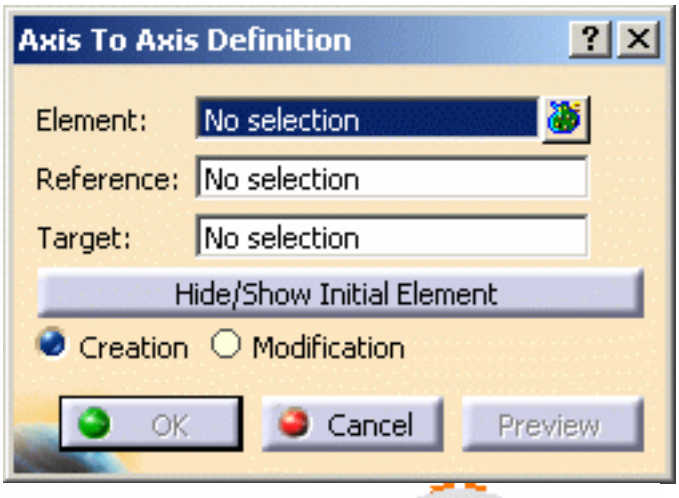
Transforming Elements From an Axis to Another

This task shows you how to transform geometry positioned according to a given axis system into a new axis system. The geometry is duplicated and positioned according to the new axis system. One or more elements can be transformed at a time, using the standard multi-selection capabilities.
See also Defining an Axis System.

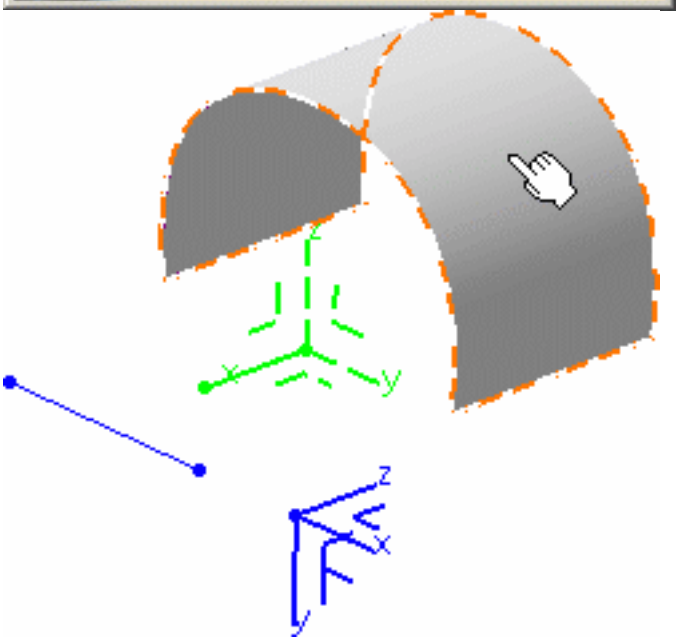
Open the Transform2.CATPart document.

1. Click the **Axis To Axis** icon .

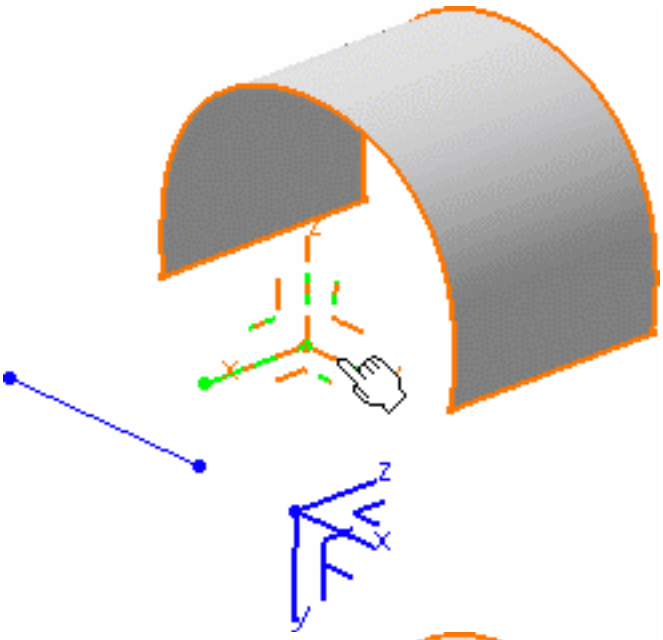
   The Axis to Axis Definition dialog box appears as well as the Multi-Selection dialog box allowing to perform multi-selection.



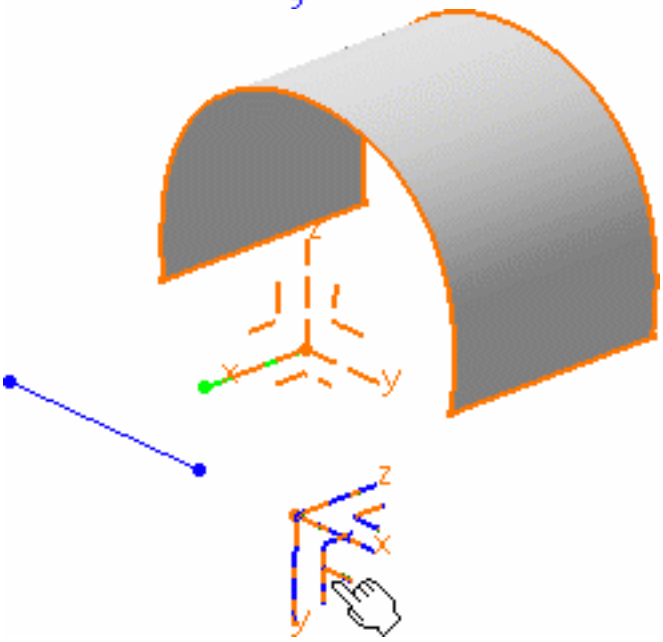
2. Select the **Element** to be transformed into a new axis system.



3. Select the initial (**Reference**) axis system, that is the current one.

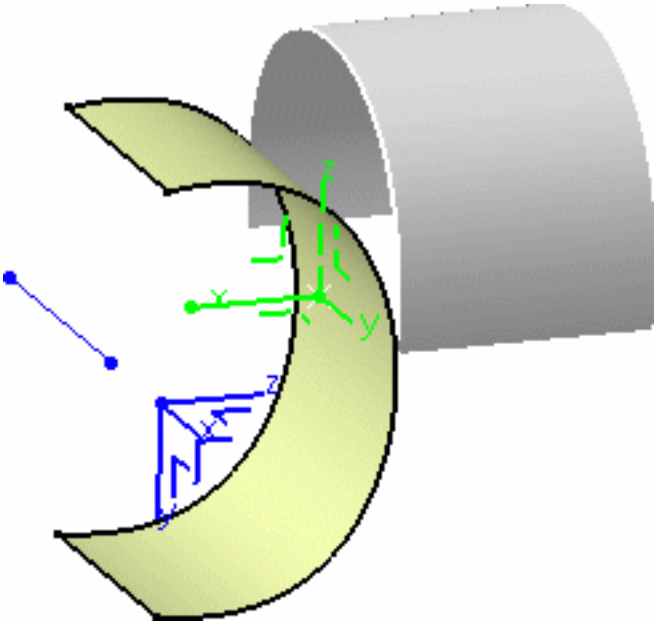


4. Select the **Target** axis system, that is the one into the element should be positioned.



5. Click **OK** to create the transformed element.
   New geometry is now positioned into the new axis system.

The element (identified as Axis to axis transformation.xxx) is added to the specification tree.



- Use the **Creation** or **Modification** checkboxes to modify the translation behavior:
  - **Creation** is the default behavior when creating a translated element
  - **Modification** enables to modify only the translated element and put the original element in no show.
    Therefore, if this option is checked, the **Hide/Show initial element** button and **Repeat object after OK** checkbox are grayed.

- You can choose to display only the elements resulting from the transformation using the **Hide/Show Initial Element** button.
- Use the contextual menus on the dialog box fields to create elements as needed.
- A contextual menu is also available to manage the list of selected elements.

The following capabilities are also available: Stacking Commands and Selecting Using Multi-Output.

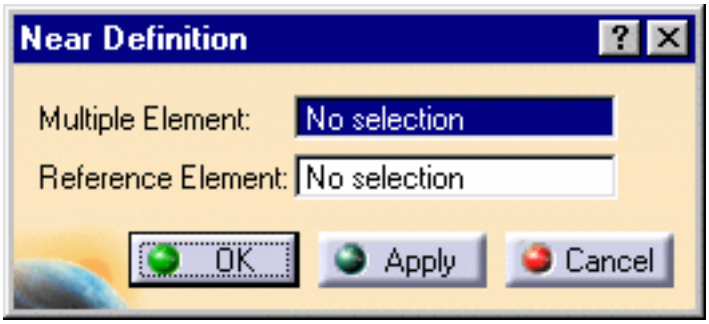
Creating the Nearest Entity of a Multiple Element

This task shows you how to create the nearest entity of an element that is made up from several sub-elements.

Open the Near1.CATPart document.

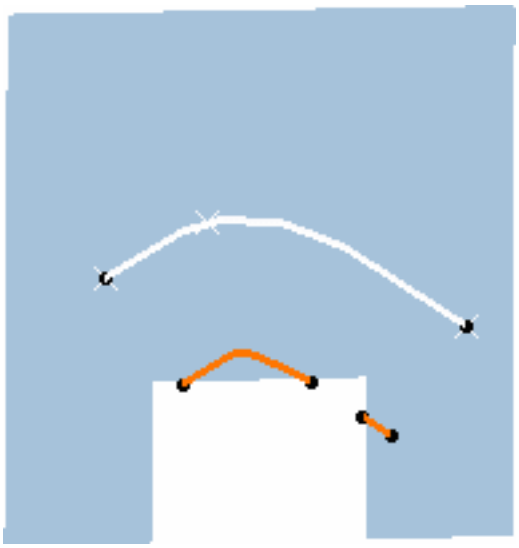
1. Select the **Insert -> Operations -> Near** command.

   The Near Definition dialog box appears.

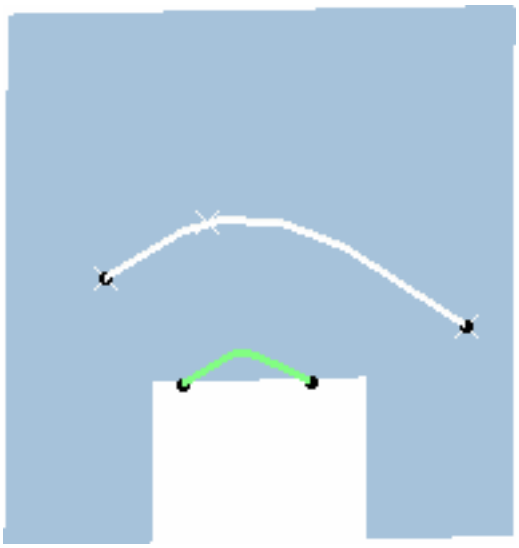


2. Select the element that is made up from several sub-elements.
3. Select a reference element whose position is close to the sub-element that you want to create.

This example shows a parallel curve comprising three sub-elements.



This example shows the sub-element that is nearest to the reference point.



4. Click OK to create the element.

This element (identified as Near.xxx) is added to the specification tree.

The Near Definition dialog box is automatically displayed, when a non-connex element is detected at creation time so that you can directly choose which element should be created.



Back Up Next

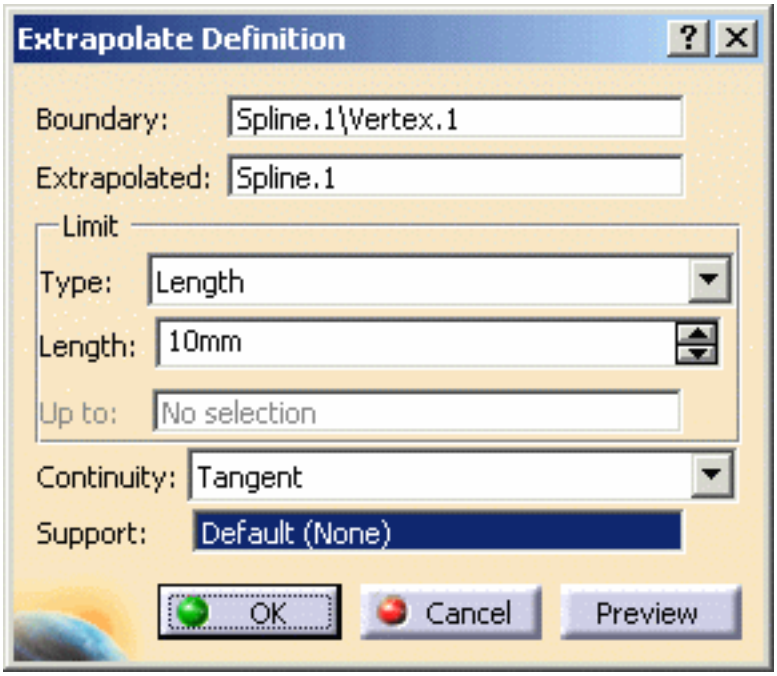
Extrapolating Curves

This task shows you how to extrapolate a curve.

Open the Extrapolate2.CATPart document.

1. Click the **Extrapolate** icon .

   The Extrapolate Definition dialog box appears.



2. Select an endpoint on a curve.

3. Select the curve to be **Extrapolated**:

- entering the value of the extrapolation **Length**.
  In **Curvature mode**, this length actually is the distance on the tangent extrapolation at which a plane normal to the curve is located. This plane is used to split the extrapolated curve.
- selecting a limit surface or plane.

4. Specify **Continuity** conditions:

- **Tangent**: the extrapolation side is tangent to the curve at the selected endpoint.
- **Curvature**: the extrapolation side complies with the curvature of the selected curve.

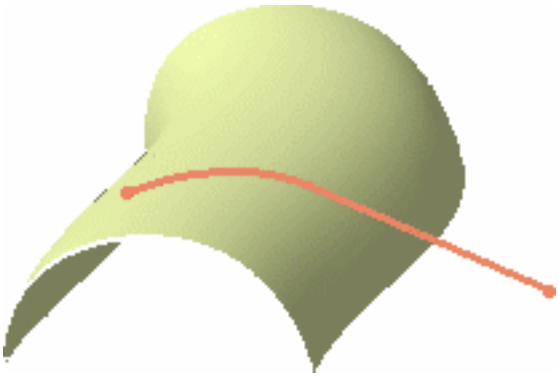


Extrapolation in Curvature mode

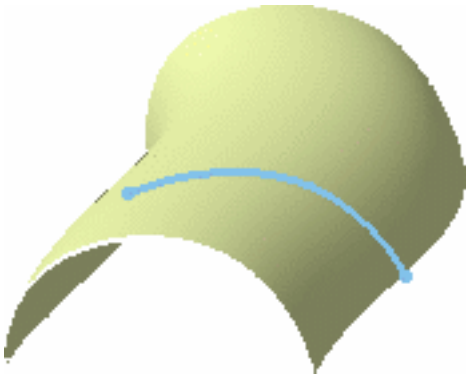


Extrapolation in Tangent mode

If needed, and provided you are working in with a tangency continuity, and the initial curve lies on a plane or surface, you can select this support. In this case the extrapolated curve lies on the surface too, and is relimited by the support boundary.



Extrapolation without support



Extrapolation with a support

5. Click OK to create the extrapolated curve.

The Curve (identified as Extrapol.xxx) is added to the specification tree.

Extrapolating Surfaces

This task shows you how to extrapolate a surface boundary.

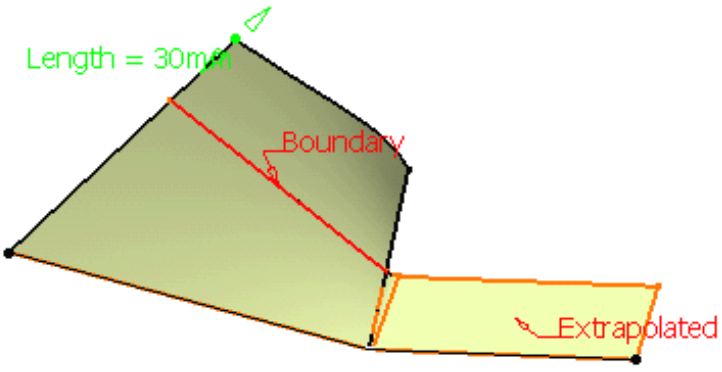
Open the Extrapolate1.CATPart document.

1. Click the **Extrapolate** icon .

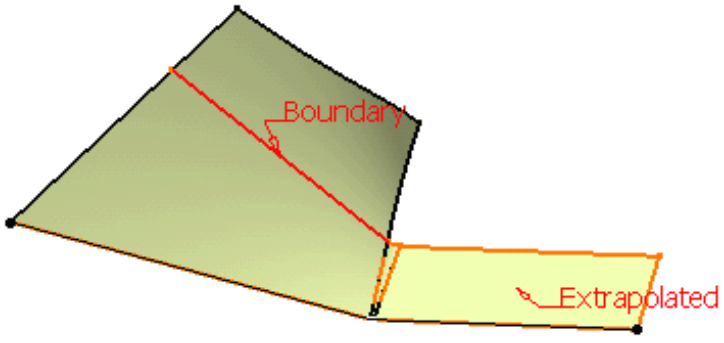
   The Extrapolate Definition dialog box appears.



2. Select a surface **Boundary**.
3. Select the surface to be **Extrapolated**.
4. Specify the **Limit** of the extrapolation by either:
   - entering the value of the extrapolation length
   - selecting a limit surface or plane
   - using the manipulators in the geometry.
5. Specify the **Continuity** type:
   - Tangent
   - Curvature



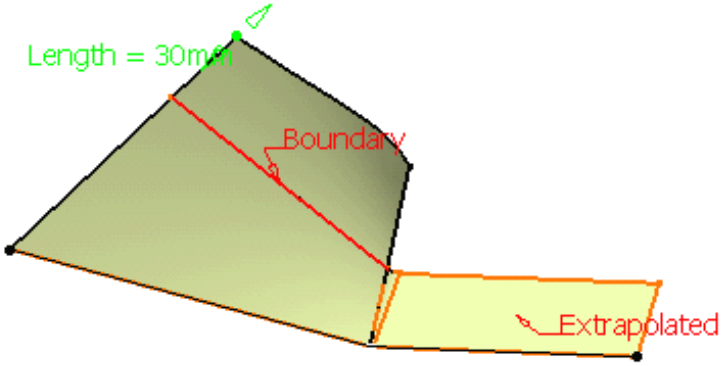
Tangent



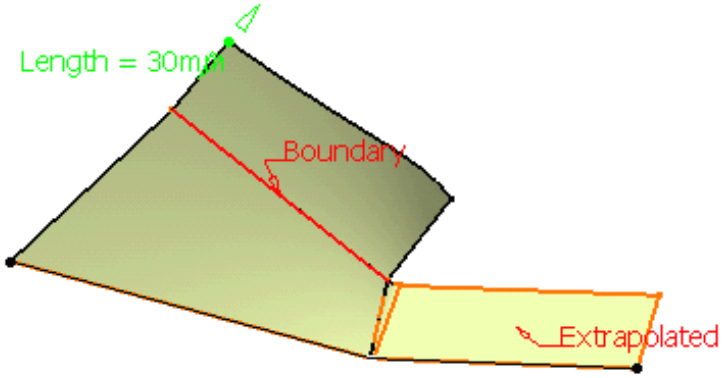
Curvature

6. Specify **Extremities** conditions between the extrapolated surface and the support surface.

   - **Tangent**: the extrapolation sides are tangent to the edges adjacent to the surface boundary.
   - **Normal**: the extrapolation sides are normal to the original surface boundary.

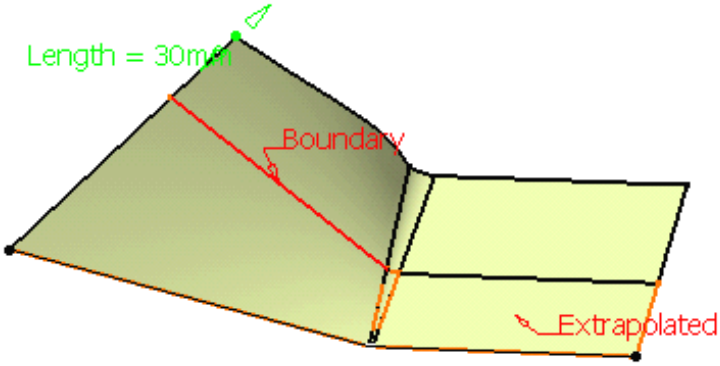


Tangent



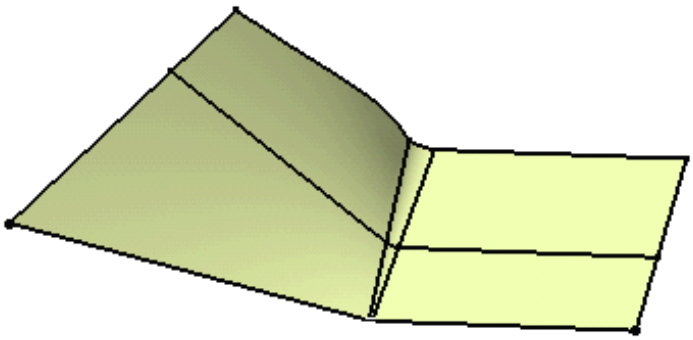
Normal

7. Select the **Tangency** propagation type to propagate the extrapolation to the boundary's adjacent edges.



8. Select the **Assemble result** check box if you want the extrapolated surface to be assembled to the support surface.

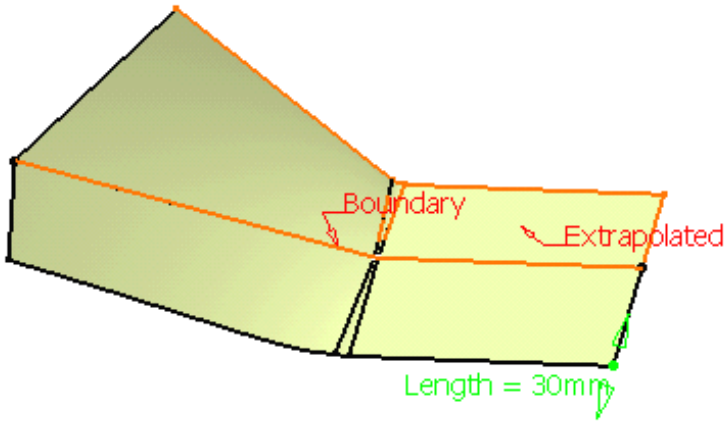
9. Click **OK** to create the extrapolated surface.



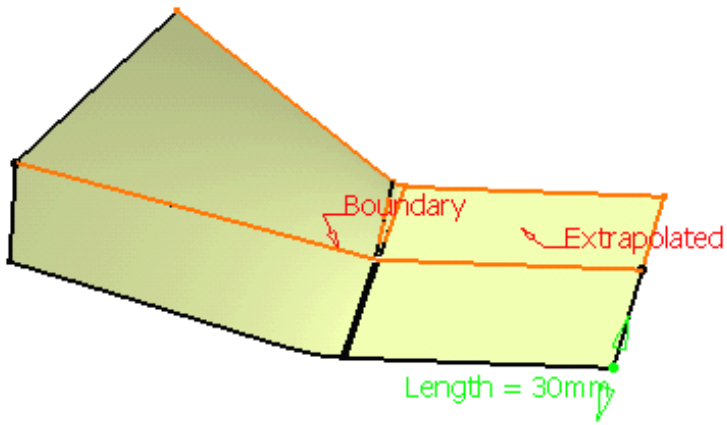
The surface (identified as Extrapol.xxx) is added to the specification tree.

The **Internal Edges** option enables to determine a privileged direction for the extrapolation. You can select one or more edges (in the following example we selected the edge of Surface.1) that will be extrapolated in tangency. You can also select a vertex once you have selected an edge in order to give an orientation to the extrapolation.

You can only select edges in contact with the boundary.



No edges selected



One selected edge

The Internal Edge option is only available with the Generative Shape Design product but not with the Wireframe and Surface product.

You can extrapolate several elements at a time. In this case, refer to Editing a List of Elements to find out how to display and manage the list of selected elements.

The **Up to element** Type, the **Extremities**, **Internal Edges**, and **Assemble result** options are not available with the **Curvature** continuity type.

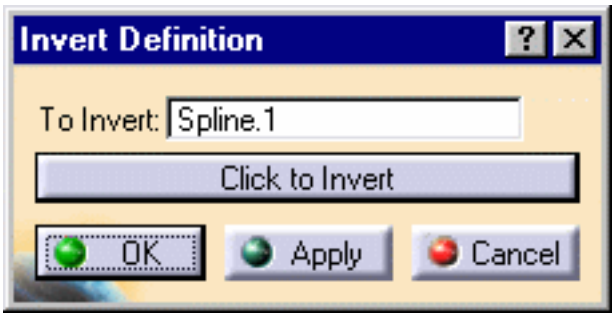
Inverting the Orientation of Geometry

This task shows you how to easily invert the orientation of a surface or curve.

Open any document containing wireframe or surface type element.

1. Select the **Insert -> Operations ->**  **Invert Orientation...** command.

2. Select the surface or curve whose orientation is to be inverted.
An arrow is displayed on the geometry indicating the orientation of the element and the Invert Definition dialog box is displayed.



2. Click the arrow to invert the orientation of the element, or click the **Click to Invert** button.
3. Click **OK** to accept the inverted element.

The element (identified as Inverse.xxx) is added to the specification tree.

Once the orientation is inverted, the **Click to Invert** button changes to **Reset Initial** whether you changed the orientation using the button itself, or the arrow.

Back Up

Using Tools

The Wireframe and Surfaces workbench provides powerful tools to help you manage your surfaces and wireframe geometry.



Display Parents and Children: Select the feature under study, the **Tools -> Parent / Children...** command and use the diverse contextual commands to display parents and children.



Scan the part and define local objects: Select the the **Edit -> Scan or Define in Work Object...** command, click the buttons to move from one local feature to the other, then the **Exit** button.



Update your design: select the element and click the icon or use the contextual menu



Define an axis-system: set the origin and X, Y, and Z directions



Work with a support: click the icon and select a plane or surface as support element.



Snap on a point: snap to the nearest intersection point when working with a support



Create Datums: click the icon to deactivate the History mode



Create constraints: select the element to be constrained, and set the specific options.

Edit geometry: double-click on the element in the tree and modify its parameters

Create elements from an external file: key in space coordinates of elements into an Excel file containing macros, then run the macro.

Select implicit elements: Press and hold the Shift key while clicking the element to which the implicit element belongs.



Copy and paste: select the element(s) to be copied, click the Copy icon, select the target Open Body, then click the Paste icon.

Delete geometry: select the element, choose the **Delete** command, set the deletion options

Manage open bodies: select an open body in the specification tree, use the **Insert -> Open Body** menu command, or **Remove Open Body** or **Change body** contextual menus.

Show/Hide open bodies or their contents: right-click an open body and use the Hide/show contextual command, or use the Hide/Show contextual command on a specific element while another command is running.


Check connections between surfaces: select the surfaces, and set the analysis type and parameters


Check connections between curves: select two curves, specify the type of analysis (distance, tangency, curvature) and set the analysis parameters.


Repeat objects: select an object, choose the **Object Repetition...**menu item and key in the number of object instances

Stack commands: right-click an editable field, choose the contextual menu item allowing the creation of another element.

Edit parameters: Select **xxx1.object -> Edit Parameters** from the contextual menu and modify the parameters.

Select using multi-selection: select one or more elements through the Multi-Selection dialog box and validate you modification to return to the current command

Select Using multi-output: select several elements, click **OK**. The Multi Output feature appears in the specification tree, grouping elements

Analyze using parameterization: select the **Tools -> Parameterization Analysis...** command and define a filter for your query


Apply a material: select an object, click the icon, and select a material.

Back Up

Parents and Children

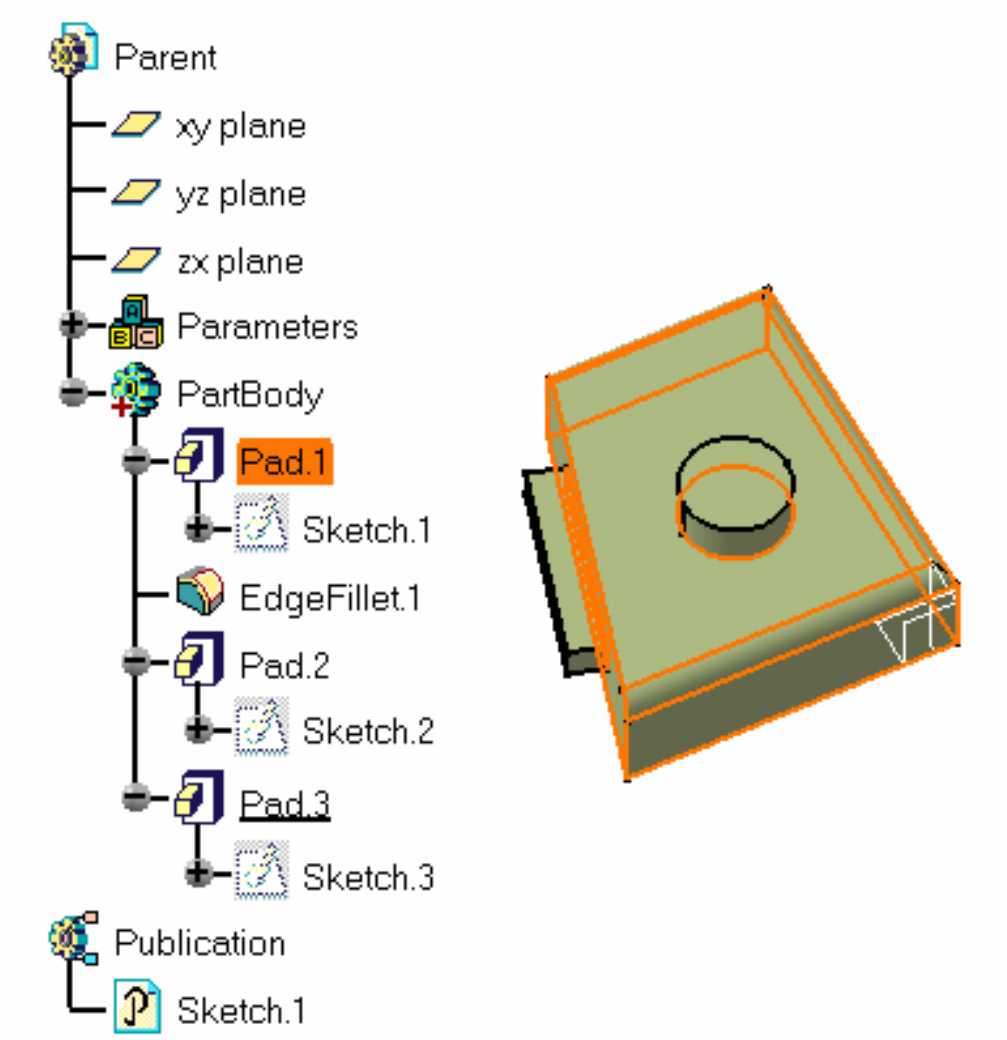
The Parent and Children command enables you to view the genealogical relationships between the different components of a part.

It also shows links to external references and explicitly provides the name of the documents containing these references.

If the specification tree already lets you see the operations you performed and re-specify your design, the graph displayed by the Parent and Children capability proves to be a more accurate analysis tool. We recommend the use of this command before deleting any feature.

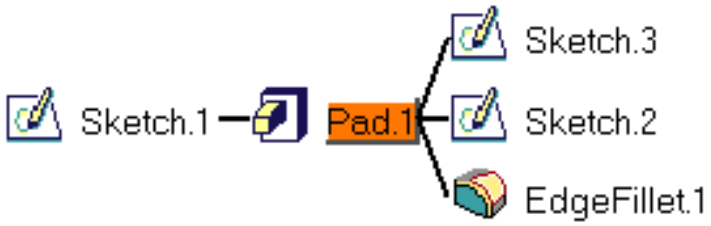
Open the Parent_R9.CATPart document.

1. Select the feature of interest, that is **Pad1**.



2. Select the **Tools -> Parent/Children...** command (or the **Parent/Children...** contextual command).

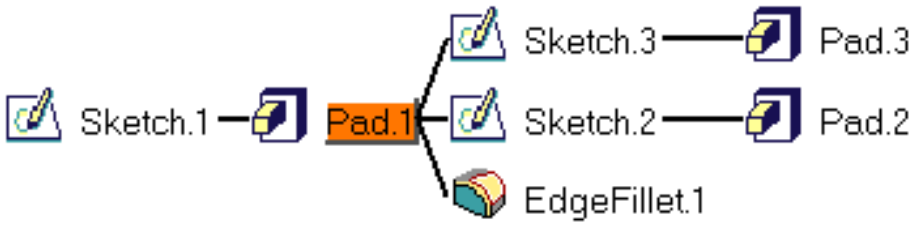
   A window appears containing a graph. This graph shows the relationships between the different elements constituting the pad previously selected.



   If you cannot see the element of interest in the specification tree because you have created a large number of elements, right-click this element in the graph then select the Center Graph contextual command: the element will be more visible in the specification tree.

3. Position the cursor on **Pad 1** and select the **Show All Children** contextual command.

   You can now see that **Sketch 2** and **Sketch 3** have been used to create two additional pads.



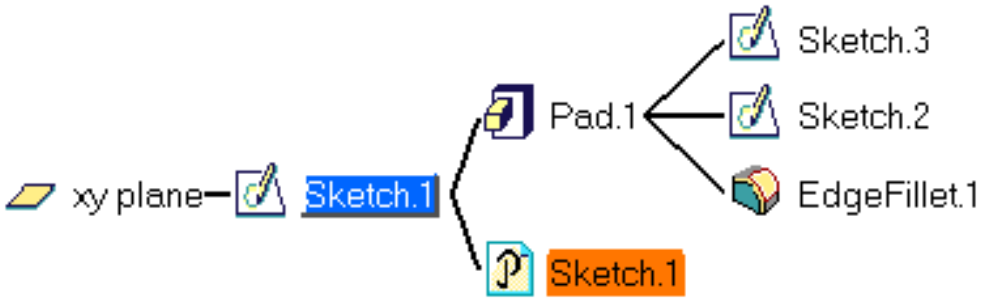
   Here is the exhaustive list of the diverse contextual commands allowing you to hide parents and children. These commands may prove quite useful whenever the view is overcrowded.

   - **Show Parents and Children**
   - **Show Children**
   - **Show All Children**
   - **Hide Children**
   - **Show Parents**
   - **Show All Parents**
   - **Hide Parents**

4. Position the cursor on **Sketch.1** and select the **Show Parents and Children** contextual command.

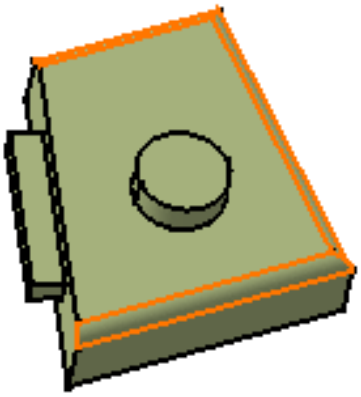
   We can see that **Sketch.1** has been created on **xy plane**.

Moreover, you can see that it is a published element.



5. Now, select **EdgeFillet1** in the graph.

   The application highlights the fillet in the specification tree, in the graph and in the geometry area.



6. Position the cursor on **EdgeFillet1** and select the **Show Parents and Children** contextual command.

   The parent **Pad.1** is displayed.



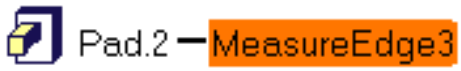
- Double-clicking on the components alternately shows or hides parents and children.

- The **Edit** contextual command can be accessed from any element. For example, right-click EdgeFillet.1 and select Edit. The Edge Fillet dialog box appears. You can then modify the fillet. When done, the Edge Fillet dialog box closes as well as the Parents and Children window close and the fillet is updated.

7. Close the window and select **MeasureEdge3** from the specification tree.

8. Select the **Tools -> Parent/Children...** command.

   The graph that displays shows **Pad.2** as **MeasureEdge3**'s parent.



9. Select the **Show All Parents** contextual command.

   Sketch.2 as **Pad.2**'s parent is now displayed. In turn, **Sketch.2's** own parent **Pad.1** is displayed and so on.



Updating Your Design

This task explains how and when you should update your design.

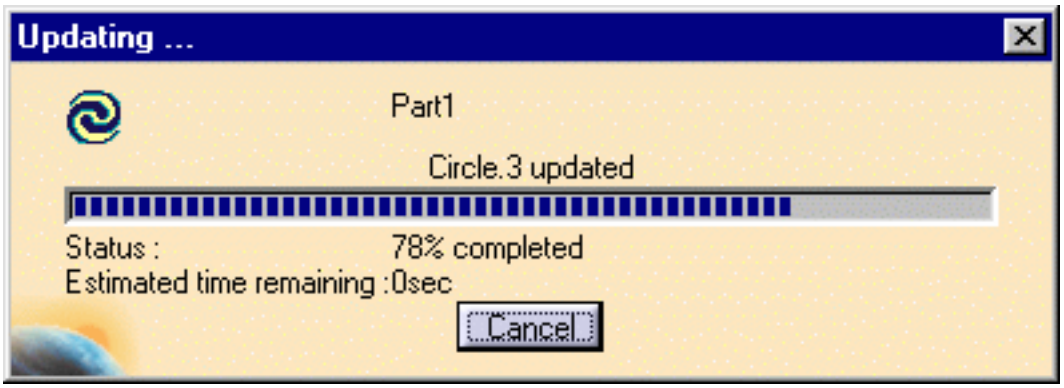
The point of updating your design is to make the application take your last operation into account. Indeed some changes to geometry or a constraint may require rebuilding the part. To warn you that an update is needed, CATIA displays the update symbol next to the part name and displays the corresponding geometry in bright red.

To update a part, the application provides two update modes:

- **automatic update**, available in **Tools -> Options -> Shape, General** tab. If checked, this option lets the application update the part when needed.
- **manual update**, available in **Tools -> Options -> Shape, General** tab, it lets you control the updates of your part. You simply need to click the Update icon whenever you wish to integrate modifications.

1. To update the part, click the **Update** icon .

   A progression bar indicates the evolution of the operation.



You can cancel the undergoing update by clicking the Cancel button available in the Updating... dialog box.

- Keep in mind that some operations such as confirming the creation of features (clicking **OK**) do not require you to use the update command. By default, the application automatically updates the operation.
- The Update capability is also available via **Edit -> Update** and the **Update** contextual command.
- To update the feature of your choice, just select that feature and use the **Local Update** contextual command.
- Besides the update modes, you can also choose to visualize the update on the geometry as it is happening by checking the Activate Local Visualization option from the **Tools -> Options -> Shape, General** tab.
  In this case, as soon as you have clicked the **Update** icon :

  1. the geometry disappears from the screen
  2. each element is displayed as it is updated, including elements in No Show mode. Once they have been updated, they remain in No Show mode.

Interrupting Updates

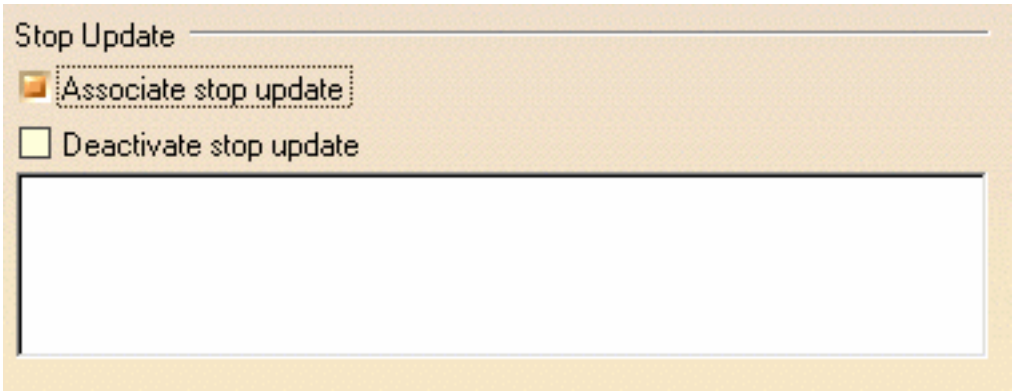
This task explains how to update a part and interrupt the update operation on a given feature by means of a useful message you previously defined.

Open any document containing geometric elements.

1. Right-click an element from the specification tree and choose the Properties contextual menu item.

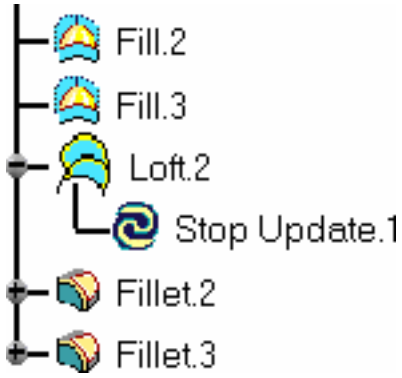
   The Properties dialog box is displayed.

2. From the **Mechanical** tab, check the **Associate stop update** option.



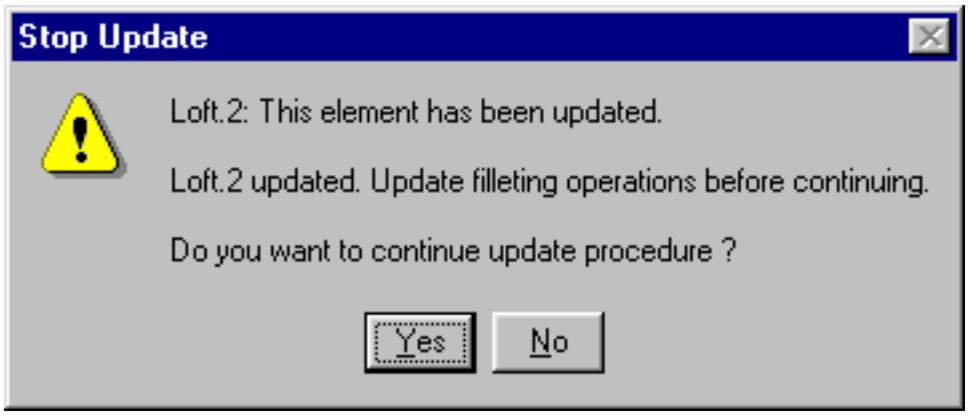
3. Enter the text to be displayed when the updating process will stop when reaching this element.
4. Click OK to confirm and close the dialog box.

The Stop Update.1 feature is displayed in the specification tree, below the element for which it was defined.



5. Whenever it is needed, click the **Update** icon to update the whole part.

The updating process stops after having updated the element selected above, and issues the message as has been defined earlier:



6. Click Yes or No, depending on what you intend to do with the geometry created based on the selected element.

Would you no longer need this capability, you can:

- right-click the element for which the stop was defined, choose the **Properties** contextual command and check the **Deactivate stop update** option from the **Mechanical** tab: the update will no longer at this element.
  You notice that when the capability is deactivated, the Stop Update icon changes to: in the specification tree.
- right-click Stop Update.1 from the specification tree, and choose the **Delete** contextual command.

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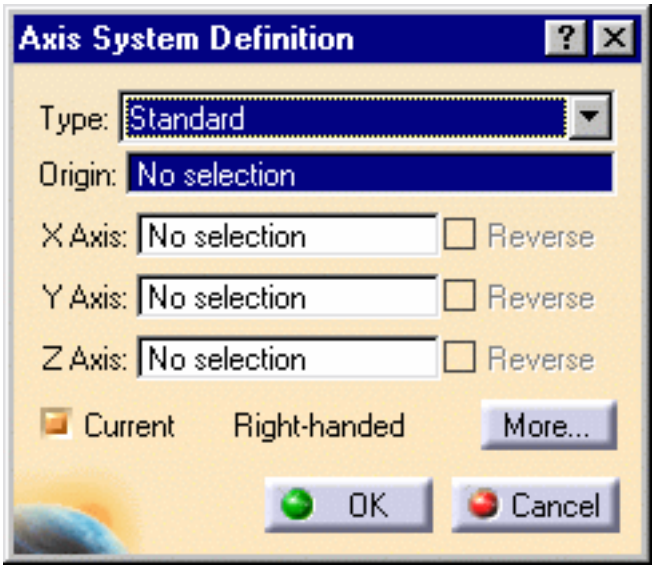
Defining an Axis System

This task explains how to define a new three-axis system locally. There are two ways of defining it: either by selecting geometry or by entering coordinates.

Open the PowerCopyStart1.CATPart document.

1. Select the **Insert -> Axis System** command or click the **Axis System** icon .

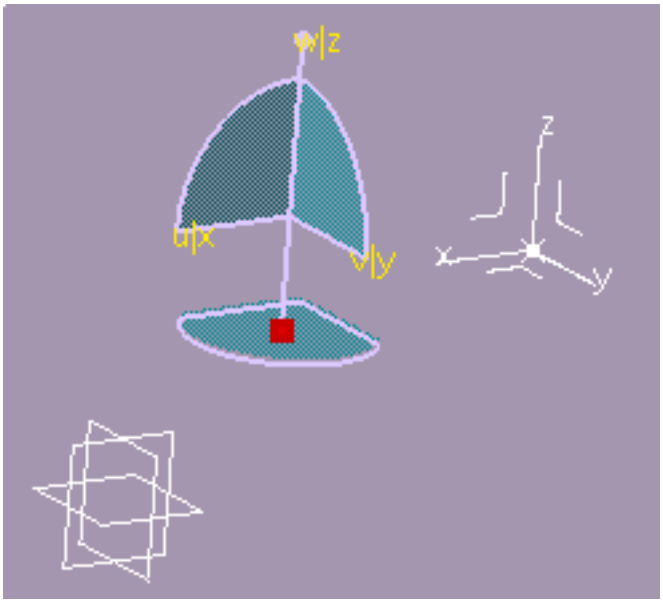
   The Axis System Definition dialog box is displayed.



An axis system is composed of an origin point and three orthogonal axes.
It can be right or left-handed. This information is displayed within the Axis System Definition dialog box.
You can choose from different types of axis system:

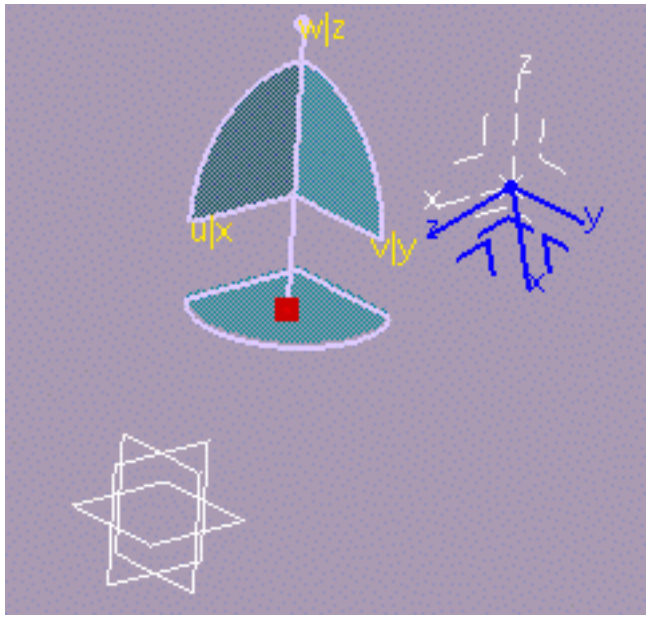
- Standard: defined by a point of origin and three orthogonal directions (by default the current directions of the compass)

Here only the point was selected and nothing specified for the axes.



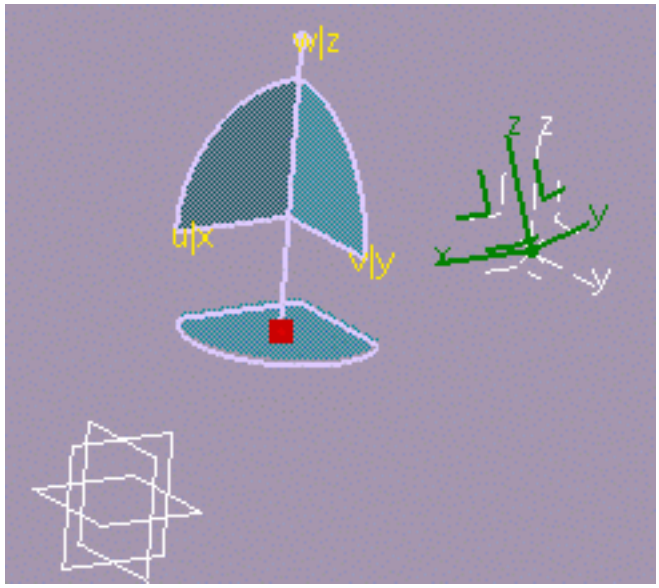
- Axis rotation: defined as a standard axis system and a angle computed from a selected reference

Here the Y axis was set to the standard axis system Y axis, and a 15 degrees angle was set in relation to an edge parallel to the X axis.

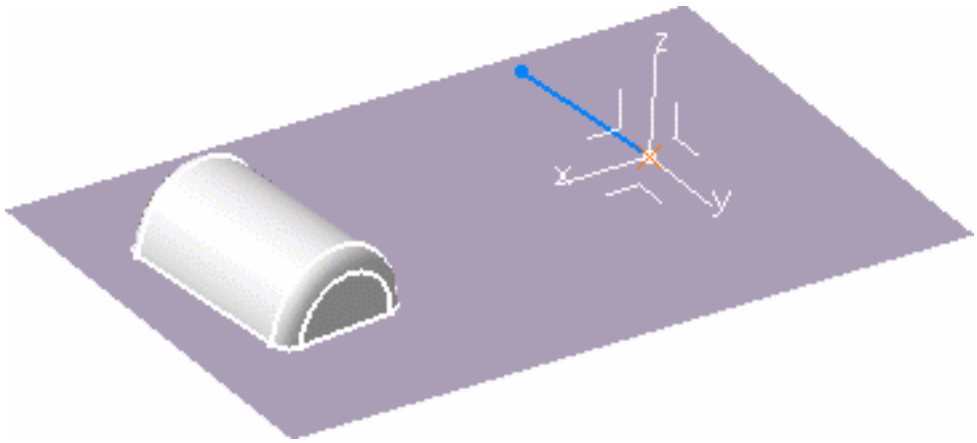


- Euler angles: defined by three angle values computed from the initial X, Y, and Z directions

Here, the **Angle 2** and **Angle 3** were set to 30 degrees.

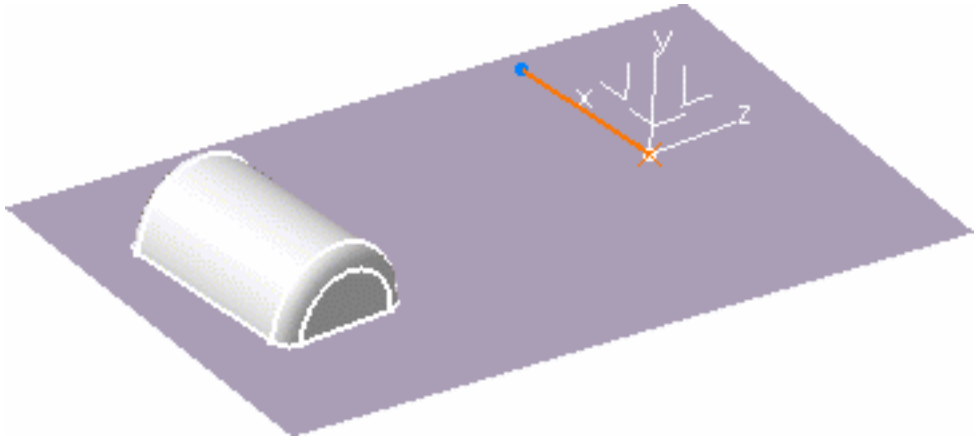


2. Select the point as shown to position the origin of the axis system you wish to create. The application then computes the remaining coordinates. Both computed axes are then parallel to those of the current system. The axis system looks like this:

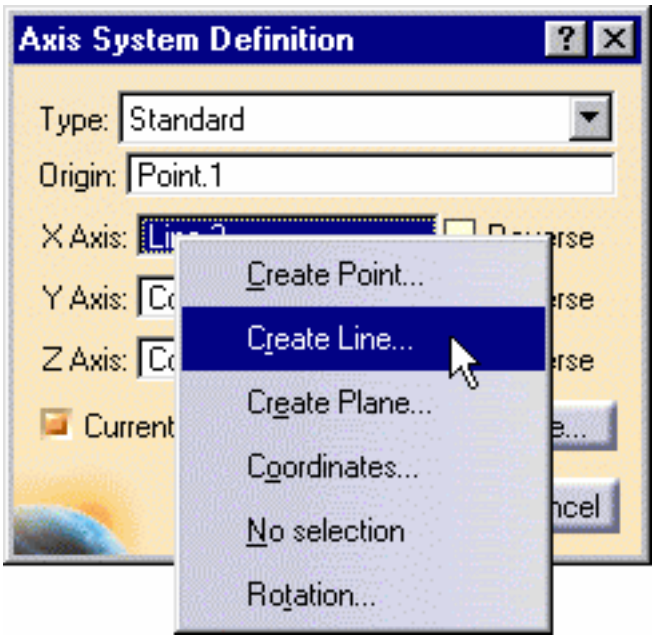


3. If you are not satisfied with x axis, for instance click the X axis field and select a line to define a new direction for x axis.

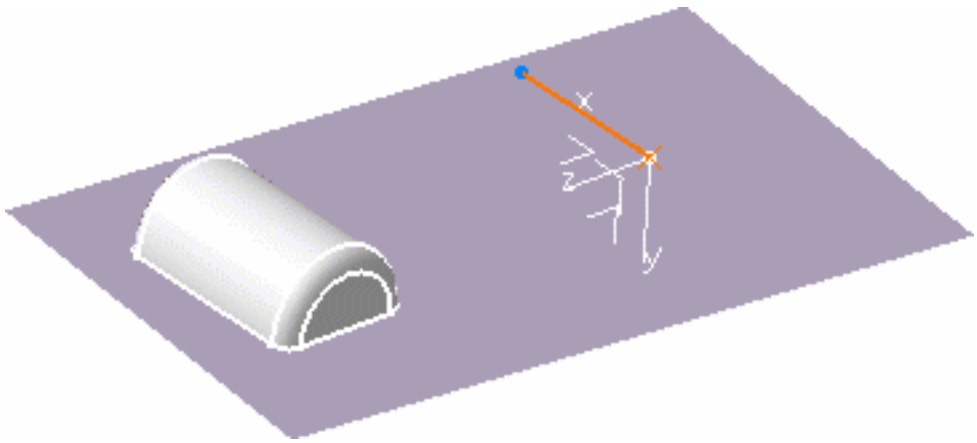
   The x axis becomes colinear with this line.



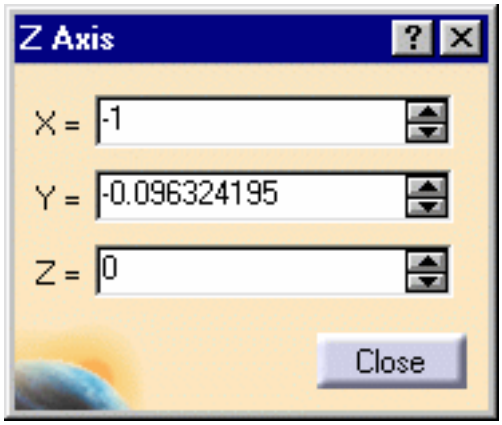
- It can be a line created along the surface edge, for example, using the Create **Line** contextual menu on the selection field, and selecting two surface vertices.
  Similarly you can create Points, and Planes.
- You can also select the **Rotation** contextual menu, and enter an angle value in the X Axis Rotation dialog box.



4. Click the y axis in the geometry to reverse it.
   Checking the **Reverse** button next to the **Y Axis** field reverses its direction too.

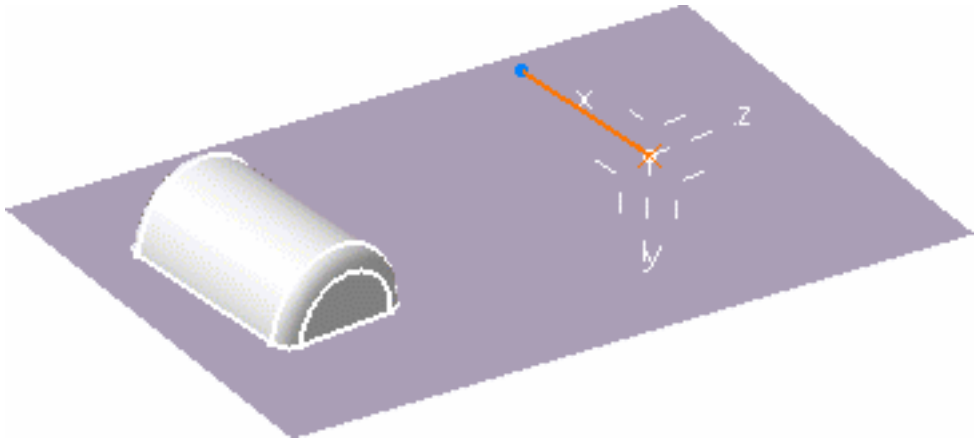


5. You can also define axes through coordinates. Right-click the Z Axis field and select the **Coordinates** contextual command. The Z Axis dialog box appears.



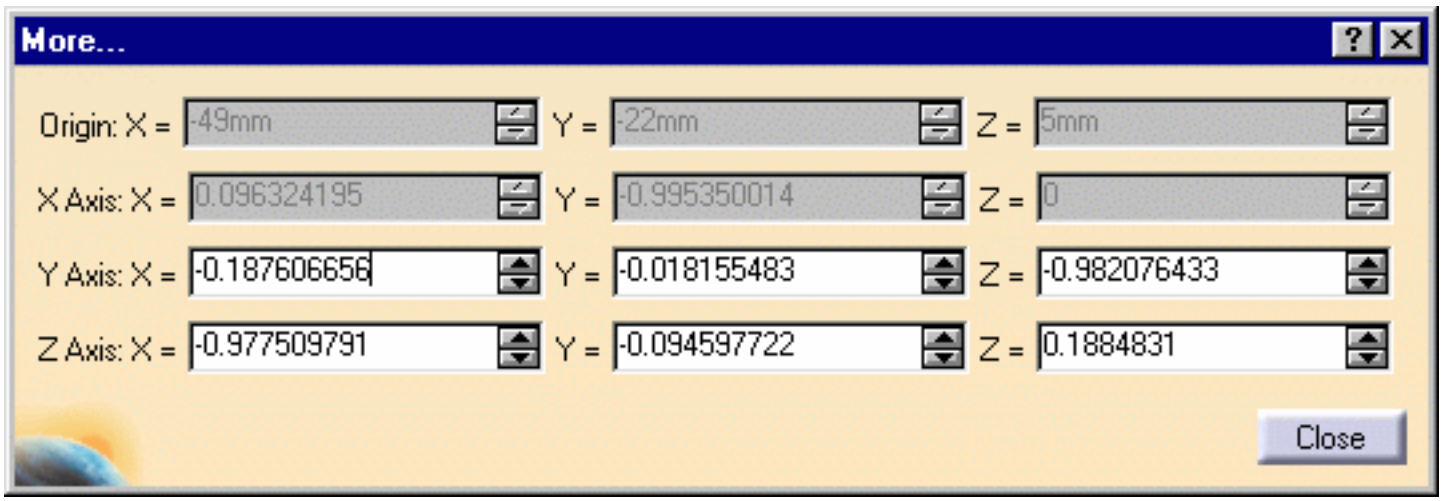
6. Key in X = -1, retain the Y and Z coordinates, and click Close.

   The axis system is modified accordingly, and is now left-handed.



7. Click More to display the More... dialog box.

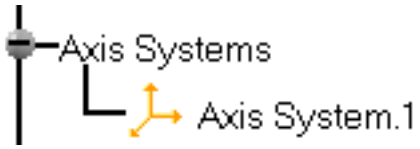
   The first rows contains the coordinates of the origin point. The coordinates of X axis are displayed in the second row. The coordinates of Y and Z axis are displayed in the third and fourth row respectively.



As you are defining your axis system, the application detects if its axes are orthogonal or not. Inconsistencies are revealed via the Update diagnosis dialog box.

8. Uncheck the Current option if you do not want to set your axis as the reference. The absolute axis at the bottom right of the document then becomes the current three axis system.

9. Click OK.



The axis system is created. It is displayed in the specification tree.
When it is set as current, it is highlighted as shown aside.

If you create a point using the coordinates method and an axis system is already defined and set as current, the point's coordinates are defined according to current the axis system. As a consequence, the point's coordinates are not displayed in the specification tree.
Local axes are fixed. If you wish to constrain them, you need to isolate them (using **Isolate** contextual command) before setting constraints otherwise you would obtain over-constrained systems.

The display mode of the axes is different depending on whether the three-axis system is right-handed or left-handed and current or not.

THREE-AXIS SYSTEM	CURRENT	AXIS DISPLAY MODE
right-handed	yes	solid
right-handed	no	dashed
left-handed	yes	dotted
left-handed	no	dot-dashed

Right-click Axis System.1 from the specification tree and select the **Axis System.1 object -> Set as current** contextual command. Axis System.1 is now current. You can then select one of its plane, to define a sketch plane for example.

Editing an Axis System

You can edit your axis system by double-clicking it and entering new values in the dialog box that appears. You can also use the compass to edit your axis system.

Note that editing the geometrical elements selected for defining the axes or the origin point affects the definition of the axis system accordingly.

Right-clicking Axis System.Xobject in the specification tree lets you access the following contextual commands:

- Definition...:redefines the axis system
- Isolate: sets the axis system apart from the geometry
- Set as Current/Set as not Current: defines whether the axis system is the reference

or not.



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Working with a Support

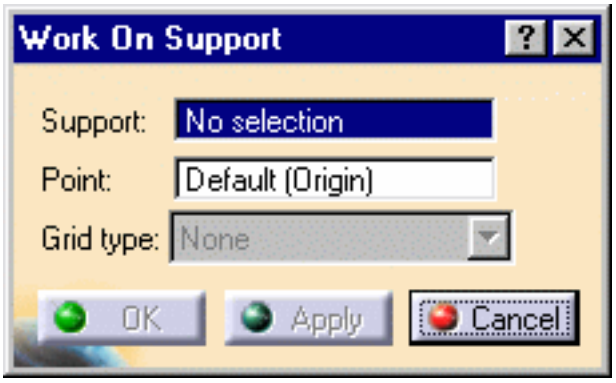
This task shows how to create a support. It may be either a plane or a surface. This will allow you to automatically reference a surface or plane as the supporting element whenever you need one, when creating Lines for example. You will no longer have to explicitly select the support element.
It will also allow you to create reference points on the fly on the support, whenever you need a reference point to create other geometric elements.

Open the WorkOnSupport1.CATPart document.

Creating a support from a surface

1. Click the **Work on Support** icon .

   The Work on Support dialog box appears.

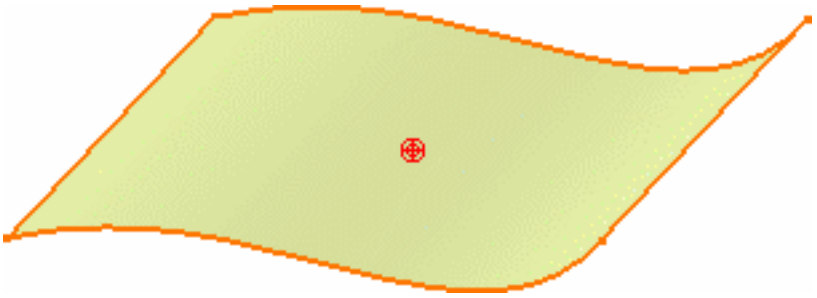


2. Select the surface to be used as support element.

   If a plane is selected, a grid is displayed to facilitate visualization.

3. Select a point.

   By default the surface's midpoint is selected.

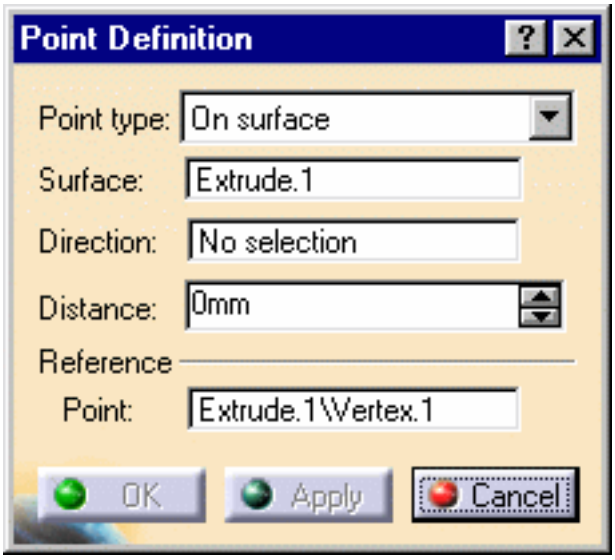


4. Click OK in the dialog box.

The element (identified as WorkingSupport.xxx) is added to the specification tree.

5. Click the **Point** icon .

   The Point definition dialog box is displayed. The Reference Point field is automatically filled in with the point selected when defining the currently active working support.

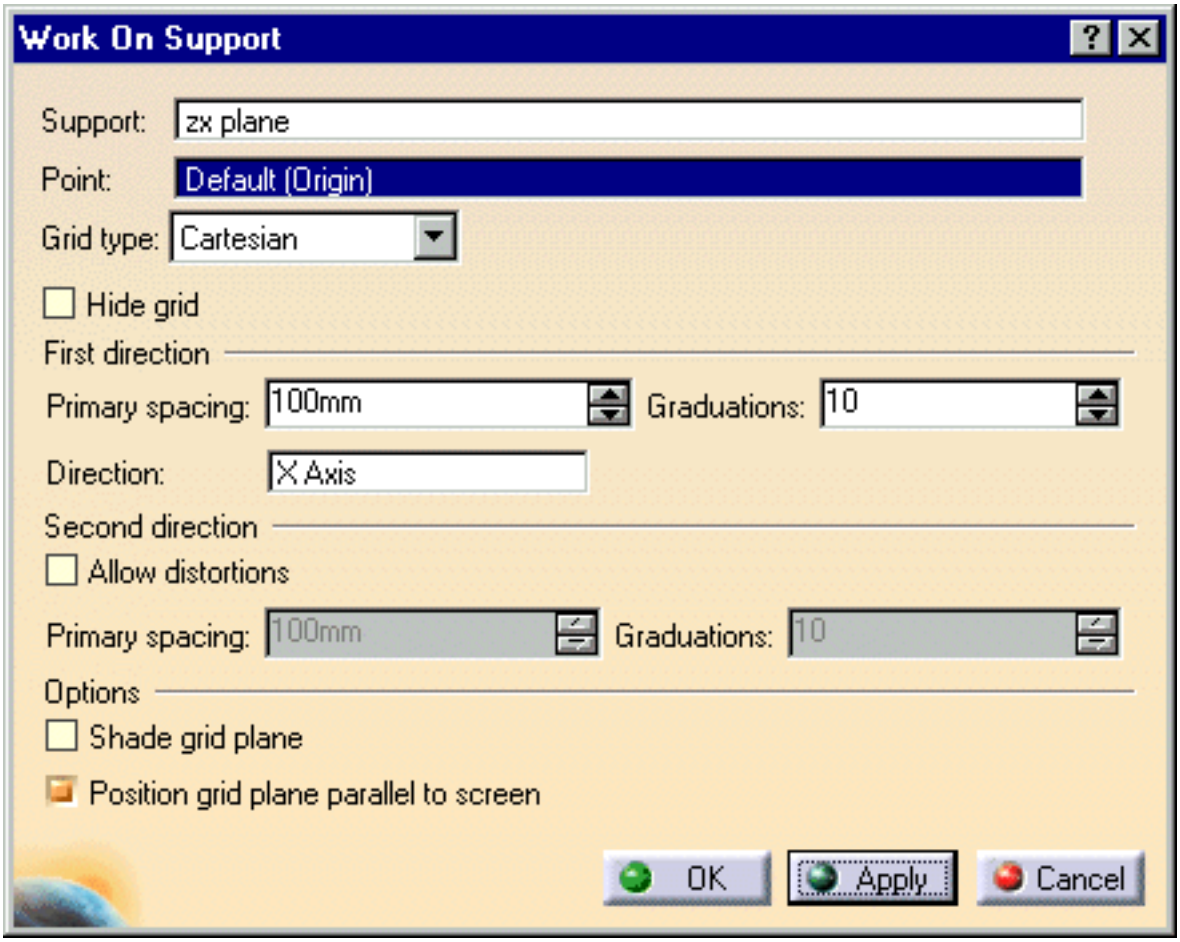


Creating a support from a plane

1. Click the **Work on Support** icon .

2. Select the plane to be used as support element.

   The Work on Support dialog box is displayed, allowing you to define the plane:



By default, the Grid type is set to Cartesian, to define a Cartesian plane.

A grid can also be displayed to facilitate visualization. You can hide it by checking the **Hide grid** option.

3. Select a point, as the support plane's origin.

By default the plane's origin is selected. Beware of the plane representation not being located at the plane's origin. In this case, the default point, really is displayed at the origin and therefore not necessarily onto the plane representation.

4. Define the **First direction** scale (H for horizontal), by setting **Primary spacing** and **Graduations** values.
5. If needed, select a direction to specify the H direction.
   You can right-click in the editable field to display the contextual menu and define the direction (by defining its vector, creating a line, and so forth).
6. If you wish, you can define another scale for the **Second direction** scale (V for vertical), thus allowing distortions of the grid. Check the **Allow distortions** option to activate the **Primary spacing** and **Graduations** fields of the second direction.
7. Check the **Shade grid plane** option to visualize the support plane as a solid geometric element.

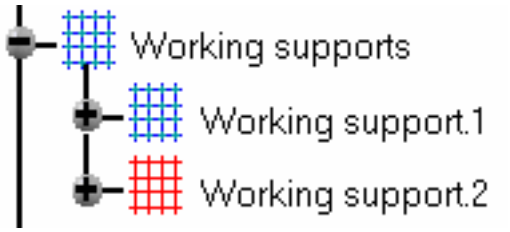
This is possible only if the View mode is adequate.

8. Check the **Position grid plane parallel to screen** to reset the grid visualization parallel to the screen.

9. Click OK in the dialog box.

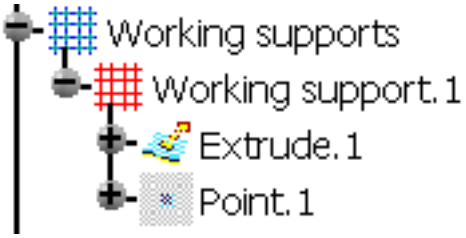
   The element (identified as WorkingSupport.xxx) is added to the specification tree.

By default the last created working support is displayed in red in the specification tree.
Use the **Set As Current**/**Set As Not Current** contextual menu, or the **Working Supports Activity** icon, to define which is the default current support that will be automatically selected when entering a command that requires a working support.



Use the **Get Features on Support** contextual menu on the working support features to retrieve the features created from a single or a multi-selection works on support. As a result, the retrieved features are selected in the current editor and highlighted in the specification tree, therefore allowing you to use them more easily.

- Regardless of the type of working support created (surface or plane) once you choose to work on the support, you can directly click onto the support to create points. This capability is available with commands such as point, line, spline, and polyline, and most commands where you need to select point as inputs.
- Working supports can be edited, updated, or deleted just as any other feature.
- Click the Snap to point icon to snap the point being created onto the nearest intersection point on the grid.
- Features created using a support are now aggregated under the parent command that created them and put in no show in the specification tree.



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Creating Datums

This task shows how to create geometry with the History mode deactivated.
In this case, when you create an element, there are no links to the other entities that were used to create that element.

1. Click the **Create Datum** icon  to deactivate the History mode.

   It will remain deactivated until you click on the icon again.

   If you double-click this icon, the Datum mode is permanent. You only have to click again the icon to deactivate the mode.

   A click on the icon activates the Datum mode for the current or the next command.

The History mode (active or inactive) will remain fixed from one session to another: it is in fact a setting.

Creating Constraints

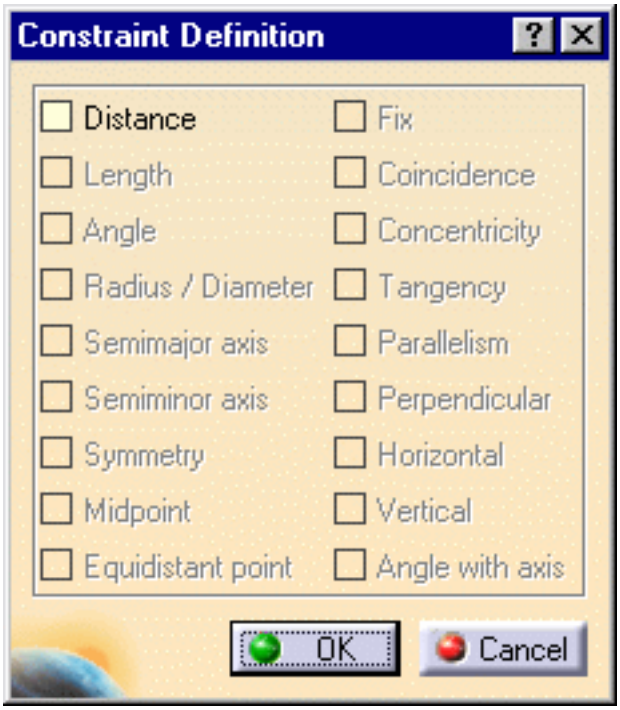
This task shows how to set geometric constraints on geometric elements.

Such a constraint forces a limitation. For example, a geometric constraint might require that two lines be parallel.

To set a constraint between elements:

1. Multi-select the two or three elements to be constrained.
2. Click the **Constraint with dialog box** icon  .

The Constraint Definition dialog box appears indicating the types of constraint you can set between the selected elements.



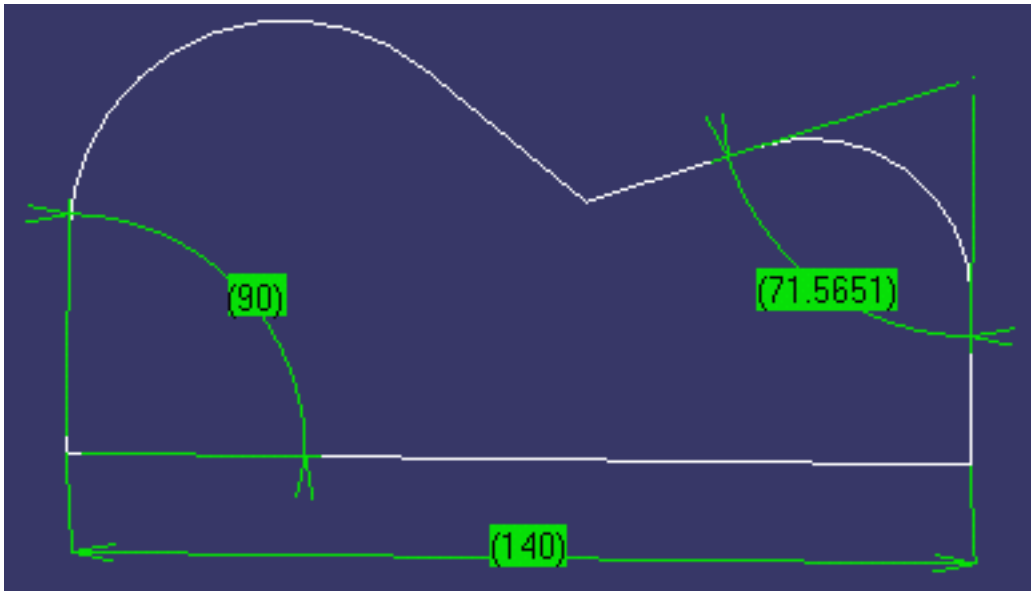
3. Select an available option to specify that the corresponding constraint is to be made.
4. Click OK.

The corresponding constraint symbol appears on the geometry.

To set a geometric constraint on a single element:

1. Select the element to be constrained.
2. Click the **Constraint** icon .

The corresponding constraint symbol appears on the geometry.





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Editing Surface and Wireframe Definitions

This task shows how to edit the definition of an already created geometric element.

1. Activate the Definition dialog box of the element that you want to edit in one of the following ways:

   - Select the element then choose the **xxx.object -> Definition** command from the contextual menu
   - Select the element then choose the **Edit -> xxx.object -> Definition** command
   - Double-click on the element identifier in the specification tree.

2. Modify the definition of the element by selecting new reference elements or by entering new values.
3. Click OK to save the new definition.



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Selecting Implicit Elements

There are many ways of selecting geometrical elements, either in the geometry as described in the *Infrastructure User's Guide*, Selecting Objects section, or in the specification tree.

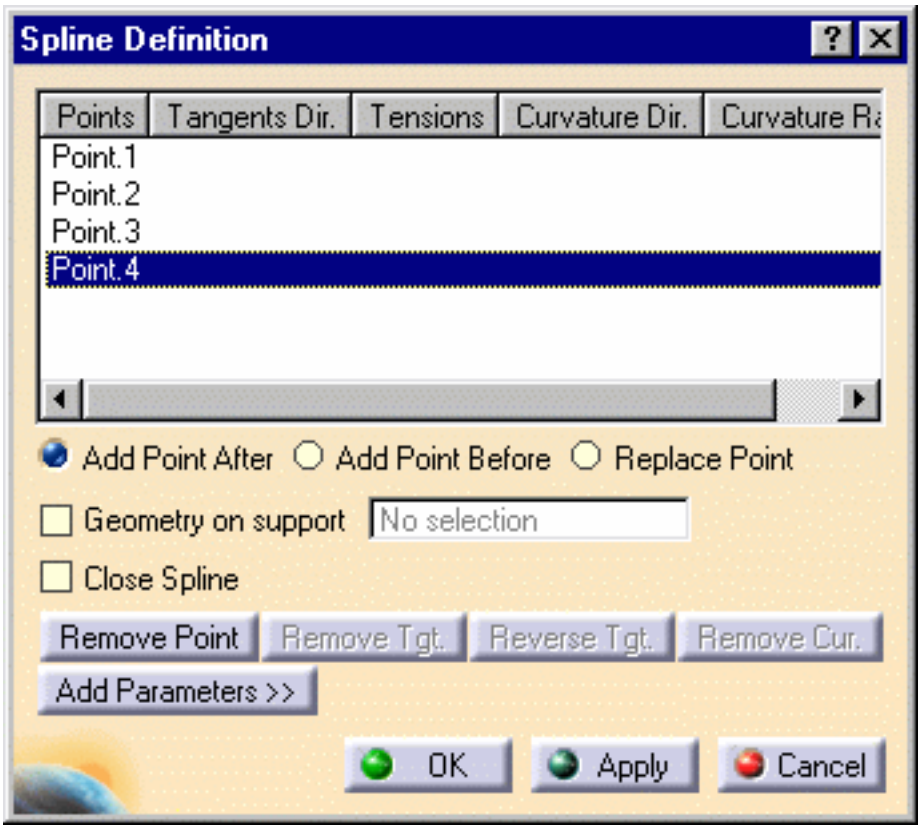
However, specific to wireframe and surface elements are some implicit elements, such as the axis of a cylinder, or the vertex of a cone for example, participating in the creation of a feature yet not directly selectable as a separate element.

This task shows how to select these implicit elements.

Open the Cylinder1.CATPart document.

1. Click the **Spline** icon and successively select the four points.

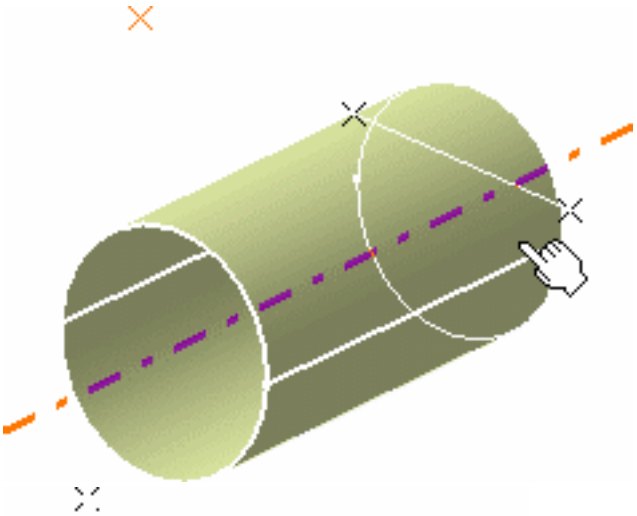
   The Spline Definition dialog box looks like this:



2. Select Point.3 from the list, to impose a tangency constraint on this point.

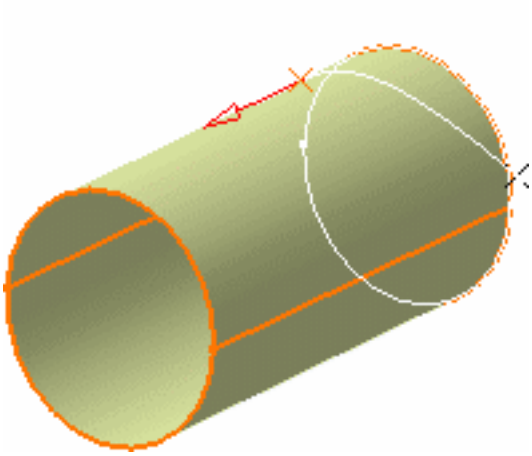
   Note that you cannot select the cylinder's surface.

3. Press and hold the Shift key, then move the pointer over the cylinder. The cylinder's axis is automatically detected as a selectable element to indicate a direction, and displayed.

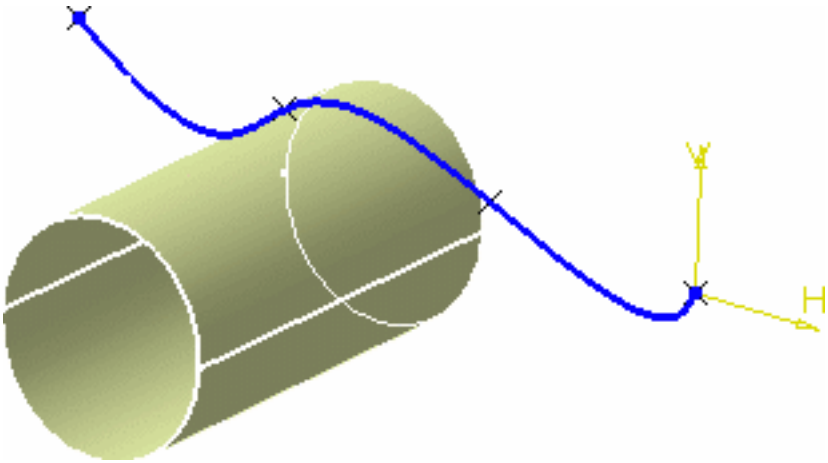


4. Click anywhere on the cylinder's surface, still holding the Shift key pressed down.

   The tangency constraint direction, based on the cylinder's axis, is displayed at the selected point.



5. Click OK to create the spline tangent to the cylinder at the selected point.



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Creating Elements From An External File

You can create points, curves, and lofted surfaces from a Microsoft ® Excel spreadsheet containing macros, and in which you define:

- the points space coordinates
- the points through which the curves pass
- the curves used as profiles for the lofted surface

Only Excel sheets created with Excel 97 and subsequent versions are supported. Therefore this capability is available with Windows™ only.

Open any .CATPart document containing an Open Body (automatically created when entering the Wireframe and Surface Design workbench).

1. Open the PointSplineLoftFromExcel.xls file from the Samples directory into Excel, and enable the macros.

   The document looks like this:

B27 =

	A	B	C	D	E
1	StartLoft				
2	StartCurve				
3	0	-90	10		
4	0	-30	60		
5	0	50	60		
6	0	110	20		
7	EndCurve				
8	StartCurve				
9	50	-60	0		
10	50	-10	40		
11	50	50	40		
12	50	70	0		
13	EndCurve				
14	StartCurve				
15	100	-100	-10		
16	100	-40	35		
17	100	0	50		
18	100	75	40		
19	100	140	0		
20	EndCurve				
21	EndLoft				
22	End				
23					

It contains:

- instructions, such as StartLoft and EndLoft, StartCurve and EndCurve between which other instructions or numerical data are given.
- numerical data that are point space coordinates: X, Y, Z respectively from the left to the right
- a final End instruction

In the above example, a loft is to be created based on three curves. The first and second curve pass through four points, and the third curve passes through five points.

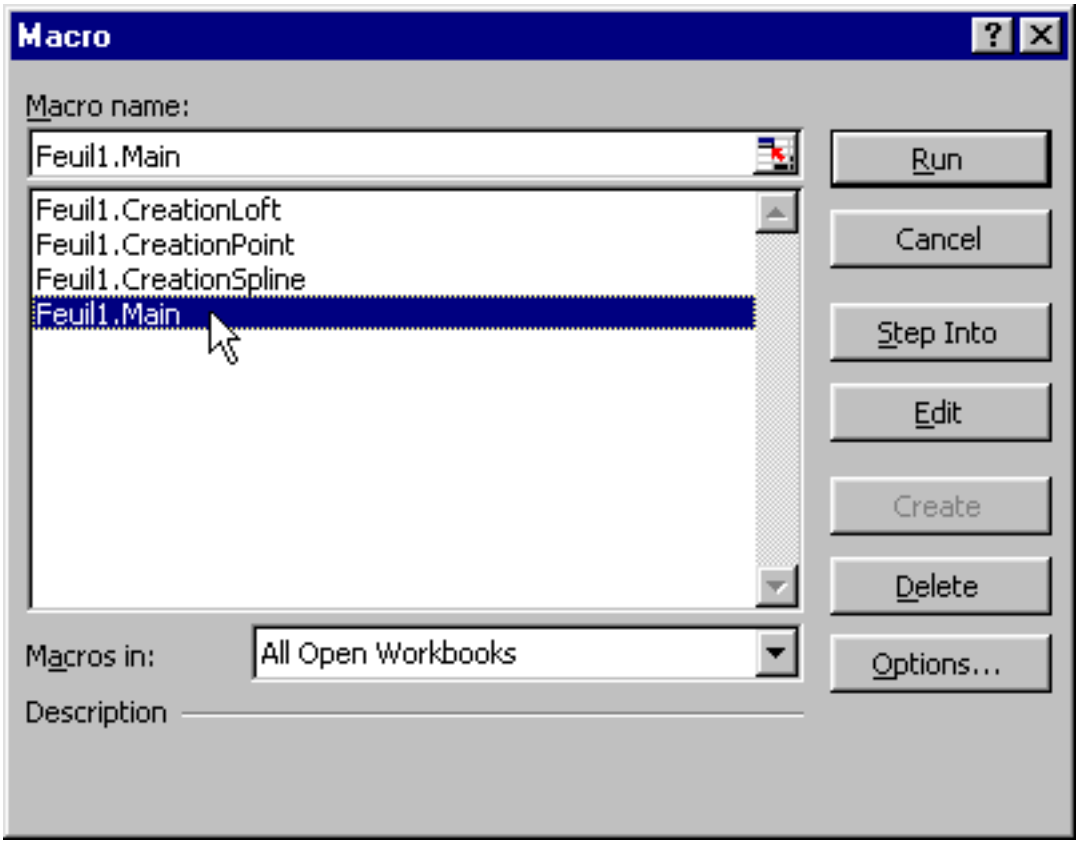
The elements will be created from top to bottom, i.e. the four points of the first curve will be created, then the curve itself, then the points making up the second curve and the latter itself, and so forth.



You can add rows to create more elements or delete rows to edit elements or delete them (point), then save the spreadsheet.

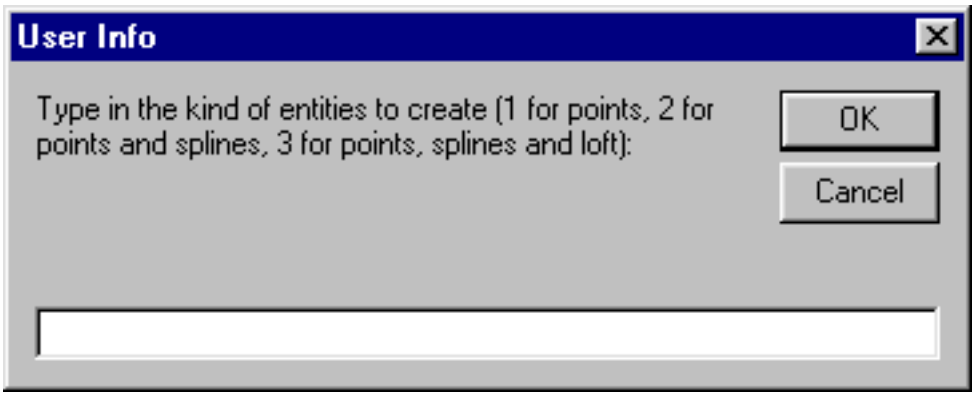
2. From Excel, select the **Tools -> Macro -> Macros** menu item.

   The Macro dialog box is displayed.



3. Select the Feuil1.Main macro and click Run.

The User Info dialog box is displayed.

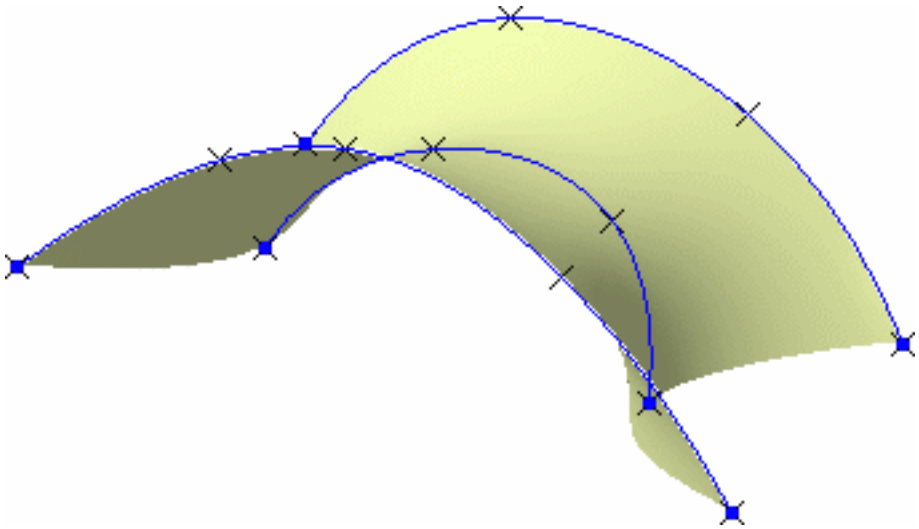


4. Key in the type of element to be generated:

- 1: to generate only the point(s)
- 2: to generate the points and the curve(s)
- 3: to generate the points, curves and lofted surface(s)

5. Click OK.

   The elements (points, curves, and lofted surface) are created in the geometry. The specification tree is updated accordingly.



The Wireframe and Surface workbench needs not to be loaded, provided a CATIA session is running and a .CATPart document is loaded.

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Copying and Pasting

This task shows how to copy and paste open body entities in your part design.

1. Select the elements that you want to copy either directly in the part geometry or in the specification tree.
2. Select the Edit > Copy command.
3. Click the Open Body entity in the tree where you want to paste the selected elements.
4. Select the Edit > Paste command.
   The elements are then copied into the target Open Body.

- The identifiers of copied elements are incremented with respect to the original elements.
- The original elements and copied elements can be edited independently.
- A few elements cannot be copied/pasted as such. They need their parent element to be copied as well. This is the case with boundaries, and extracts, for example. In this case, you may also consider using PowerCopies.

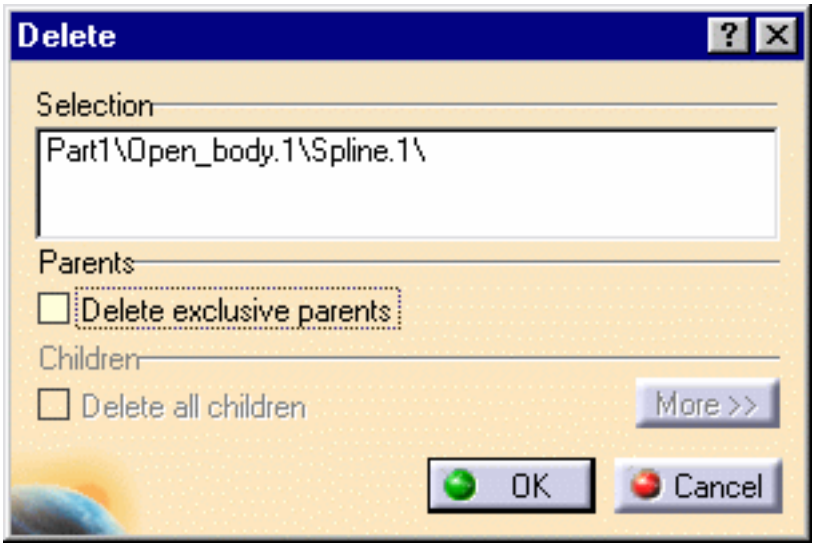
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Deleting Surfaces and Wireframe Geometry

This task shows how to delete geometry from your design.

1. Select the entity you want to delete.
2. Select the **Delete** command either from the the Edit menu or the contextual menu.

   The Delete dialog box appears.



3. Set your desired options for managing the deletion of Parent and Child entities.

   Two options are available:

   1. **Delete exclusive parents**: deletes the geometry on which the element was created. This geometry can be deleted only if it is exclusively used for the selected element
   2. **Delete all children**: deletes the geometry based upon the element to be deleted, in other words, dependent elements

4. Click OK to confirm the deletion.

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Managing Open Bodies in the Specification Tree

This task shows how to manage the specification tree. This involves:

- inserting open body entities
- removing open body entities
- changing body

An open body enables to reorganize the specification tree when it becomes too complex or too long. You can put any element you wish in the open body, it does not have to be structured in a logical way.

- You can insert and manipulate open bodies in the specification tree in much the same way as you manage files in folders.
- These management functions have no impact on the part geometry.
- You should refer to the Copying and Pasting section for information about how open bodies can be used in a part edition context.
- When loading the Generative Shape Design workbench, an Open Body automatically becomes the current body.
  This also means that only the results of the Part Body, i.e. the result of all the operations performed on geometry, is visible and not any intermediate state of the Part Body.

Open any .CATPart document containing OpenBodies.
You can also open the OpenBodies1.CATPart document.

Inserting an Open Body

1. In the specification tree, select the branch of your choice.
   This branch will be considered as a child of the new open body and can be an open body or a feature.

2. Select the **Insert -> Open Body** menu command.

   The result is immediate. CATIA displays this new Open_body.x, incrementing its name in relation to the pre-existing bodies, in the specification tree. It is underlined, indicating that it is the active open body.

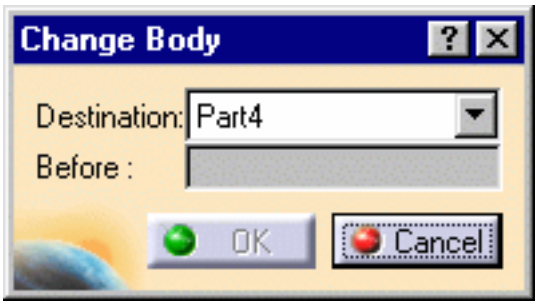
Removing an Open Body

This is only possible when the father location of the open body is another open body.

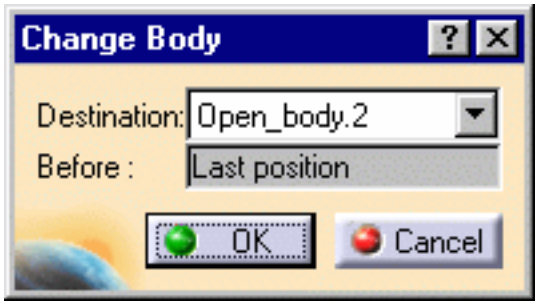
1. Right-click the desired open body then select the **Remove Open Body** contextual command.
   The open body is removed and its constituent entities are included in the father open body.

Moving an open body to a new body

1. Right-click the desired open body in the specification tree and select the **Change Body** command from the contextual menu.
   The Change Body dialog box appears.



2. Select the new body where the open body is to be located.
3. Click OK to move the open body to the new body.



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Hiding/Showing Open Bodies and Their Contents

This task shows how to use the Hide/Show command on different level of open bodies and for different purposes. Indeed you can:

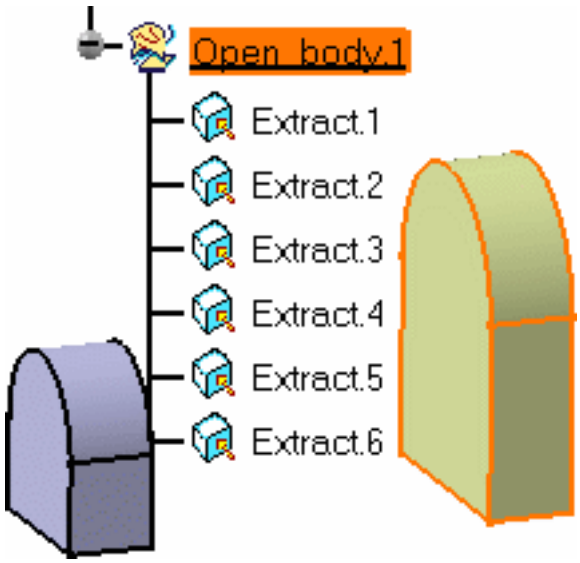
- hide/show complete open bodies
- hide/show contextually on any element making up the open body

Open any .CATPart document containing OpenBodies.
You can also open the OpenBodies1.CATPart document.

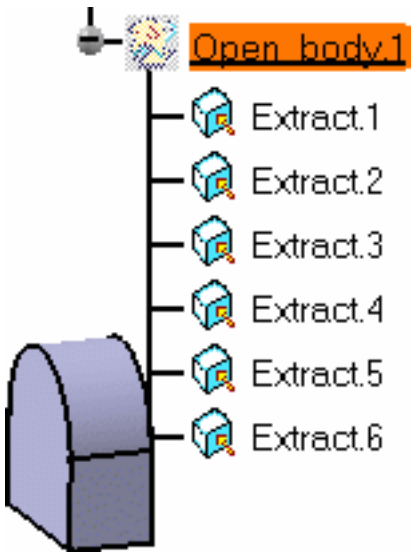
Hiding/Showing an Open Body

1. In the specification tree, select the open body you wish to hide/show
2. Right-click to display the contextual menu and choose the Hide/show command.

The open body is hidden, if it was visible, or becomes visible, if it was hidden.



Visible open body



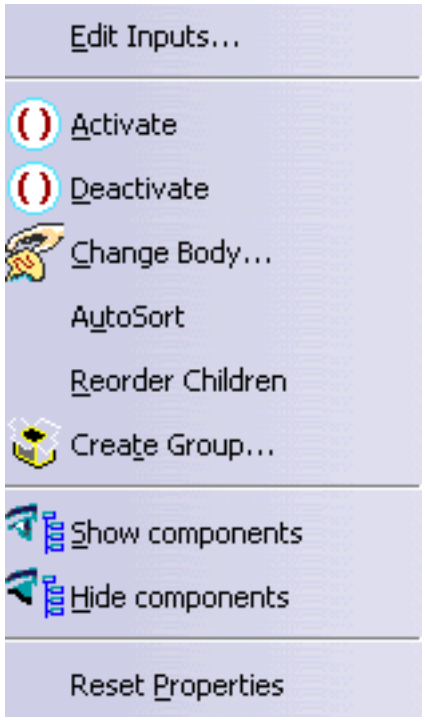
Hidden open body

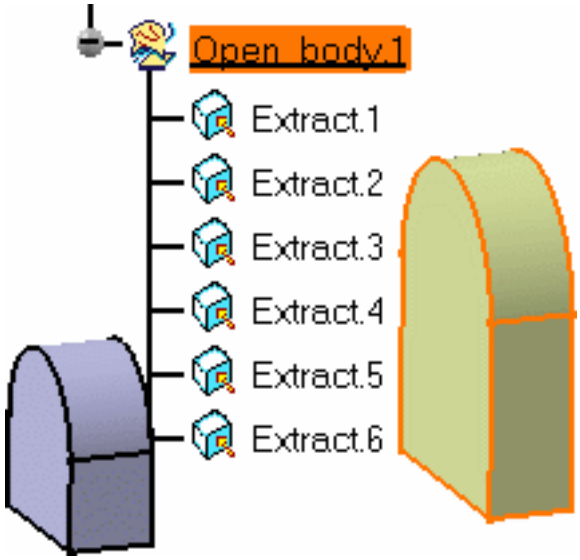
Hiding or Showing an open body as a whole can also be done using the Hide/Show icon.

Hiding/Showing Contents of an Open Body

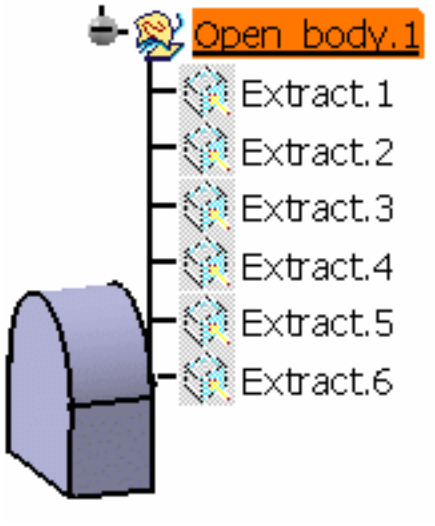
This contextual menu allows you to hide/show all features in an open body (even sketches), whether current or not.

1. In the specification tree, select the open body whose solid elements you want to hide/show.
2. Right-click and choose **Open_Body.x object -> Show components** contextual command to restore the view if the elements were hidden, or **Open_Body.x object -> Hide components** contextual command to hide visible elements.





Visible contents



Hidden contents

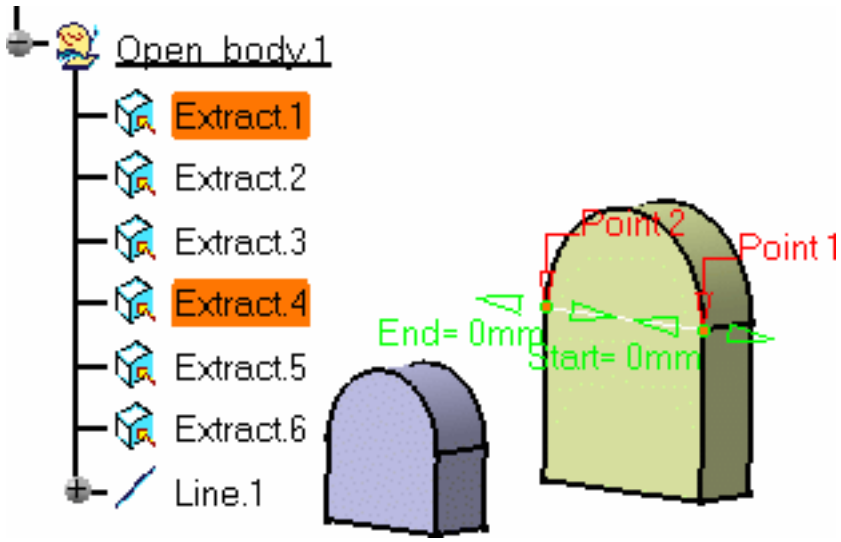
It is advised to use this method to hide contents of an open body, rather than using the **Hide/Show** contextual command: indeed when a body is in show, its contents are as well. This method enables to quickly show an element of an open body.

Hiding or Showing an open body as a whole can also be done using the Hide/Show icon.

Hiding/Showing an element of an Open Body

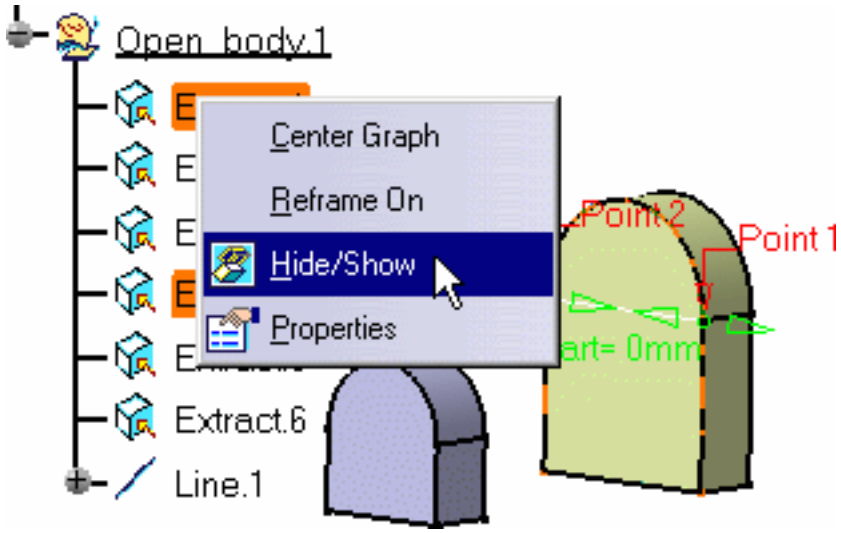
This contextual menu allows you to hide/show an element of the current body, while using a command:

1. Click the **Line** icon and select two points to create a line.

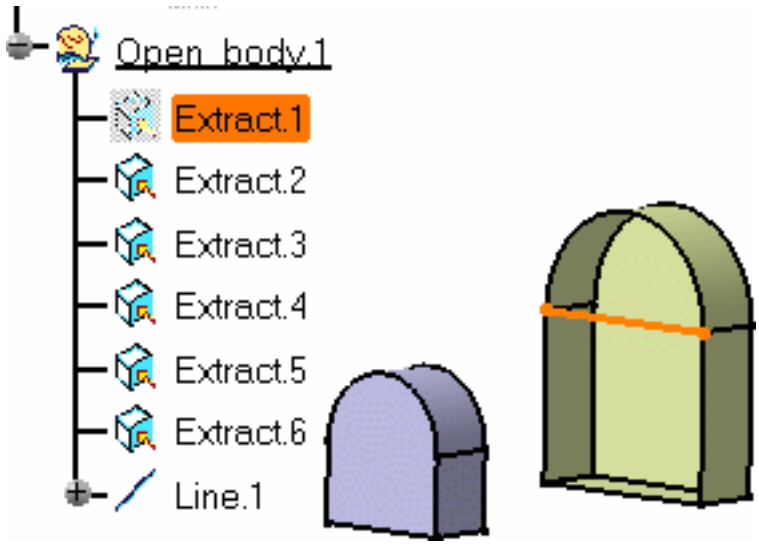


2. Right-click the element to be hidden from the specification tree or the geometry, and choose the Hide/Show contextual command.

   The selected element is hidden without exiting the currently active command.



3. Click OK in the Line dialog box to create the line.



Repeat the operation on the element again to re-display it.

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Checking Connections Between Surfaces

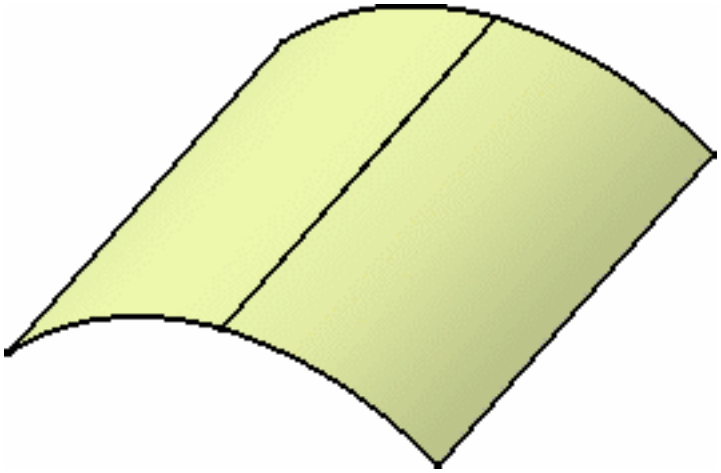


This task shows how to analyze how two surfaces are connected, following a blend, match, or fill operation for example.
Three types of analyses are available.

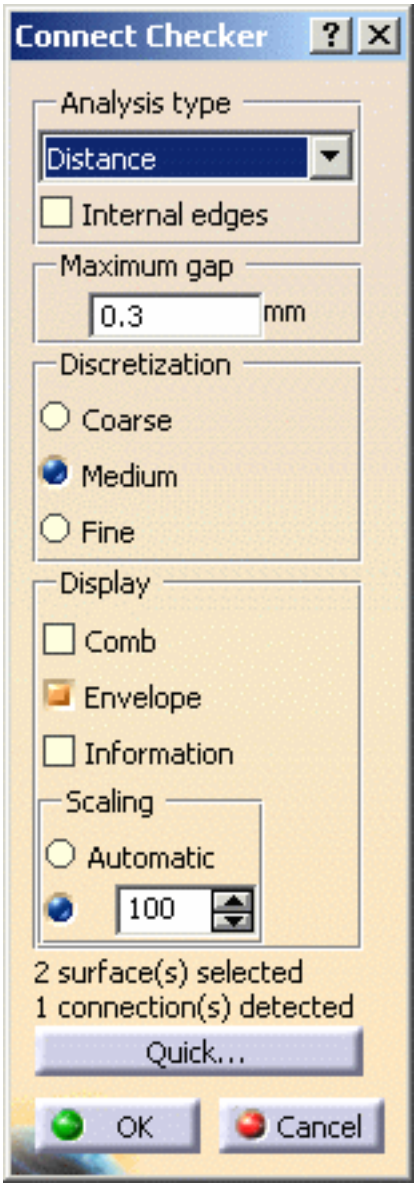
- **Distance**: the values are expressed in millimeters
- **Tangency**: the values are expressed in degrees
- **Curvature**: the values are expressed in percentage.

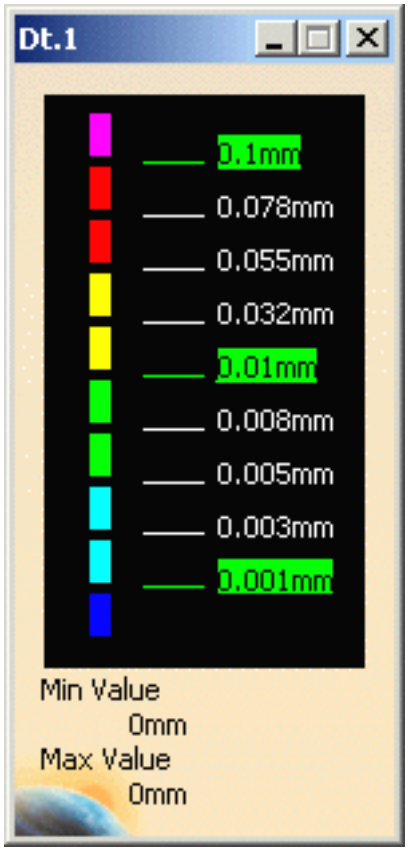
Open the ConnectChecker1.CATPart document.

1. Select both surfaces to be analyzed.
2. Click the **Connect Checker** icon  in the Shape Analysis toolbar.



The Connect Checker dialog box is displayed as well as another dialog box showing the color scale and identifying the maximum and minimum values for the analysis type.



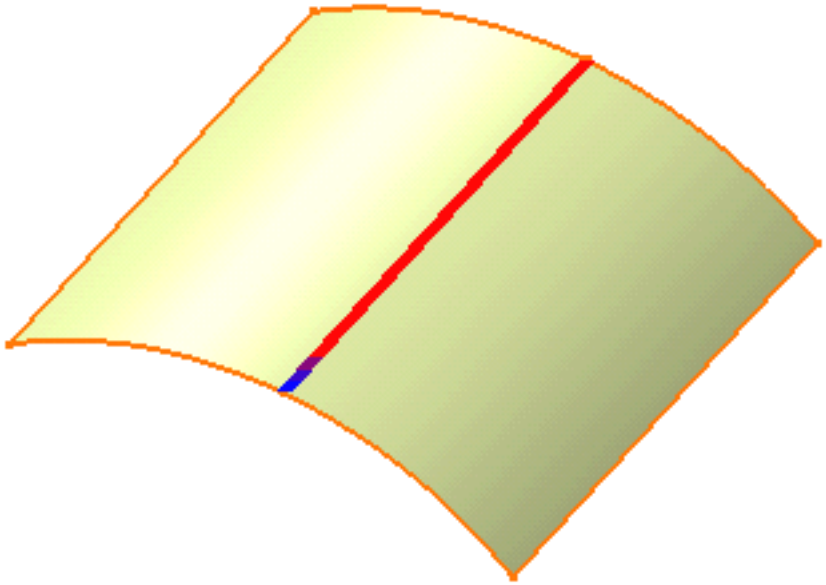


Check the Internal edges option if you want to analyse the internal connections
By default, the check box is unchecked.

3. Choose the analysis type to be performed: **Distance**, **Tangency** or **Curvature**.
4. Set the **Maximum gap** above which no analysis will be performed. All elements apart from a greater value than specified in this field are considered as not being connected, therefore do not need to be analyzed.
   Be careful not to set a **Maximum gap** greater than the size of the smallest surface present in the document.

5. Check the analysis results on the geometry.

   Here we are analyzing the distance between the surfaces. Each color section indicates on the geometry the distance between the surfaces.



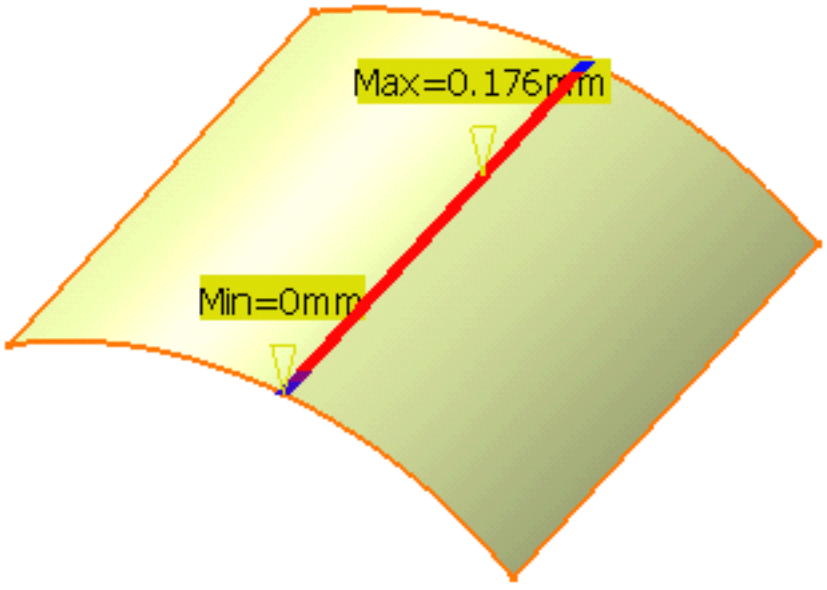
From the Connect Checker dialog box, you can choose a number of visualization and computation options:

- the **comb**: that is the spikes corresponding to the distance in each point
- the **envelope**: that is the curve connecting all spikes together
- **Information**: that is the minimum and maximum values

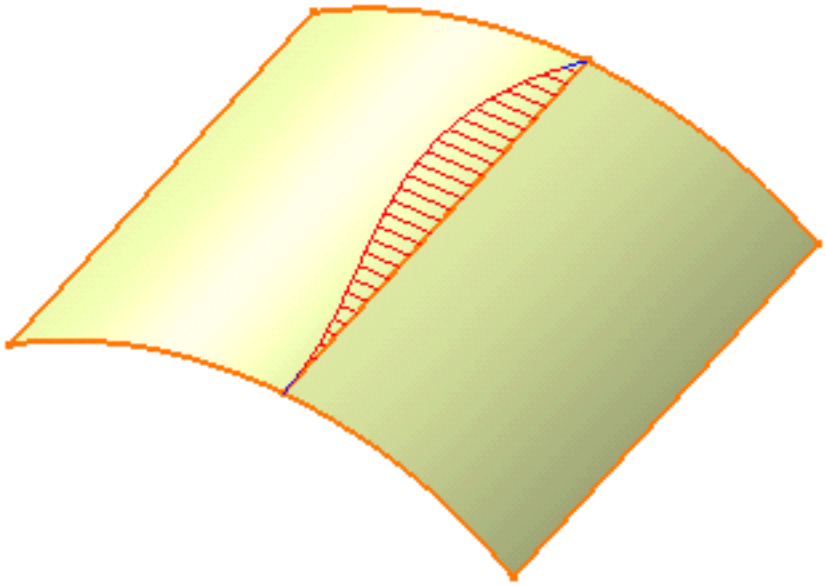
Finally, the scaling option lets you define the visualization of the comb. In automatic mode the comb size is zoom-independent and always visible on the screen, otherwise you can define a coefficient multiplying the comb exact value.

6. Check the **Information** button:

   Two texts are displayed on the geometry localizing the minimum and maximum values of the analysis as given in the Connect Checker dialog box.



You can also choose the discretization, that is the numbers of spikes in the comb (check the **Comb** option to see the difference):



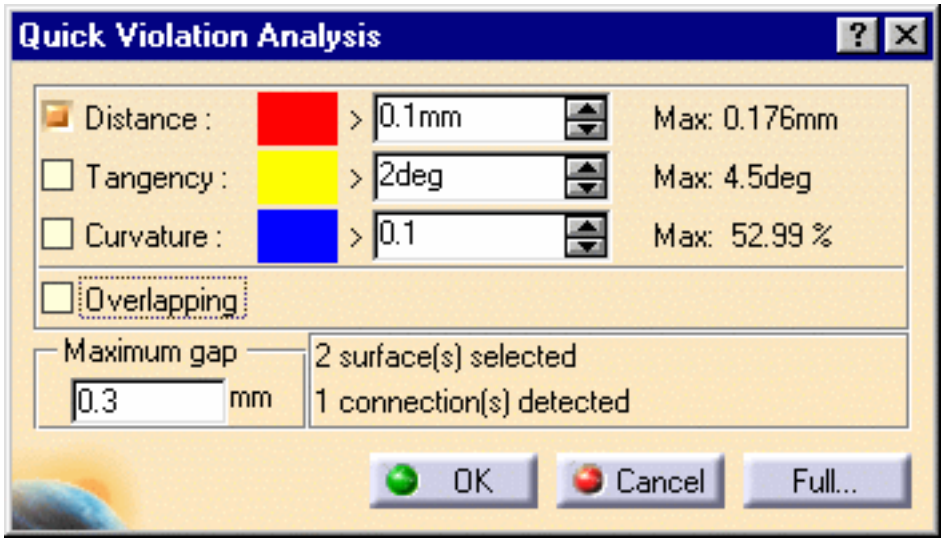
- **Coarse**: 15 spikes are displayed
- **Medium**: 30 spikes are displayed
- **Fine**: 45 spikes are displayed.

The Full result is only available with the Generative Shape Design product.

The number of selected elements and the number of detected connections are displayed below the color range.

7. Click the **Quick...** button to obtain a simplified analysis taking into account tolerances (either distance, tangency, or curvature).

The comb is no longer displayed. The Connect Checker dialog box changes to this dialog box.



You can use the check button to switch from one analysis type to another.
The **Maximum gap** and information are

retained from the full analysis.
The maximum deviation value is also displayed on the geometry.

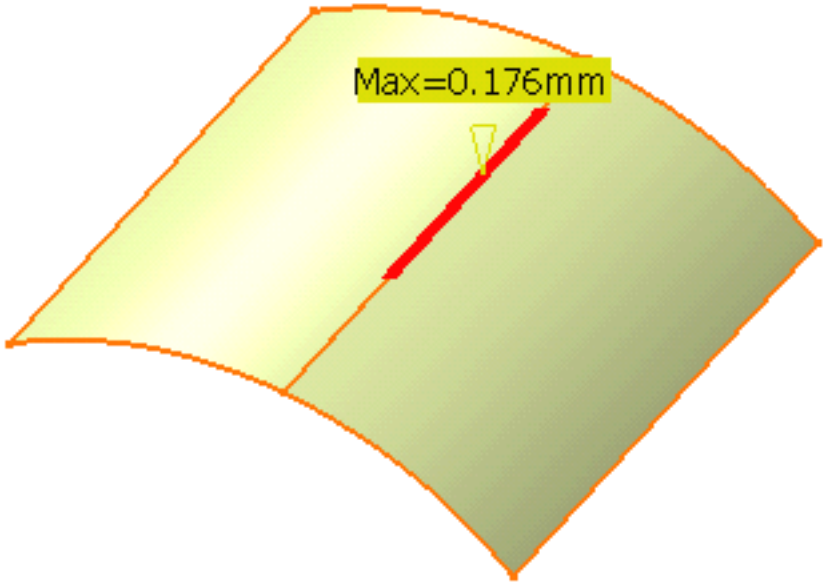
You can check the **Overlapping** button to highlight where, on the common boundary, the two surfaces overlap. In this case the other analysis types are deactivated.

P1 In P1 mode, only the quick analysis is available.

8. Use the spinners to define the tolerances.

   For example, the red area indicates all points that are distant of more than 0.1 mm.

   The maximum deviation values on the current geometry are displayed to the right of the dialog box.



9. Click **OK** to create the analysis.

   The analysis (identified as Surface Connection Analysis.x) is added to the specification tree (P2 only).

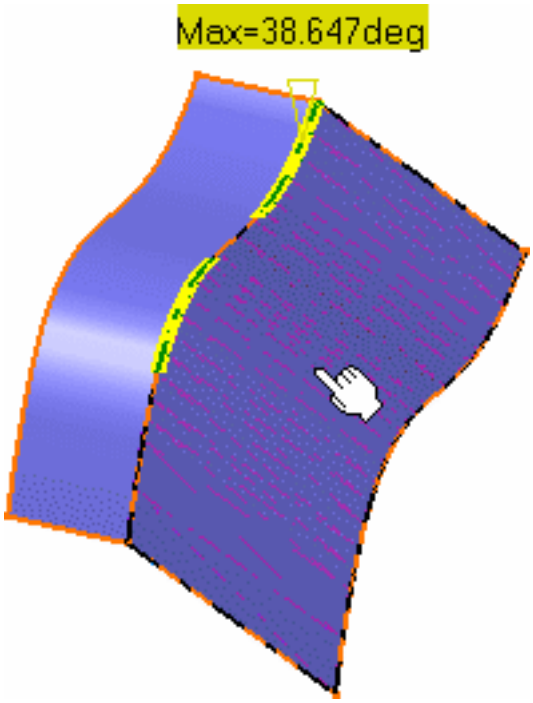
   This allows the automatic update of the analysis when you modify any of the surfaces, using the control points for example.
   If you do not wish to create the analysis, simply click **Cancel**.

- You can edit the color range in both dialog boxes by double-clicking the color range manipulators (Connect Checker) or color areas (Quick Violation Analysis) to display the **Color** chooser.
- If you wish to edit the Connection Analysis, simply double-click it from the specification tree.
- If you no longer need the Connection Analysis, right-click **Connection Analysis** in the specification tree, and choose **Delete**.
- The curvature difference is calculated with the following formula:

  $(|C_2 - C_1|) / ((|C_1 + C_2|) / 2)$

  The result of this formula is between 0% et 200%.

- You can analyze internal edges of a surfacic element, such as a Join for example, by selecting only one of the initial elements:



- You can create an analysis on an entire open body simply by selecting it in the specification tree.

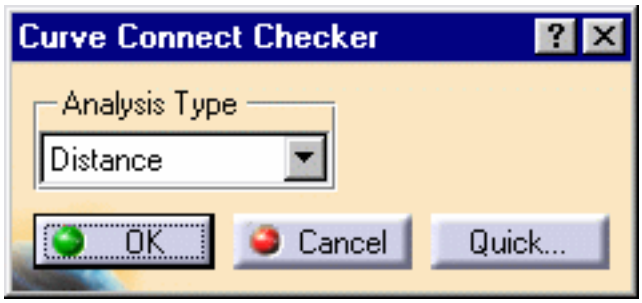
Checking Connections Between Curves

This task shows how to analyze how two curves are connected, following a blend, or match operation for example.
Four types of analyses are available.

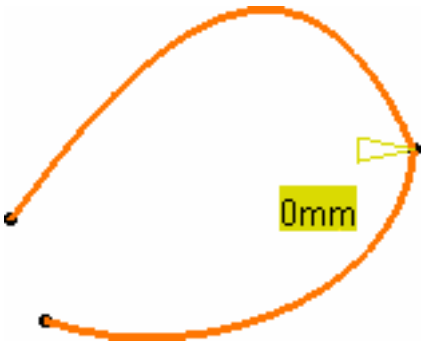
- **Distance**: the values are expressed in millimeters
- **Tangency**: the values are expressed in degrees
- **Curvature**: the values are expressed in percentage
- **Overlapping**: the system detects overlapping curves

Open the ConnectChecker2.CATPart document.

1. Select both curves to be analyzed.
2. Click the **Curve Connect Checker** icon in the Shape Analysis toolbar.



The Connect Checker dialog box is displayed. At the same time a text is displayed on the geometry, indicating the value of the connection deviation.

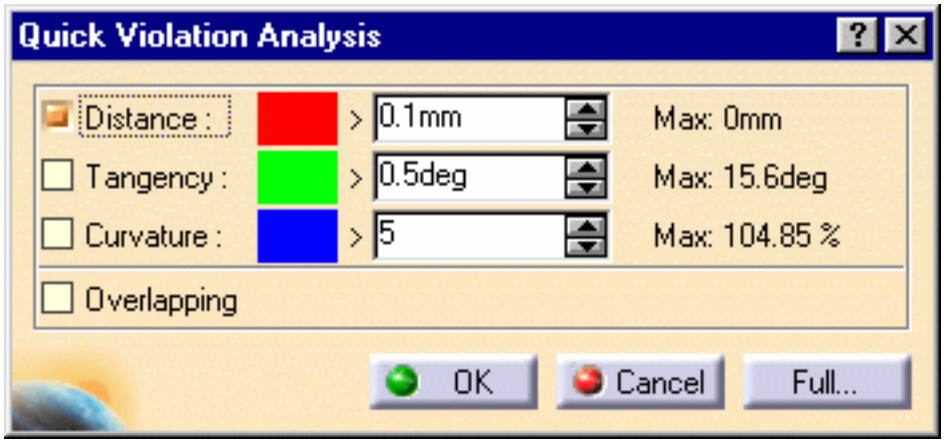


You can choose the type of analysis to be performed using the combo: distance, tangency or curvature.

In P1 mode, only this mode is available (no quick mode available).

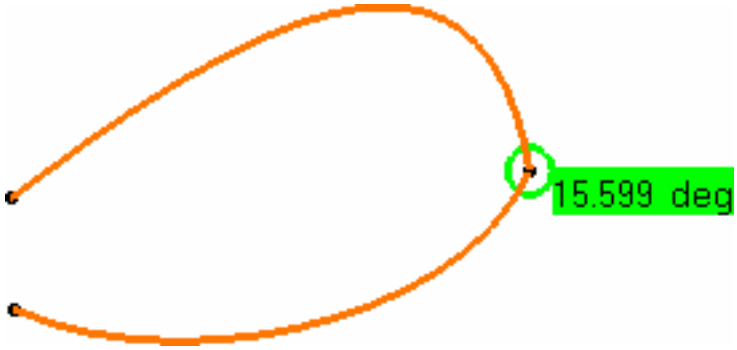
P2 This step is P2 only for Wireframe and Surface.

3. Press the Quick button.



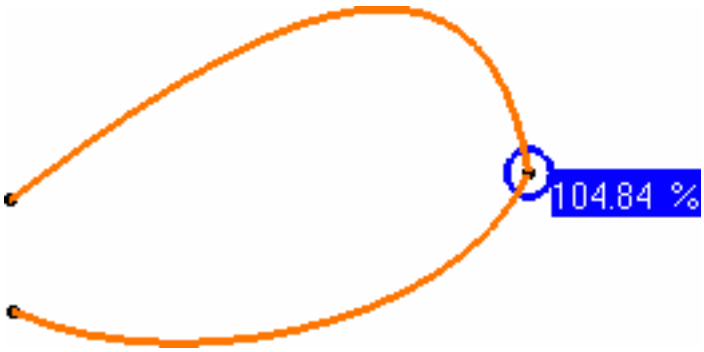
The dialog box changes along with the text on the geometry..
With our example, the text in the geometry disappears because the distance between the two curves is smaller than the set Distance value.

4. Check the **Tangency** button:



A text is displayed on a green background (as defined by default for the Tangency criterion) to indicate that the Tangency criterion is not respected, because the first text displayed is the one for which the set tolerance is not complied with. You can then increase the Tangency value, or modify the geometry to comply with your needs.

5. Similarly, if you check the **Curvature** value, the displayed text indicates that the curvature between the two analyzed curves is greater than the set value.



6. Modify the tolerance values, or the geometry to comply with the tolerances.

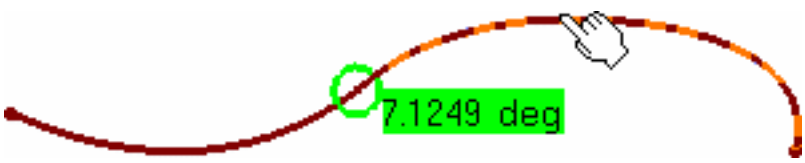
   For example, if you modify the Tangency value to set it to 16 degrees, the geometry instantly reflects the compliance with the new value.

The maximum deviation values on the current geometry are displayed to the right of the dialog box.

7. Click **OK** to create the analysis.

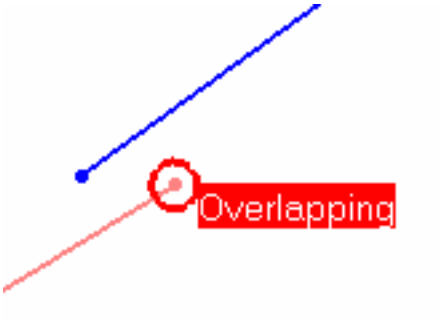
   The analysis (identified as Curve Connection Analysis.x) is added to the specification tree.
   This allows the automatic update of the analysis when you modify any of the curves, using the control points for example (see Editing Curves Using Control Points).
   If you do not wish to create the analysis, simply click **Cancel**.

- Double-click the Curve Connection Analysis from the specification tree to edit it.
- You can analyze internal edges of a element, such as a Join for example, by selecting only one of the initial elements:



- Use the **Overlapping** mode to highlight where, on the common boundary, the two curves overlap.
  When the **Overlapping** button is checked, other analysis types are deactivated.
  In Full mode, a text is displayed indicating whether the curves overlap.

The **Overlapping** mode is not available with the Wireframe and Surface product.





- The curve connection checking analysis is permanent in P2 mode only, i.e. it is retained in the specification tree for later edition and on the geometry till you reset or delete it, whereas in P1 mode, it is present at a time, but not retained when exiting the command.

Repeating Objects

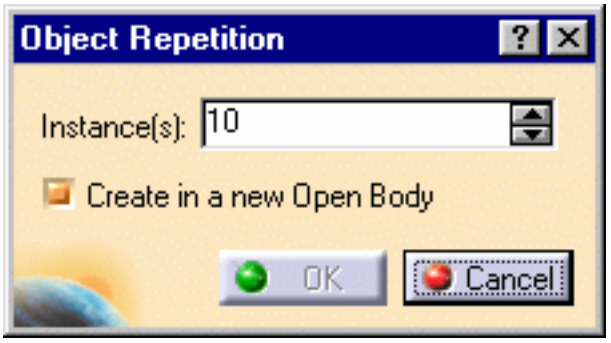
This task shows how to create several instances of objects as you are currently creating one object.
This command is available for:

- points on a curve
- lines at an angle or normal to a curve
- planes at an angle
- offset planes
- offset surfaces
- or when performing a translation, a rotation or a scaling on an object.

1. Select an object, as listed above.

2. Click the **Object Repetition** icon or select the **Insert -> Advanced Replication Tools -> Object Repetition...** menu item.

   The **Object Repetition** dialog box is displayed.



3. Key in the number of instances of the object you wish to create.

4. Check the **Create in a new open body** if you want all object instances in a separate Open Body.
   A new Open Body will be created automatically.

   If the option is not checked the instances are created in the current Open Body.

5. Click OK.
   The object is created as many times as required in the **Object Repetition** dialog box.

See each specific object creation for further details on the what parameter is taken into account for the repetition.



Back Up Next

Stacking Commands

This task shows how to stack commands, that is create another basic object in the current command without leaving it.

Let's take an example with the Line functionality.

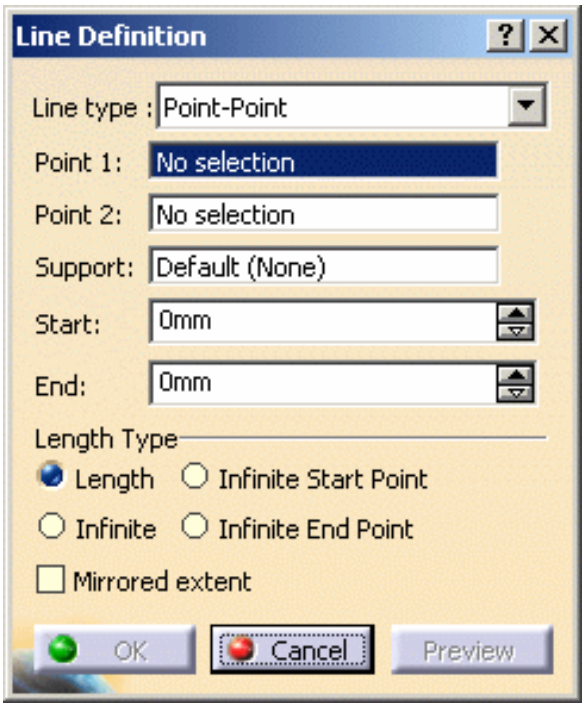
Open a new Part document.

1. Click the **Line** icon .

   The Line Definition dialog box appears.

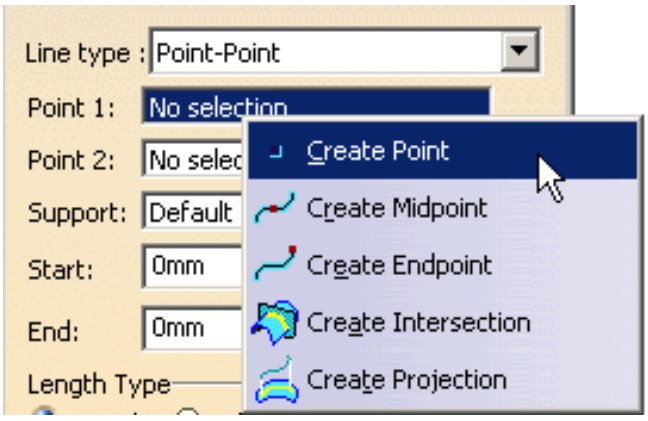
2. Use the combo to choose the desired line type.

   Here we chose the **Point-Point** line type: two points are required to create the line.



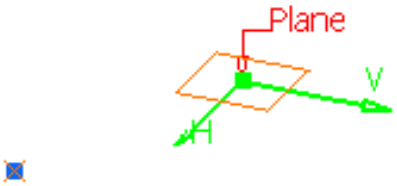
As no point already exits, you will have to create them.

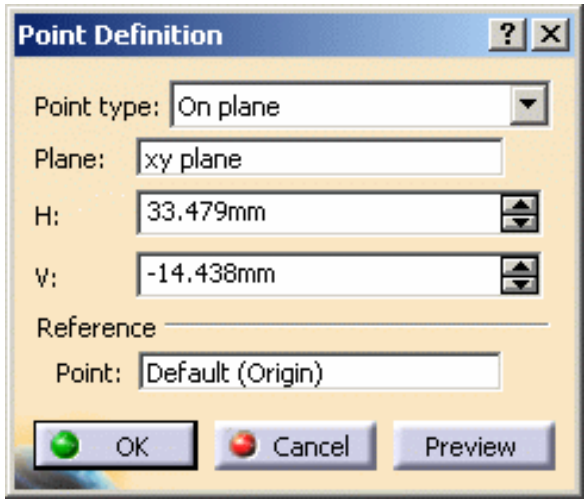
3. Right-click the Point 1 field.
4. Select **Create Point** from the contextual menu.



The Point Definition dialog box displays.

5. Use the combo to choose the desired point type.
   Here we chose the **On Plane** type.
6. Choose the **Plane**.
7. Click **OK**.

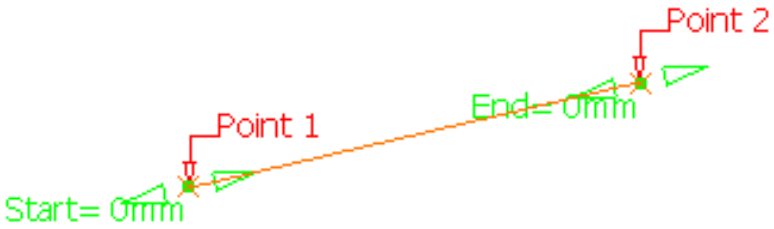


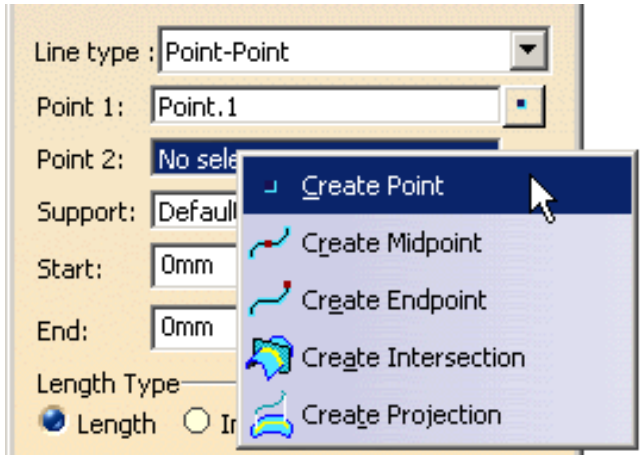


The Point Definition dialog box closes and you return to the Plane Definition dialog box. The Point.1 field is valuated with the point you just created.

8. Right-click the Point 2 field.
9. Repeat steps 4 to 7.

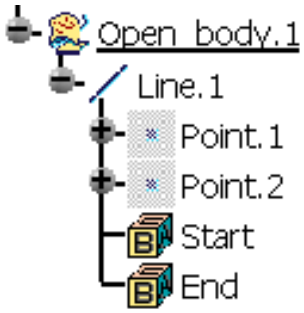
The Point Definition dialog box closes and you return to the Plane Definition dialog box.
The Point.2 field is valuated with the point you just created and a line is previewed between Point 1 and Point 2.





10. Click OK to create the line.

Features created using stacked commands are now aggregated under the parent command that created them and put in no show in the specification tree.



Editing Parameters

This task shows how to view dimensions in the 3D geometry when creating or editing a feature.

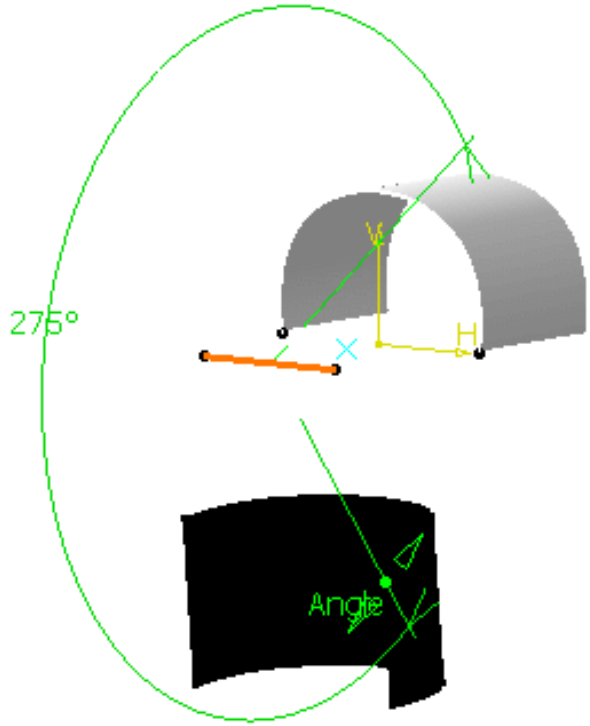
This command is available on the following commands:

Operator	Type	Sub- Type	Parameter displayed
Circle	Center and Radius		Radius, Start Angle, End Angle
	Center and Point		Start Angle, End Angle
	Two Points and Radius		Radius
	Bitangent and Radius		Radius
	Center and Tangent	Point as center element	Radius
Corner			Radius
Curve Parallel			Constant (Offset Distance)
Diabolo			Draft Angle
Extrapolate	Length		Length, Limit Type
Extrude			Length 1, Limit 1 Length 2, Limit 2
Helix			Taper Angle, Starting Angle
Line	Angle/Normal to Curve		Angle
	Point-Point		Length : Start, End Infinite Start Point: End Infinite End Point: Start Infinite: /
	Point-Direction		
	Angle-Normal to Curve		
	Tangent to Curve		

	Normal to Surface		
	Bisecting		
Offset			Offset Value
Plane	Angle/Normal to Plane		Angle (Angle/Normal to Plane and Angle/ Normal to Curve)
	Offset from Plane		Length, Offset Distance
Point	Coordinates		Length, X, Y, Z coordinates
	On Curves		Length, Length
	On Plane		Length, H, V
	On Surface		Length, Distance
Polyline			Radius, Radius at point
Reflect Line			Angle
Revolve			Angle1, Angle2
Rotate			Rotation Angle
Shape Fillet	Bi-Tangent Fillet		Radius
Sphere			Parallel Start Angle, Parallel End Angle, Meridian Start Angle, Meridian End Angle
			Radius, Radius
Spiral			End Angle
Sweep	Explicit Sweep		Angle
	Linear Sweep	Two Limits	Length1, Length2
		With Reference Surface	Angle, Length1, Length2
		With Reference Curve	Angle, Length1, Length2
		With Draft Direction	Angle, Length1, Length2
Translate	Distance and Direction		Distance

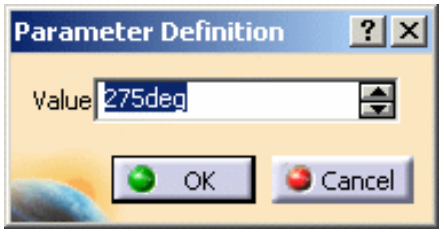
Create any of the features above.
Let's take an example by performing a rotation.

1. Once you selected the inputs to create the rotated element, click **Preview** to display the associated parameters in the 3D geometry.



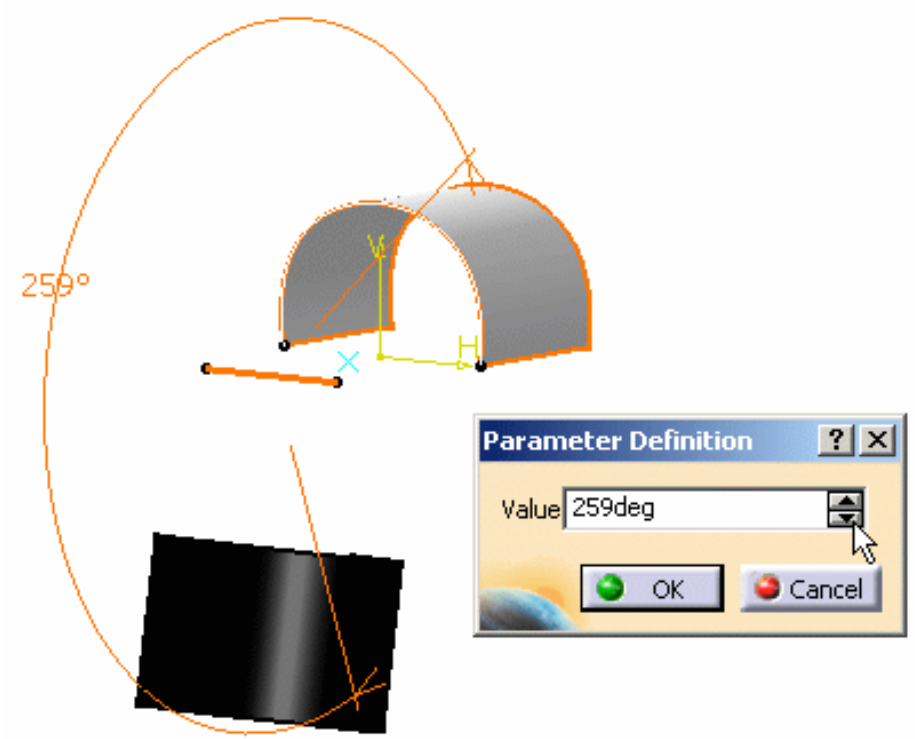
2. Double-click the angle value in the 3D geometry.

   The Parameter Definition dialog box appears.

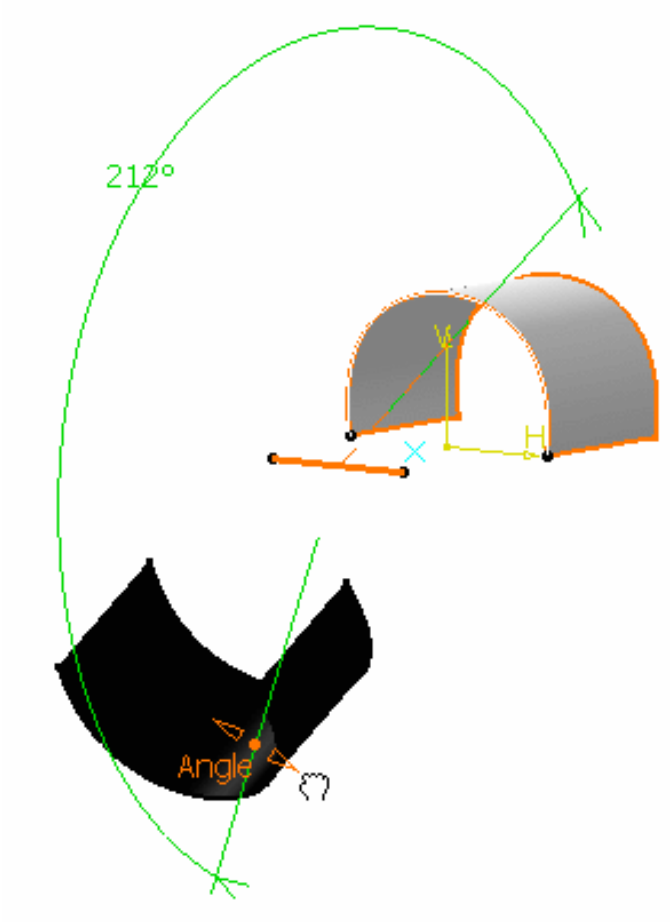


3. Use the spinners to modify the value.

The display automatically updates and the object is modified accordingly.



You can also modify the angle value using the Angle manipulators.



- To display the parameters' values, you need to click the **Preview** button. Otherwise, only manipulators are displayed.
- To edit the parameters once the feature is created, select it in the specification tree, right-click **xxx.1object -> Edit Parameters** from the contextual menu.
- If you want the parameters to be kept permanently, check the Parameters of features and constraints option in **Tools -> Options -> Infrastructure -> Part Infrastructure -> Display**.

Selecting Using Multi-Selection

This capability enables you to perform multi-selection of elements and validate the selection.
It is available with the following functionalities:

- translate
- project
- axis transformation
- rotate
- symmetry
- scaling
- affinity
- intersection
- extract
- develop

Select one of the commands above: the definition dialog box displays, as long as the Tools Palette toolbar.



1. Choose the selection type:

- **Select** : enables you to select elements or deselect elements in the 3D geometry or in the specification tree.
  Use the Ctrl key to select several elements, and the Shift key to deselect already selected elements.
- **Selection Trap** : enables you to select elements by drawing a trap.
  Elements must be entirely located inside the trap to be selected.
- **Intersecting Trap** : enables you to select elements by drawing a trap.
  Elements can either be located inside the trap or be intersected by the trap to be selected.

- **Polygon Trap** : enables you to select elements by drawing a closed polygon. Any element inside the polygon will be selected.
- **Paint Stroke Trap** : enables you to select elements by drawing a paint stroke across them.

The Tools Palette closes and you go back to definition dialog box.


Multi-selection is available when editing a single feature: double-click it in the specification tree to display the Tools Palette and perform multi-output selection.



Creating Projections

This task shows you how to create geometry by projecting one or more elements onto a support. The projection may be normal or along a direction.
You can project:

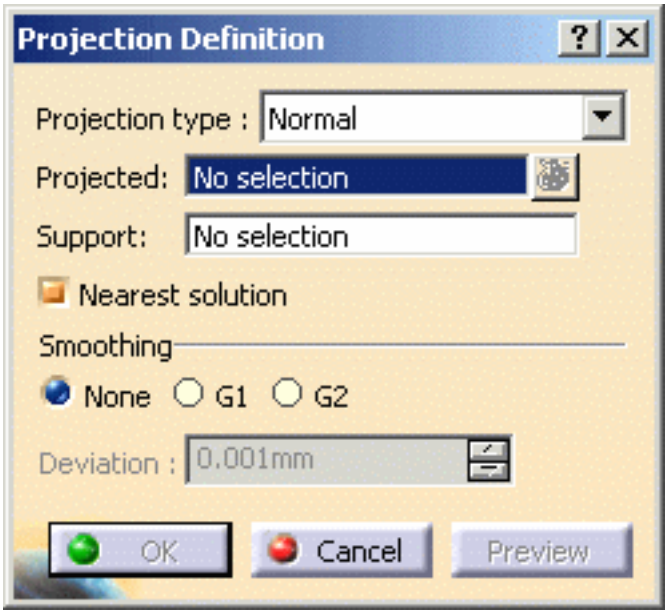
- a point onto a surface or wireframe support
- wireframe geometry onto a surface support
- any combination of points and wireframe onto a surface support.

Generally speaking, the projection operation has a derivative effect, meaning that there may be a continuity loss when projecting an element onto another. If the initial element presents a curvature continuity, the resulting projected element presents at least a tangency continuity. If the initial element presents a tangency continuity, the resulting projected element presents at least a point continuity.

Open the Projection1.CATPart document.

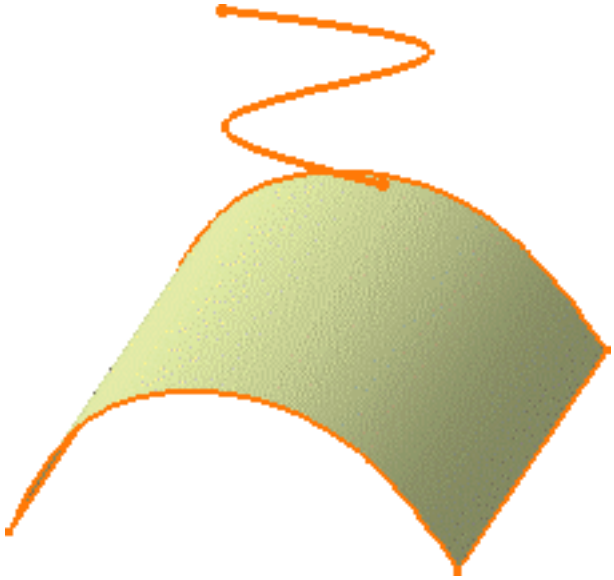
1. Click the **Projection** icon .

The Projection Definition dialog box appears as well as the Multi-Selection dialog box allowing to perform multi-selection.



2. Select the element to be **Projected**.

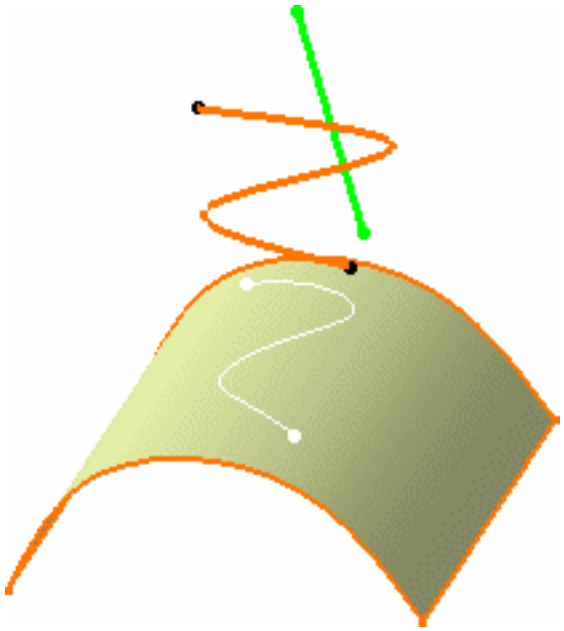
   You can select several elements to be projected. In this case, the **Projected** field indicates: **x elements**



3. Select the **Support** element.

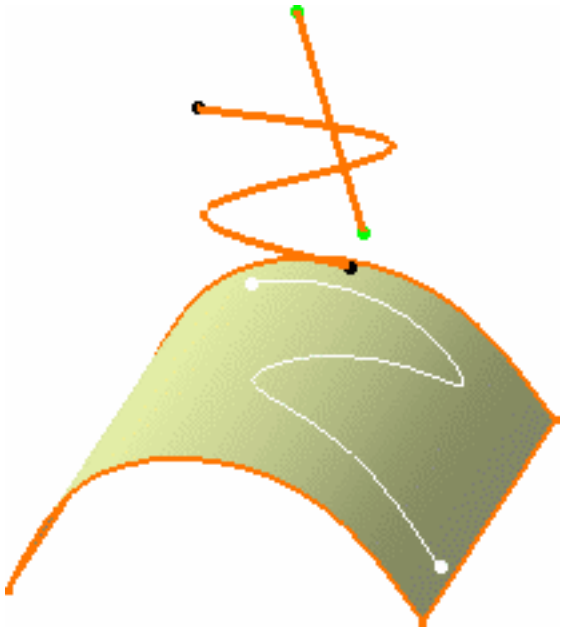
4. Use the combo to specify the direction type for the projection:

- **Normal**: the projection is done normal to the support element.



- **Along a direction**: you need to select a line to take its orientation as the translation direction or a plane to take its normal as the translation direction.

You can also specify the direction by means of X, Y, Z vector components by using the contextual menu on the **Direction** field.



- Whenever several projections are possible, you can select the **Nearest Solution** check box to keep the nearest projection.
  The nearest solutions are sorted once the computation of all the possible solutions is performed.
- You can smooth the element to be projected by checking either:
  - **None**: deactivates the smoothing result
  - **G1** : enhances the current continuity to tangent continuity
  - **G2** : enhances the current continuity to curvature continuity
- You can specify the maximum **deviation** for G1 or G2 smoothing by entering a value or using the spinners.

  If the element cannot be smoothed correctly, a warning message is issued.

  Moreover, a topology simplification is automatically performed for G2 vertices: cells with a curvature continuity are merged.



7. Click OK to create the projection element.

The projection (identified as Project.xxx) is added to the specification tree.

The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.



Selecting Using Multi-Output

This capability enables to keep the specification of a multi-selection input in a single operation.
It is available with the following functionalities:

- Intersections
- Projections
- Extracting
- All transformations: translation, rotation, symmetry, scaling, affinity and axis to axis
- Developed wires (Generative Shape Optimizer)

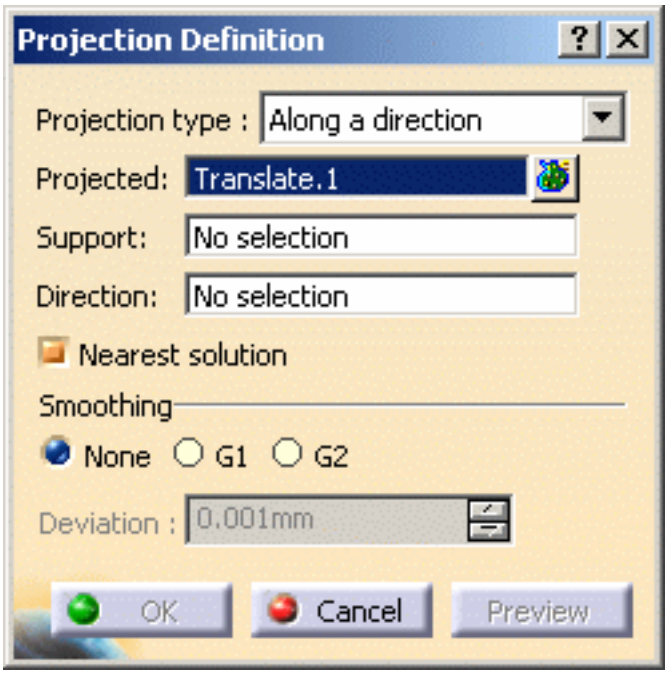
Let's take an example using the Projection and Translation functionalities.

Open the Multi-Output1.CATPart document.

1. Click the **Projection** icon .

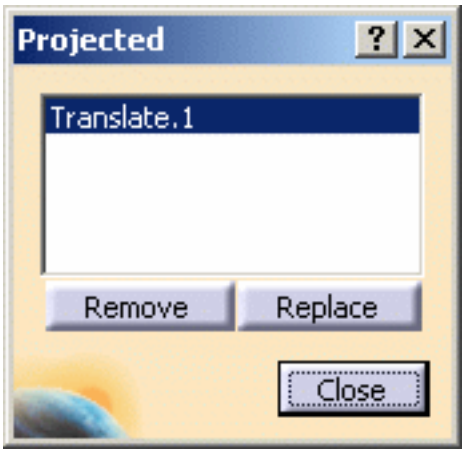
The Projection Definition dialog box appears, as long as the Tools Palette toolbar.

2. Select Translate.1 as first element to be **Projected**.



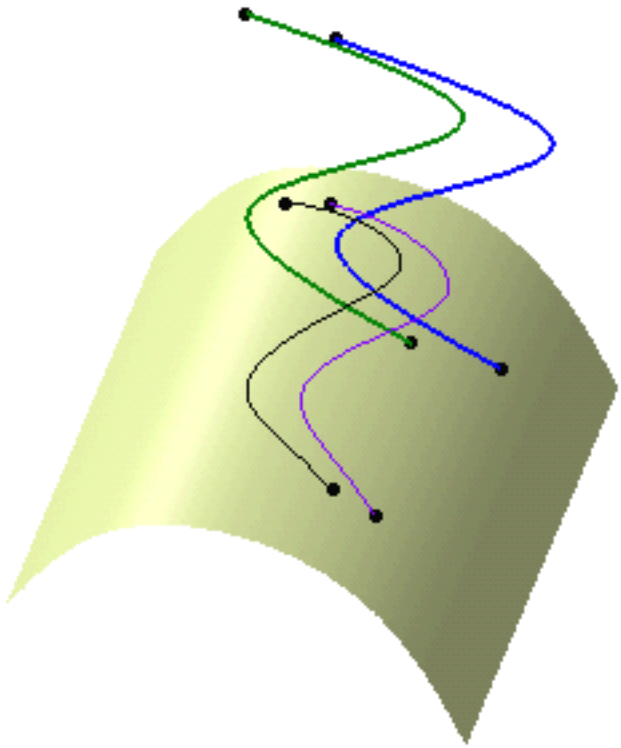
2. Click the bag icon  to display the elements list.
   The Projected dialog box opens.
3. Select as many elements as you need for your projection.
4. Click **Close** to return to the Projection Definition dialog box.

   The number of selected elements is displayed **Projected** field.

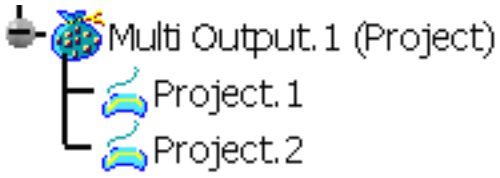


Use the **Remove** and **Replace** buttons to modify the elements list.

2. Select Extrude.1 as the **Support** element.
3. Select **Normal** as Projection type.
4. Click **OK** to create the projection elements.



The projection is identified as Multi Output.1 (Project) in the specification tree.

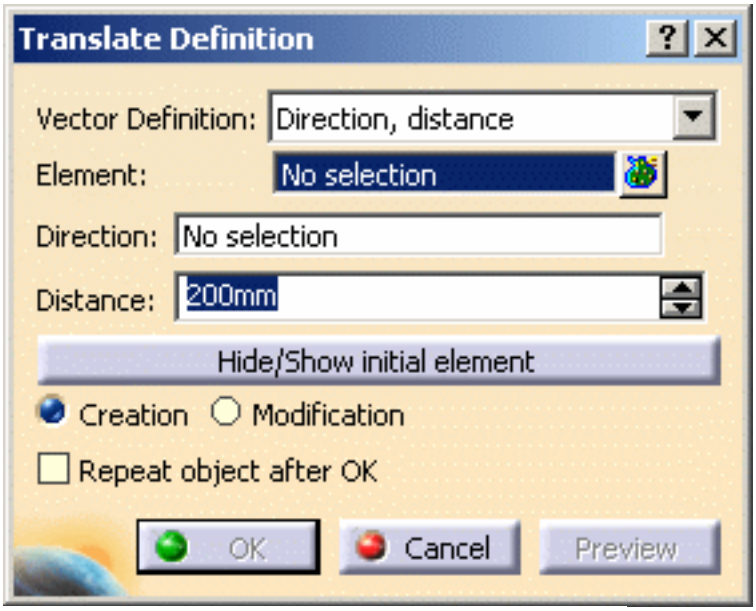


The created elements are aggregated under Multi Output.1.

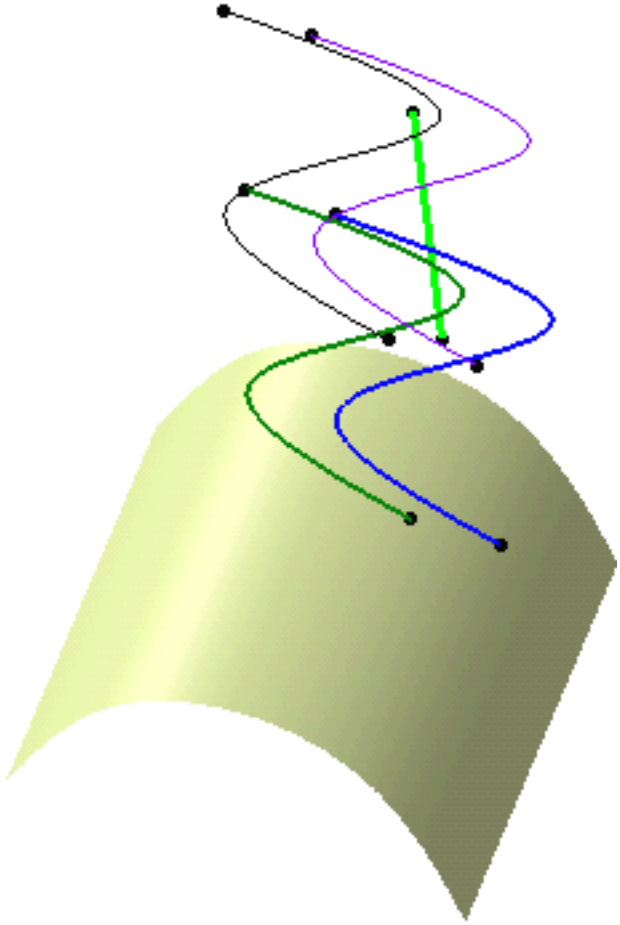
You can create several multi-outputs in the specification tree, each one grouping one type of elements.

5. Click the **Translate** icon .

   The Translate Definition dialog box appears.



6. Select Element.1 and Element.2 as the **Element** to be translated.
7. Select Direction, distance as the **Vector Definition**.
8. Select Extract.2 as the **Direction**.
9. Select -50mm as the **Distance**.
10. Click **OK** to create the translated element.



The translation is identified as Multi Output.2 (Translate) in the specification tree and appears below Multi Output.1.

The created elements are aggregated under Multi Output.2.



When one or several features are in error under a multi-output (during creation or edition), an error message is issued after clicking **Preview** or **OK** and displays all features in error.
You are able to manually delete or deactivate the feature(s) in error. When editing the multi-output, deactivated features are not automatically activated.

- Multi-selection is available when editing a single feature: double-click it in the specification tree and click the bag icon to replace it or add new elements.
- Multi-outputs and elements aggregated under a multi-output can be edited separately, simply by double-clicking it in the specification tree. Elements can be modified (added, replaced, or removed): the corresponding multi-output automatically updates.

Unshared features are aggregated under the parent command that created them and put in no show in the specification tree.
Shared features are not aggregated under the parent command.

- The Datum capability is available. If an element is in error, it cannot be created as a datum element; only elements that could be generated from the multi selection are created.



Applying a Material

This task explains how to apply a pre-defined material as well as to interactively re-position the mapped material.

A material can be applied to:

- a PartBody, Surface, Body or OpenBody (in a .CATPart document).

  Note: you can apply different materials to different instances of a same CATPart.

- a Product (in a .CATProduct document)
- instances of a .model, .cgr, .CATPart (in a .CATProduct document).

Within a CATProduct, you should not apply different materials to different instances of a same Part because a material is part of the specific physical characteristics of a Part. Therefore, this could lead to inconsistencies.

Materials applied to .CATPart, .CATProduct and .cgr documents can now be saved in ENOVIAVPM.
For detailed information on ENOVIAVPM, refer to the *ENOVIAVPM User's Guide*.

Open the ApplyMaterial.CATProduct document.

Select the **View ->Render Style->Apply Customized View** command to make sure that the "Shading" and "Materials" options are on. The Custom View Modes dialog box is not displayed if it has been previously activated during the session.

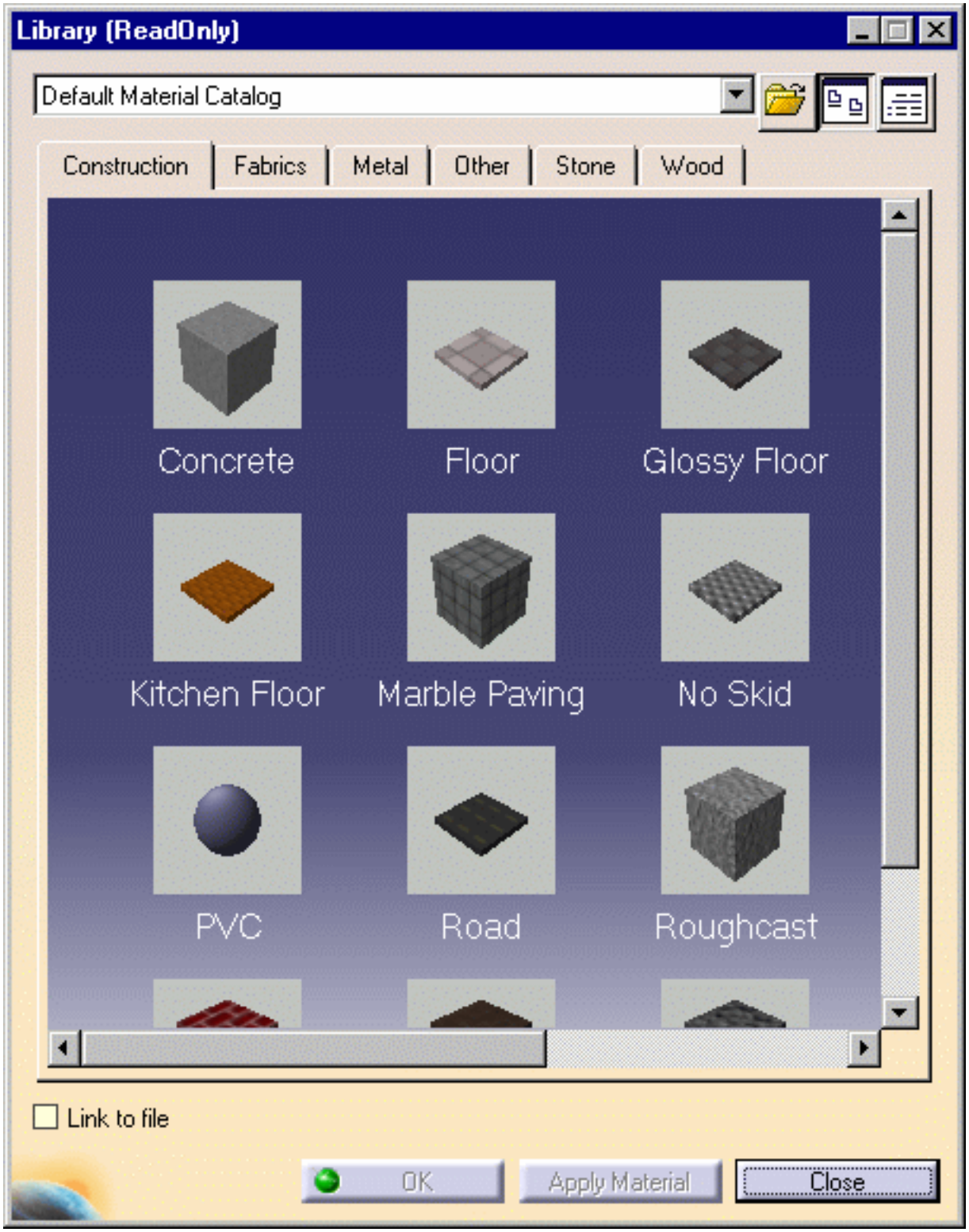


1. Select the element on which the material should be applied.

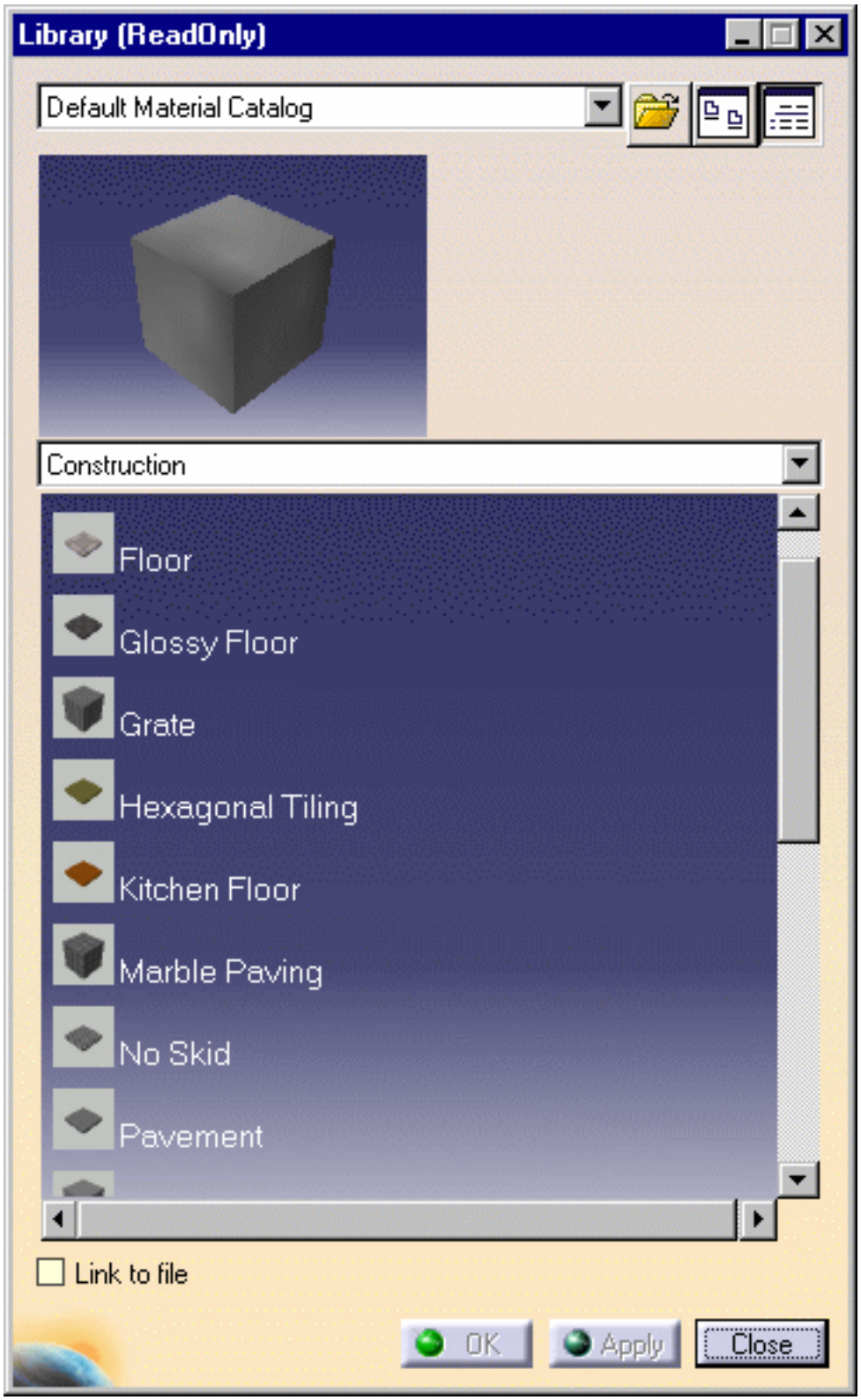
   Note: you can also apply a material simultaneously to several elements. To do so, simply select the desired elements (using either the pointer or the traps) before applying the material.

2. Click the **Apply Material** icon .

   The Library dialog box opens. It contains several pages of sample materials from which to choose. Each page is identified by a material family name on its tab (each material being identified by an icon) if you select the Display icons mode...


Library (ReadOnly)
Default Material Catalog
Construction
Fabrics
Metal
Other
Stone
Wood
Concrete
Floor
Glossy Floor
Kitchen Floor
Marble Paving
No Skid
PVC
Road
Roughcast
Link to file
OK
Apply Material
Close

...or each page is identified by a material family name in a pulldown list if you select the Display list mode:



Note that clicking the **Open a material library**  icon opens the File Selection dialog box which lets you navigate through the file tree to your own material libraries.
You can, of course, use the default library (see *What You Should Know Before You Start* in this guide) by choosing "Default Material Catalog".
The pulldown list will display the list of previously opened material libraries.
Note: when you reopen the dialog box, the last chosen material library will be placed on top of the list and used by default unless you select another one.

3. Select a material from any family, by a simple click.

   Once a material is selected, you can drag and drop or copy/paste it onto the desired element directly from the material library.

   You can also double-click a material or click it once then select the *Properties* contextual menu to display its properties for analysis purposes.

4. Click the Link to file checkbox if you want to map the selected material as a linked object and have it automatically updated to reflect any changes to the original material in the library.

   Two different icons (one with a white arrow  and one without ) identify linked and non-linked materials respectively in the specification tree.

   Note: You can edit linked materials. Doing so will modify the original material in the library. If you want to save changes made to the original material, use the **File->Save All** command.



When no object is selected in the specification tree, you can select the **Edit->Links...** command to identify the library containing the original material. You can then open this library in the Material Library workbench if desired.

You can also use the **Paste Special...** command to paste material as a linked object. You can copy both unlinked and linked materials. You can, for example, paste a linked material on a different element in the same document as well as on an element in a different document. For more information, see *Copying & Pasting Using Paste Special...* in this guide.

5. Click **Apply Material** to map the material onto the element.

The selected material is mapped onto the element and the specification tree is updated. In our example, the material was not mapped as a linked object.



A yellow symbol may be displayed to indicate the material inheritance mode. For more information, refer to *Setting Priority between Part and Product* in this guide.

Material specifications are managed in the specification tree: all mapped materials are identified. To edit materials (for more information, see Modifying Materials), simply right-click the material and select **Properties** from the contextual menu or double-click the material. You can also run searches to find a specific material in a large assembly (for more information, see *Finding Materials* in this guide) as well as use copy & paste or drag & drop capabilities.

Unless you select in the specification tree the desired location onto which the material should be mapped, dragging & dropping a material applies it onto the lowest hierarchical level (for instance, dragging and dropping onto a part will apply the material onto the body and not onto the part itself).

However, note that a material applied onto a body has no impact on the calculation of the part physical properties (mass, density, etc.) since only the physical properties of the part, and not those of the body, will be taken into account.

6. Click **OK** in the Library dialog box.

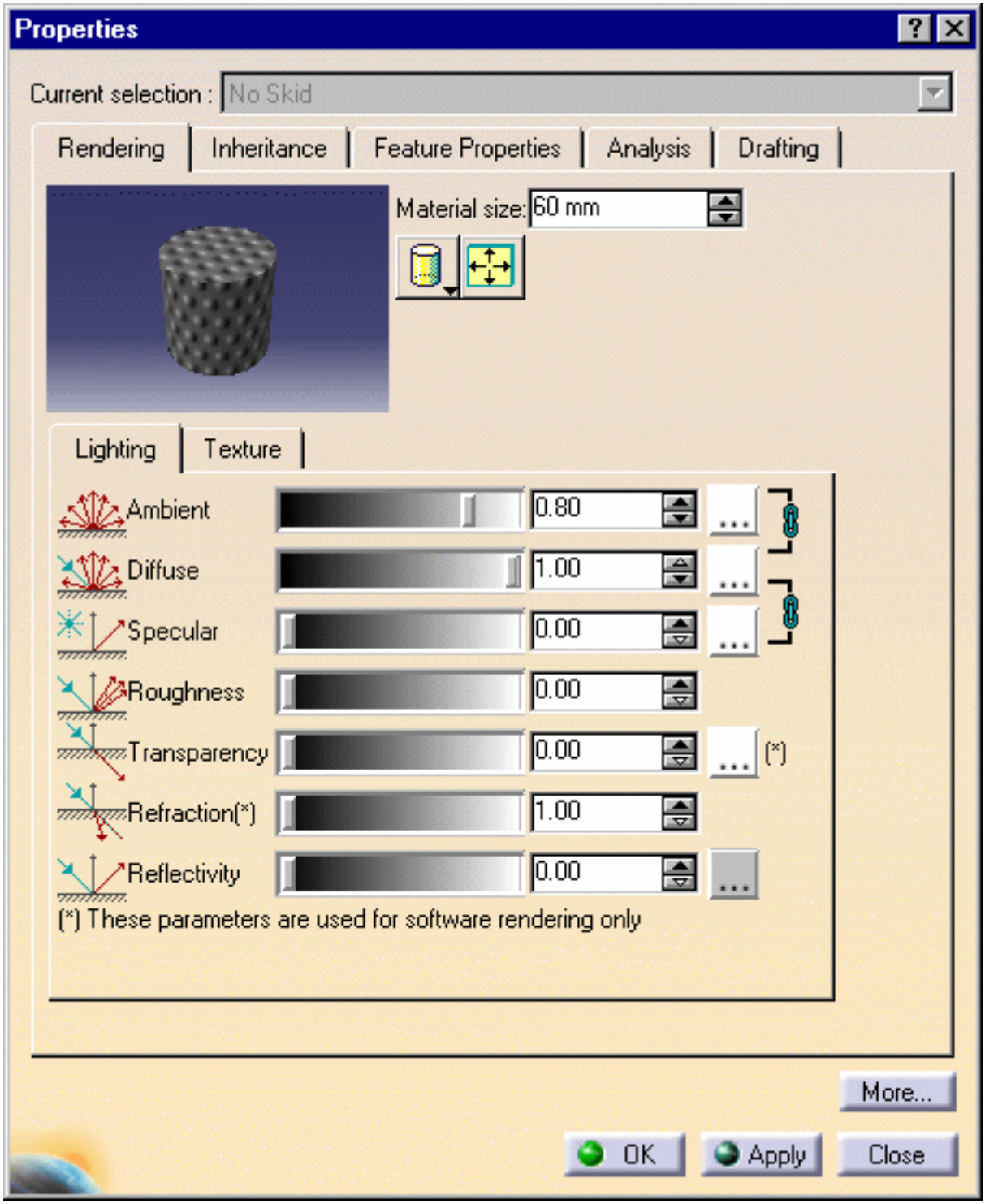
   The object looks the following way:



   Note: applying materials to elements affect the physical and mechanical properties, for example the density, of elements.

7. Right-click the material just mapped in the specification tree and choose the **Properties** item.

   The Properties dialog box is displayed:



8. Choose the Rendering tab to edit the rendering properties you applied on the element.

9. If necessary; change the material size to adjust the scale of the material relative to the element.

10. Click **OK** in the Properties dialog box, when you are satisfied with the material mapping on the element.

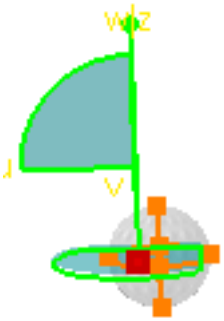
    Note: Appropriate licenses are required to use the Analysis and Drafting tabs.

If you are working in "Materials" visualization mode (i.e. Materials option is checked in the Custom View Modes panel) with no material applied to your object, this object will be visualized using default parameters which only take into account the color defined in the object graphic properties.
As a consequence, an object with no mapped material will appear as if made of matte plastic, non-transparent and without any relief.

11. Use the 3D compass to interactively position the material:

    Note that material positioning with the 3D compass is only possible in the Rendering, Product Structure, Part Design and DMU Navigator workbenches.

    - Select the material in the specification tree:

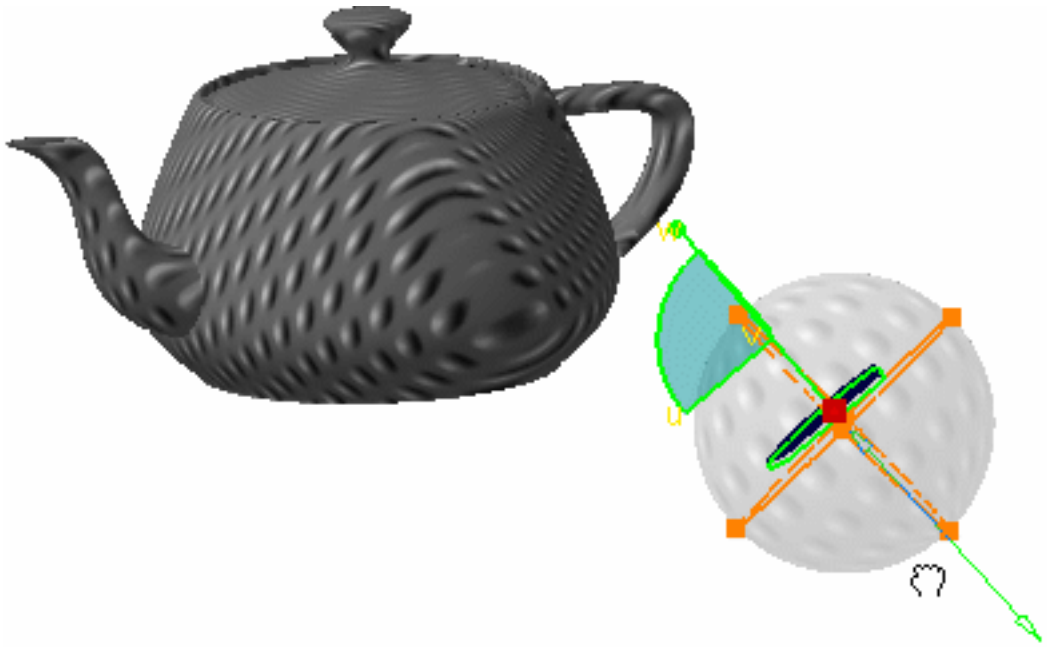


The compass is automatically snapped and the mapping support (in this case, a cylinder) appears, showing the texture in transparency.
If necessary, zoom in and out to visualize the mapping support which reflects the material size.

- Pan and rotate the material until satisfied with the result. You can:
  - Pan along the direction of any axis (x, y or z) of the compass (drag any compass axis)
  - Rotate in a plane (drag an arc on the compass)
  - Pan in a plane (drag a plane on the compass)
  - Rotate freely about a point on the compass (drag the free rotation handle at the top of the compass):



- Use the mapping support handles to stretch the material texture along u- and v- axes (as you can do it with the slider in the *Scale U, V* fields displayed in the Texture tab):



For more information on manipulating objects using the 3D compass, refer to the *Version 5 Infrastructure User's Guide*.



Advanced Tasks

The advanced tasks you will perform in the Wireframe and Surface Design workbench include managing higher level entities than single geometric elements, that is the PowerCopies.

Managing PowerCopies
Measure Tools

Managing Power Copies



Create PowerCopies: Select the **Insert -> Advanced Replication Tools -> PowerCopy Creation** command, select the elements making up the PowerCopy from the specification tree, define a name for the PowerCopy and its reference elements then choose an icon for identifying it.



Instantiate PowerCopies: Select the **Insert -> Instantiate From Document** command, select the document or catalog containing the powercopy, complete the **Inputs** within the dialog box selecting adequate elements in the geometric area.

Up Next

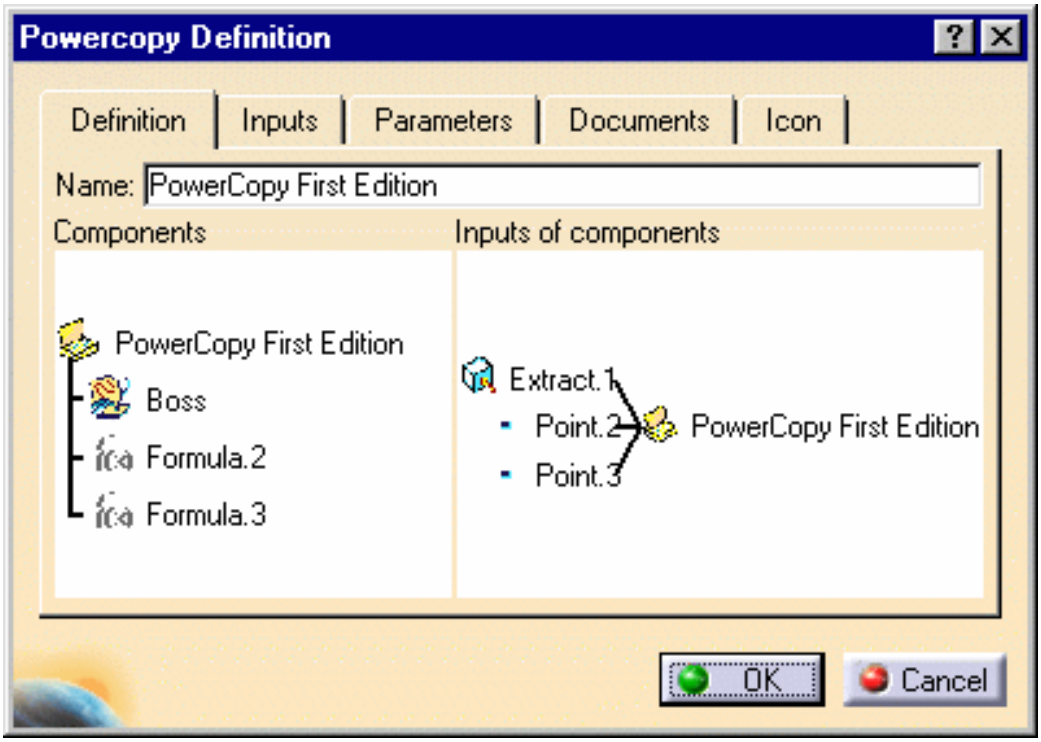
Creating Power Copies

This task shows how to use create PowerCopy elements, to be reused later.
A PowerCopy is a set of features (geometric elements, formulas, constraints 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 PowerCopy captures the design intent and know-how of the designer thus enabling greater reusability and efficiency.

Open the PowerCopyStart1.CATPart document.

1. Click the **PowerCopy Creation** icon, or select the **Insert ->Advanced Replication Tools -> PowerCopy Creation** menu item.
   The **PowerCopy Definition** dialog box is displayed.
2. Select, from the specification tree, the elements to be included in the PowerCopy.

   The **PowerCopy Definition** dialog box is automatically filled with information about the selected elements.

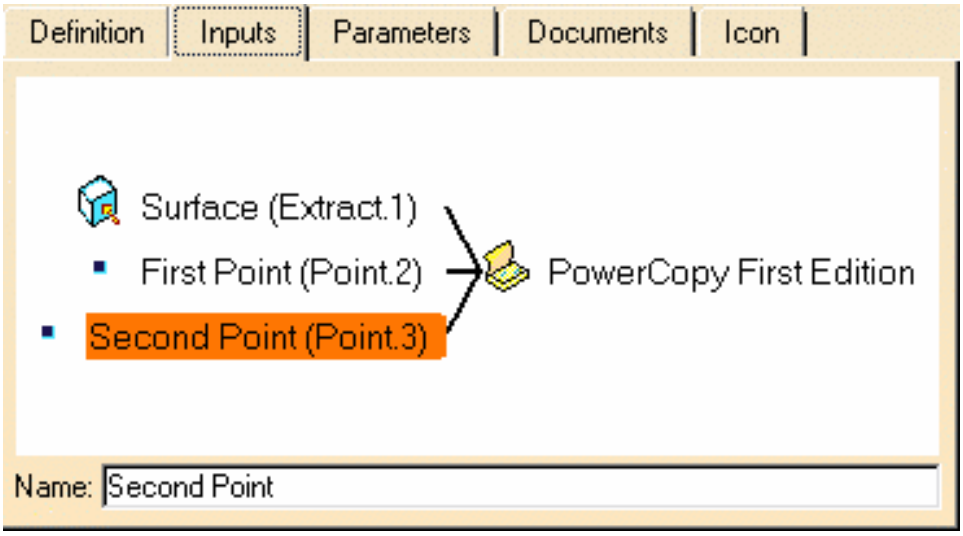


3. Define the PowerCopy as you wish to create it:

   The **Definition** tab lets you assign a name to the PowerCopy and presents its components in the 3D viewer.

The **Inputs** tab lets you rename the reference elements making up the PowerCopy.

You can do that for clarification purposes as to their roles, by selecting the elements in the viewer and entering a new name in the **Name** field.
In this example, we renamed all three elements and in brackets you still can read the elements' default name based on their type.

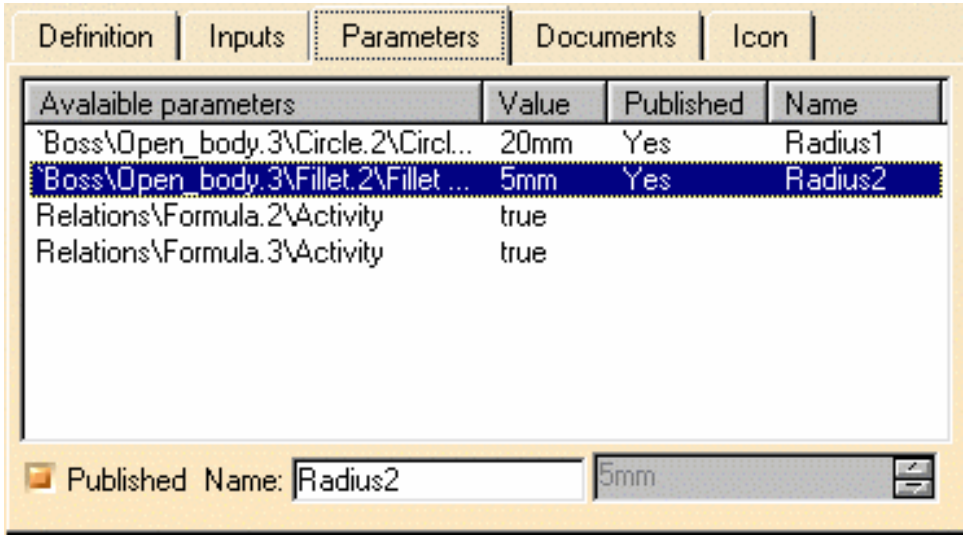


The **Parameters** tab lets you define which of the parameter values used in the PowerCopy you will be able to modify at instantiation

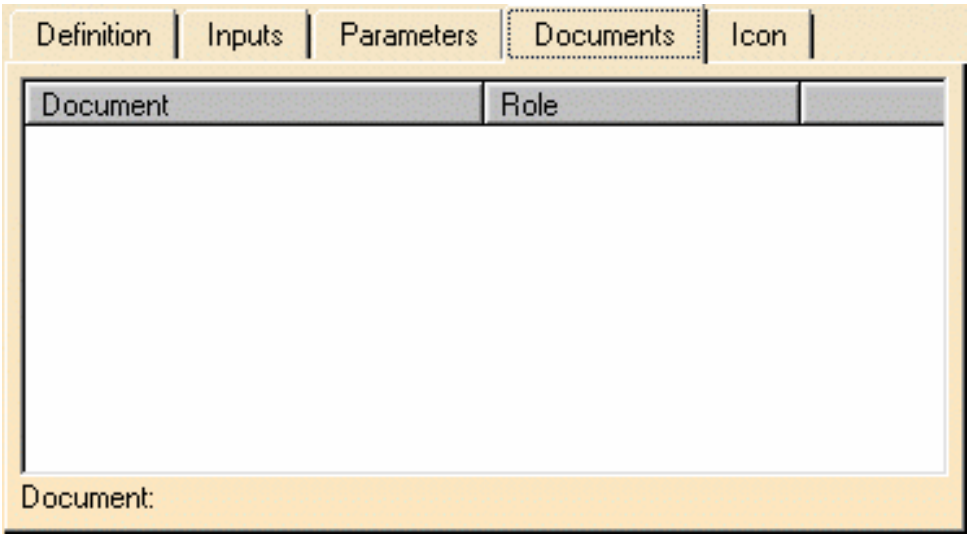
time.

Simply check the Published button.

Use the **Name** field to give a more explicit name to the element.

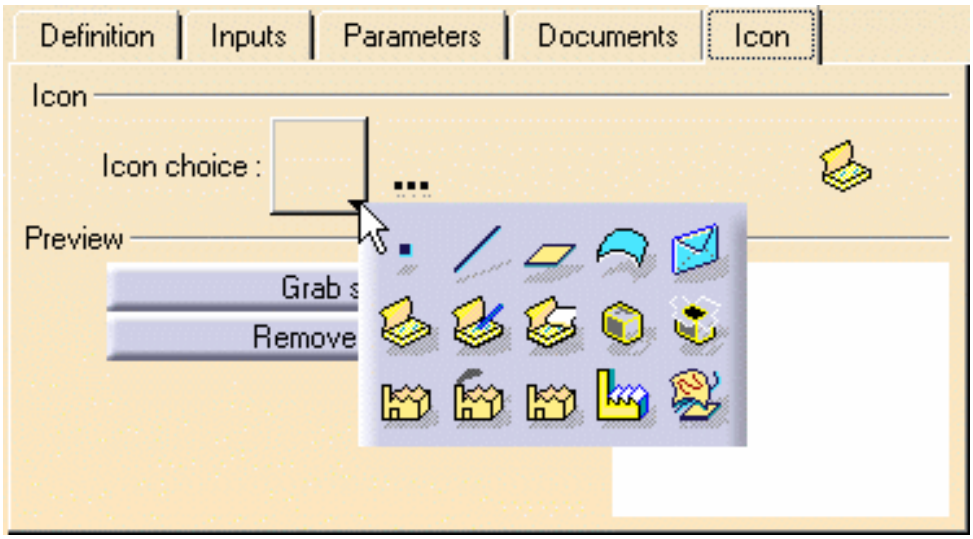


The Documents tab shows the complete path and role of Design tables that are referenced by an element included in the Power Copy.

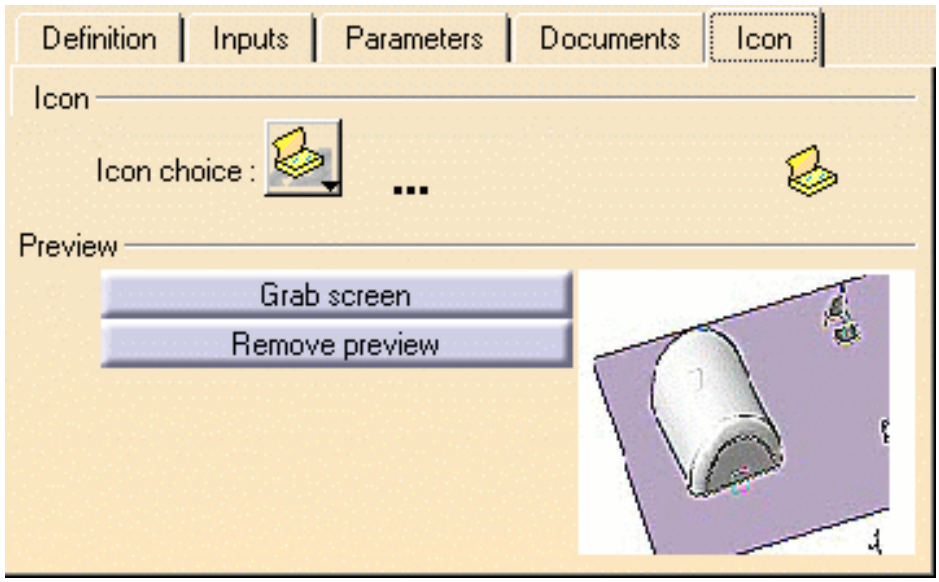


The **Icon** tab lets you modify the icon identifying the PowerCopy in the specifications tree. A subset of icons is available from the Icon choice button.
If you click **...** the Icon Browser

opens, giving you access to all the graphic icons installed with the CATIA software.



Use the **Grab screen** button to capture an image of the PowerCopy to be stored with its definition in the catalog.



Use the **Remove preview** button to delete the image captured with the **Grab screen** button.

4. Click OK to create the PowerCopy.

The PowerCopy is displayed close to the top of the specification tree.

- Double-click the PowerCopy in the specification tree to display the **PowerCopy Definition** dialog box and edit its contents.
- A formula is automatically included in a Power Copy definition when all its parameters are included.
  Otherwise, i.e. if at least one parameter is not selected as part of the Power Copy, you have to manually select the formula to make it part of the definition. If you do so, all the formula's parameters that have not been explicitly selected, are considered as inputs of the Power Copy.

Up Next

Instantiating Power Copies

This task shows how to instantiate PowerCopies once they have been created as described in Creating PowerCopies. There are two ways to do this:

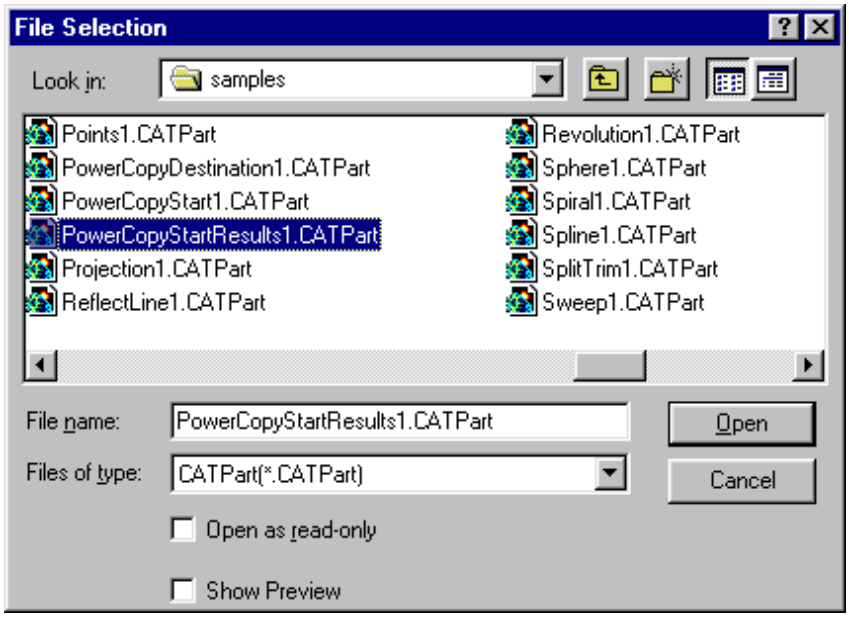
1. using the PowerCopy Instantiation menu item
2. using a catalog

Furthermore, the use of the Replace viewer, regardless of the instantiation type, is detailed.

Open the PowerCopyDestination1.CATPart document.

Using the menu item:

1. Click the **PowerCopy Instantiation** icon or select the **Insert -> Instantiate From Document** menu item.
   The **File Selection** dialog box is displayed allowing you to navigate to the document or catalog where the power copy is stored.

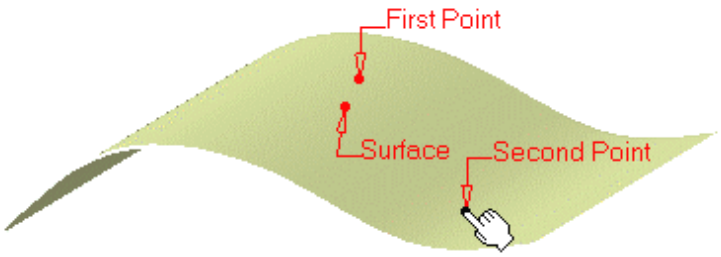


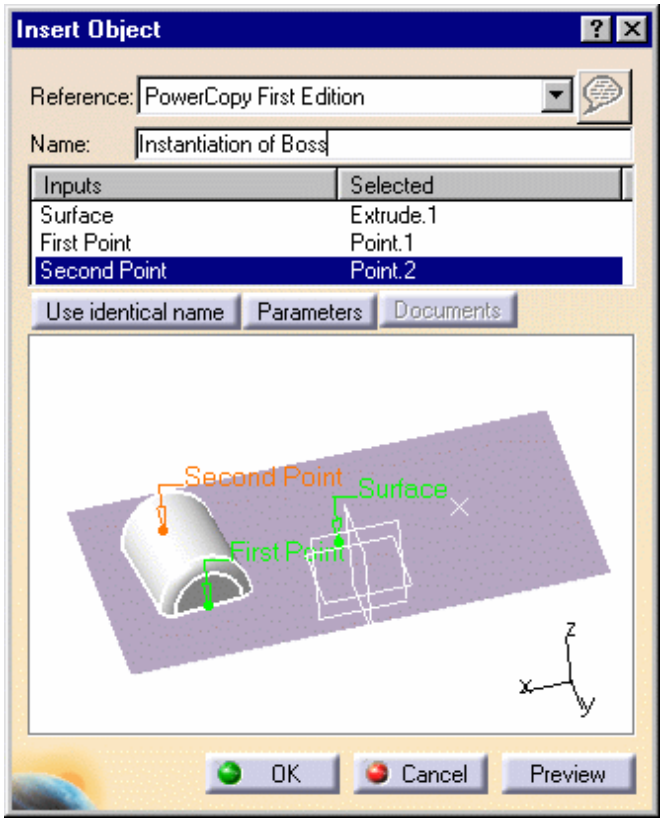
2. Select the document containing the Powercopy, and click Open.
   Here we selected the PowerCopyStartResults1.CATPart document.

   The **Insert Object** dialog box is displayed.

   Use the **Reference** list to choose the correct PowerCopy when several have been defined in the document.

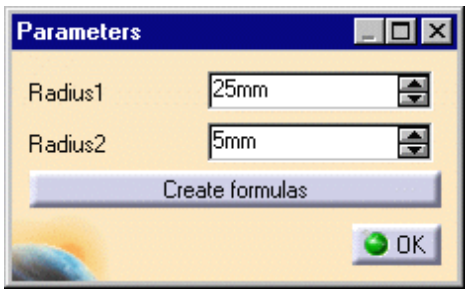
3. Complete the **Inputs** within the dialog box by selecting the adequate element in the geometric area.





4. If needed, click on the **Use identical name** button to automatically select all the elements with the same name. This is especially useful when the input is the same one repeated several time.

5. You can also click on the **Parameters** button to display the **Parameters** dialog box and modify values.
   Here we increased the **Radius1** value to 25 mm.
6. Use the Create formulas button to automatically create a formula on every parameters with the same name provided there are any.
7. Click OK.



The Documents button lets you access the list of documents (such as design tables) pointed by one of the elements making up the Power copy.
If there are documents, the Documents dialog box opens and you can click the Replace button to display the File Selection dialog box and navigate to a new design table to replace the initial one.
When no document is referenced, the Documents button is grayed within the Insert Object dialog box.

8. Click OK to create the PowerCopy instance.

   The PowerCopy is instantiated in context, meaning its limits are automatically re-defined taking into account the elements on which it is instantiated.



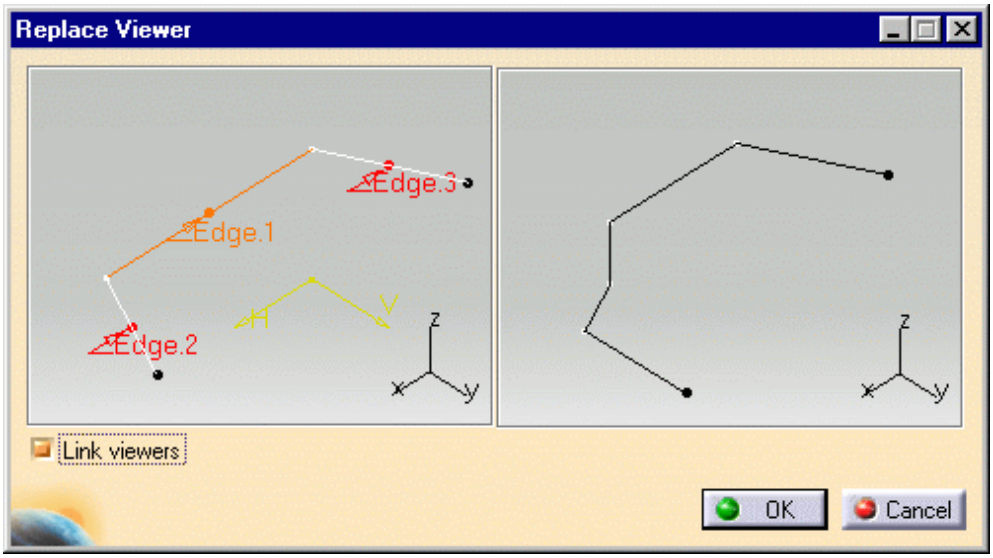
- When instantiating from the same document, use the **PowerCopy object -> Instantiate** contextual menu to display the Insert Object dialog box directly.
- The  icon is always grayed when instantiating Power Copies. It is available with User Features and allows you to create and modify URLs.

Using the Replace Viewer

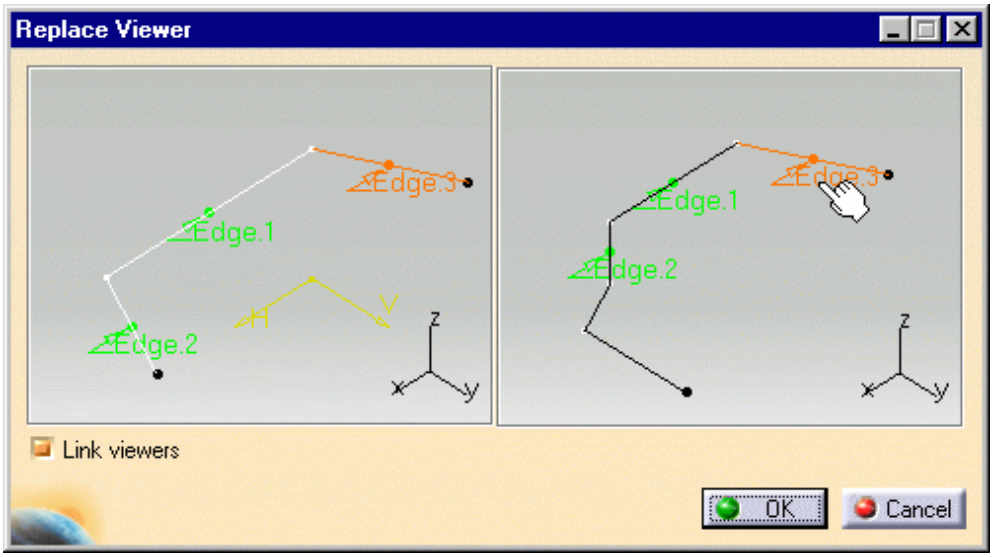
In some cases, when instantiating a powercopy, the replacing element does not present the same sub-elements as the replaced element. Therefore you need to clearly indicate in a specific dialog box, the Replace Viewer, how to rebuild the geometry from the replacing element. In the following example, the replacing sketch does not have the same number of vertices as the initial sketch, and you are prompted to indicate on what edge the filleted surfaces are to be created.

Open the PowerCopyReplace1.CATPart document.

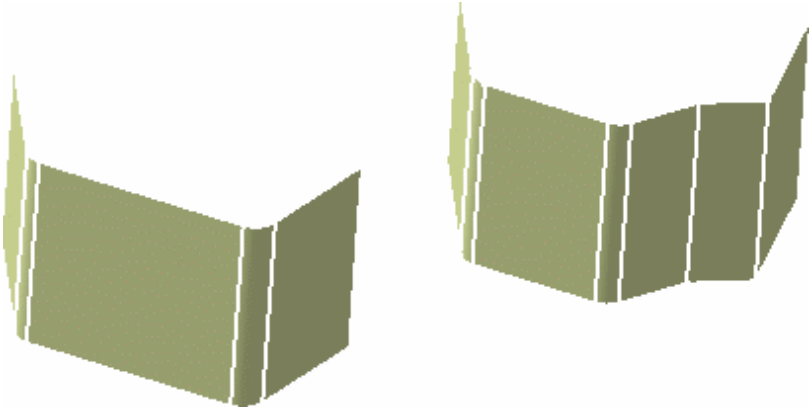
1. Expand the PowerCopy entry in the specification tree, right-click the PowerCopy.1 feature, and choose **PowerCopy.1 object -> Instantiate** command.
2. Select Sketch.2 to replace Sketch.1.
   The Replace Viewer is displayed, showing to the left the initial sketch and the edges selected to create the two fillets in the initial geometry, and to the right the replacing sketch on which you are prompted to specify edges.



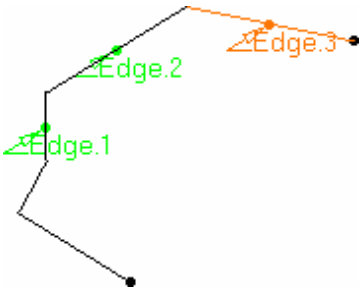
3. Select the edges on the replacing sketch.



4. Click OK in the Replace Viewer.
5. Select the XY plane, or click the Use Identical Name to select it as the needed plane.
6. Click OK in the Insert Object dialog box.
   The PowerCopy is instantiated and the filleted surfaces are computed as per the selection in the Replace Viewer.



Make sure to select the edges as proposed in the Replace Viewer. For example, you cannot invert Edge.1 and Edge.2 if Edge.3 remains where specified in the example above. Otherwise, the system will not be able to re-build the geometry based on these specifications, and the Update Diagnosis dialog box will be displayed prompting you to edit the geometry.



A new panel now allows you to select alternate document access methods.
See Opening Existing Documents Using the Browse Panel in *CATIA Infrastructure User Guide*.

Back Up

Measure Tools

You can create a link between a measure and a parameter (length or angle) using two methods:



Measure distances and angles: Right-click the appropriate field, select **Measure Between**, set the measure type and mode, then select two entities.



Measure properties: Right-click the appropriate field, select **Measure Item**, set the measure type and mode, then select an item.

Back Up

Measuring Distances & Angles between Geometrical Entities

This task explains how to measure minimum or maximum distances and, if applicable, angles between geometrical entities (points, surfaces, edges, vertices and entire products).

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysa\samples.

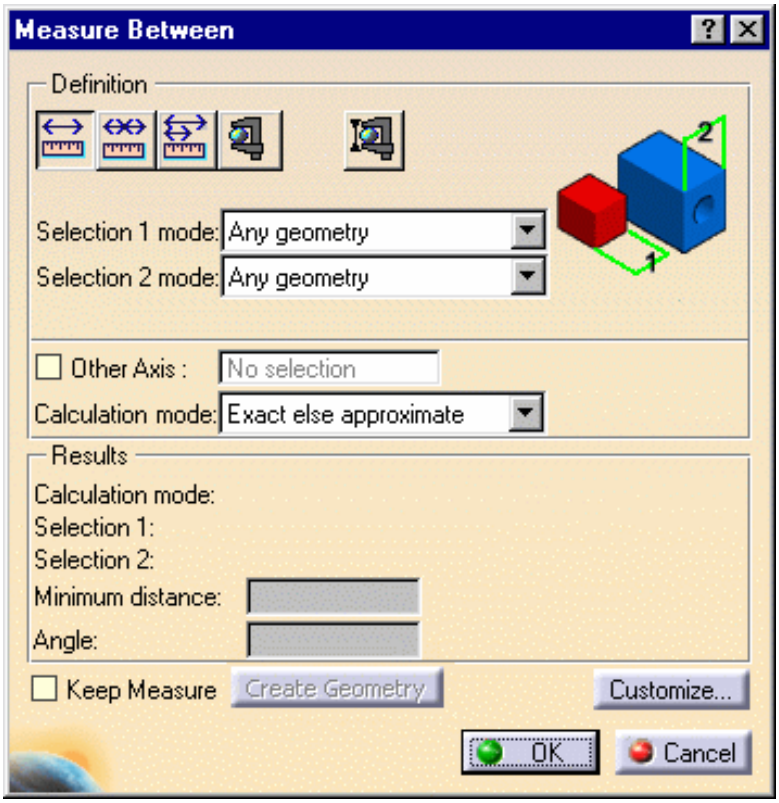
Restriction: Neither Visualization Mode nor cgr files permit selection of individual vertices.

Note: In the No Show space, this command is not accessible.

1. Click the Measure Between icon.

   In DMU, you can also select **Analyze-> Measure Between** from the menu bar.

   The Measure Between dialog box appears.



By default, minimum distances and if applicable, angles are measured.

By default, measures made on active products are done with respect to the product axis system. Measures made on active parts are done with respect to the part axis system.
Note: This distinction is not valid for measures made prior to Version 5 Release 8 Service Pack 1 where all measures are made with respect to the absolute axis system.

You can also measure distances and angles with respect to a local V5 axis system.

A Keep Measure option in the dialog box lets you keep the current and subsequent measures as features. This is useful if you want to keep the measures as annotations for example.
Note: This option is not available in the Drafting workbench.

Some measures kept as features are associative and can be used to valuate parameters or in formulas.

In the Drafting workbench, measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

The Measure Item command  is accessible from the Measure Between dialog box.

In DMU, the Measure Thickness command is also accessible from the Measure Between dialog box. For more information, see the *DMU Space Analysis User's Guide*.

P1-Only Functionality

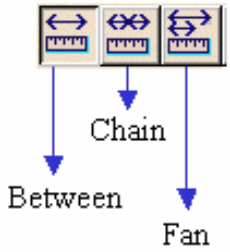
In P1, the Measure Tools toolbar appears.
This toolbar has two icons:

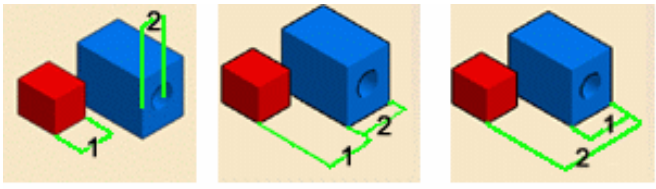


- Measure Dialogs : lets you show or hide the associated dialog box.
- Exit Measure : lets you exit the measure. This is useful when the dialog box is hidden.

2. Select the desired measure type.

Notice that the image in the dialog box changes depending on the measure type selected.





Defining Measure Types

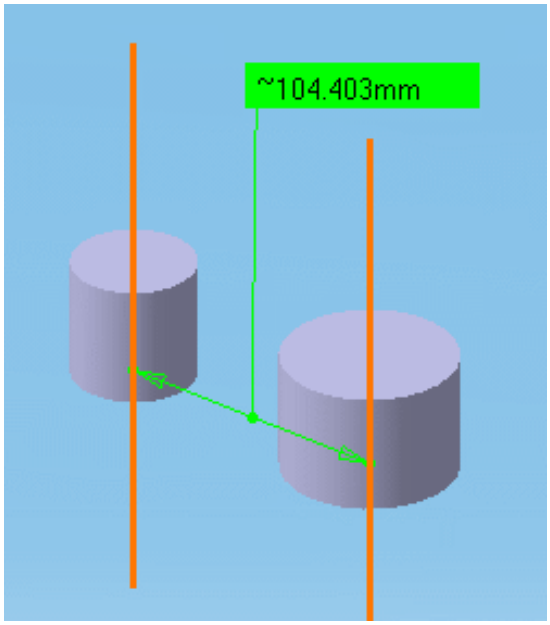
- Between (default type): measures distance and, if applicable, angle between selected items.
- Chain: lets you chain measures with the last selected item becoming the first selection in the next measure.
- Fan: fixes the first selection as the reference so that you always measure from this item.

3. Set the desired mode in the Selection 1 and Selection 2 mode drop-down list boxes.

Defining Selection 1 & Selection 2 Modes

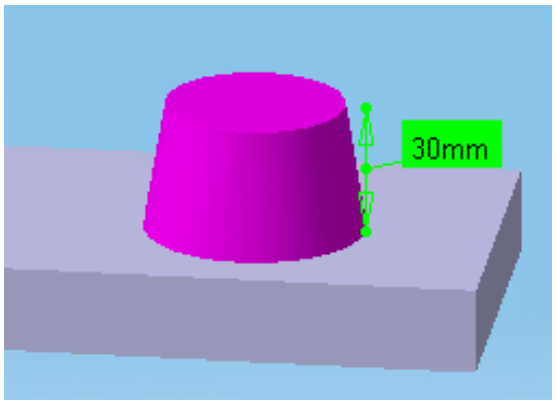
- Any geometry (default mode): measures distances and, if applicable, angles between defined geometrical entities (points, edges, surfaces, etc.).
  Note: The Arc center mode is activated in this selection mode.

  This mode recognizes the axis of cylinders and lets you measure the distance between two cylinder axes for example.

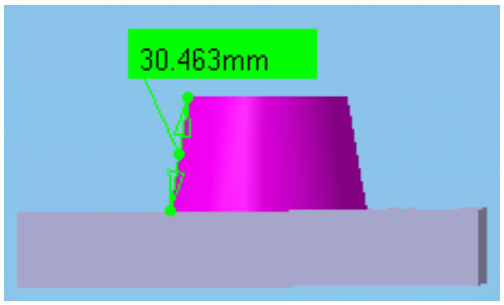


- Any geometry, infinite: measures distances and, if applicable, angles between the infinite geometry (plane or line) on which the selected geometrical entities lie. The Arc center mode is activated and this mode also recognizes cylinder axes. For all other selections, the measure mode is the same as any geometry.

  Any geometry, infinite

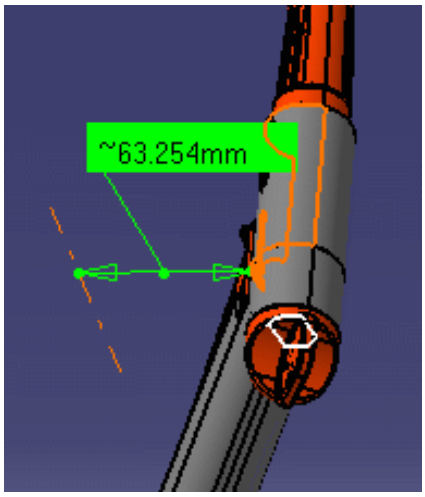


  Any geometry



- Picking point: measures distances between points selected on defined geometrical entities. Always gives an approximate measure.
- Point only: measures distances between points. Dynamic highlighting is limited to points.
- Edge only, Surface only: measures distances and, if applicable, angles between edges and surfaces respectively. Dynamic highlighting is limited to edges or surfaces and is thus simplified compared to the Any geometry mode. All types of edge are supported.
- Picking axis: measures distances and, if applicable, angles between an entity and an infinite line perpendicular to the screen.

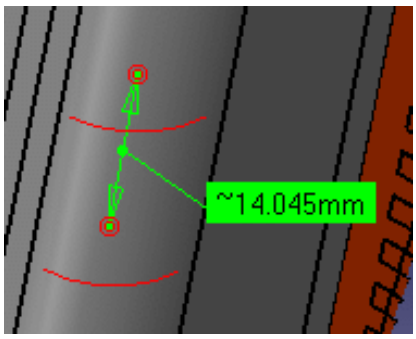
Simply click to create infinite line perpendicular to the screen.



- Intersection: measures distances between intersection points between two edges or an edge and a surface. In this case, two selections are necessary to define selection 1 and selection 2 items.
- Edge limits: measures distances between endpoints or midpoints of edges. Endpoints only are proposed on curved surfaces.
- Arc center: measures distances between the centers of arcs.
- Center of 3 points arc: measures distances between the centers of arcs defined by 3 points.

To define arc center, click three points on the geometry.

Note: The resulting measure will always be approximate.



- Coordinate: measures distances between coordinates entered for selection 1 and/or selection 2 items.

4. Set the desired calculation mode in the Calculation mode drop-down list box.



Defining the Calculation Mode

- Exact else approximate (default mode): measures access exact data and wherever possible true values are given. If exact values cannot be measured, approximate values are given (identified by a ~ sign).
- Exact: measures access exact data and true values are given. Note that you can only select exact items in the geometry area or specification tree.
  In certain cases, in particular if products are selected, a warning dialog box informs you that the exact measure could not be made.
- Approximate: measures are made on tessellated objects and approximate values are given (identified by a ~ sign).

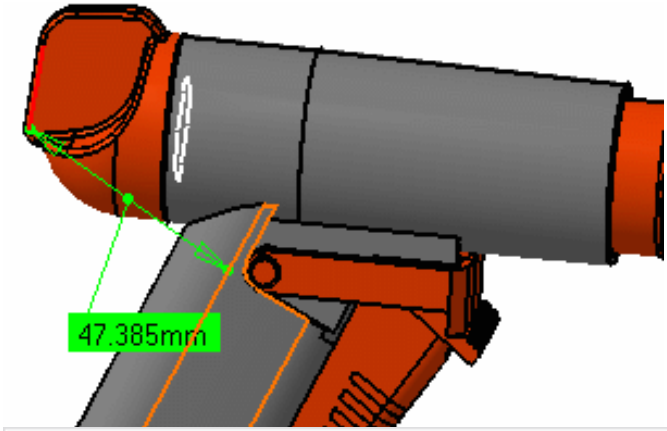
Note: You can hide the display of the ~ sign using the **Tools -> Options** command (**General -> Parameters and Measure -> Measure Tools**).

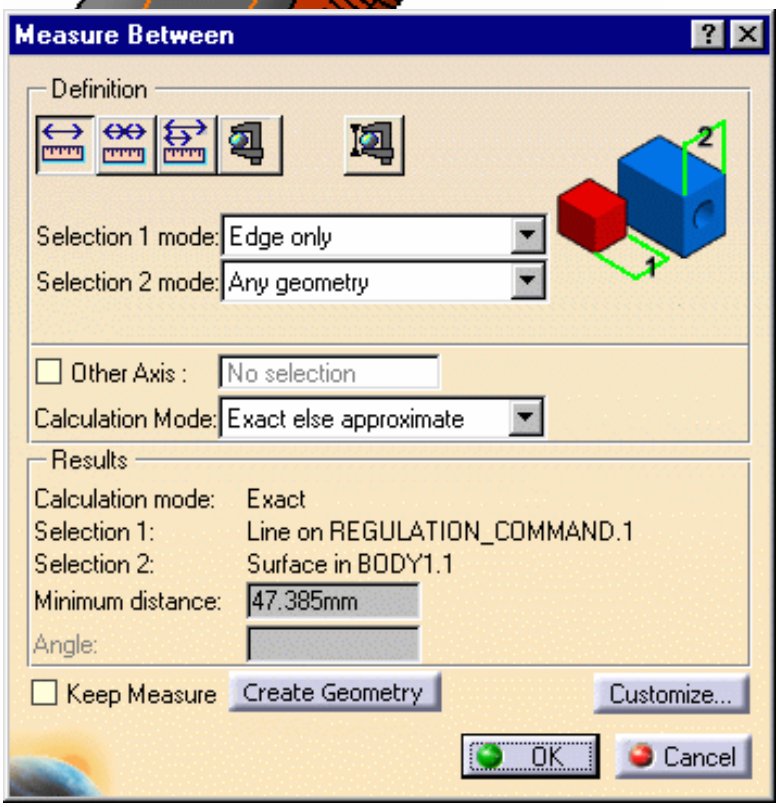
5. Click to select a surface, edge or vertex, or an entire product (selection 1).
Notes:

- The appearance of the cursor has changed to assist you.
- Dynamic highlighting of surfaces, points and vertices, etc. helps you locate items to click on.

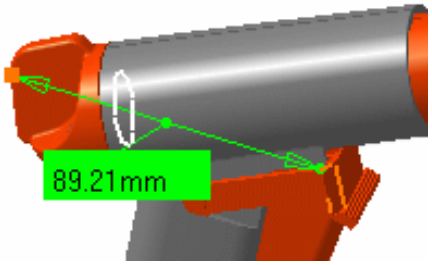
6. Click to select another surface, edge or vertex, or an entire product (selection 2).

   A line representing the minimum distance vector is drawn between the selected items in the geometry area. Appropriate distance values are displayed in the dialog box.

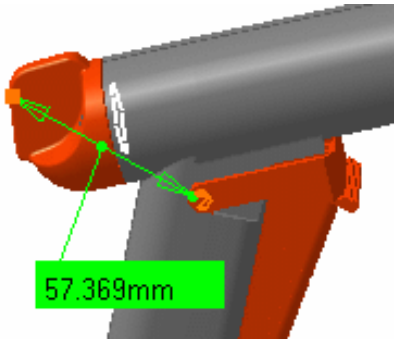




   By default, the overall minimum distance and angle, if any, between the selected items are given in the Measure Between dialog box.
   The number of decimal places, the display of trailing zeros and limits for exponential notation is controlled by the Units tab in the Options dialog box (**Tools ->Options**, **General ->Parameters and Measure**). For more information, see the *Infrastructure User's Guide*.
7. Select another selection and, if desired, selection mode.
8. Set the Measure type to Fan to fix the first selection so that you can always measure from this item.
9. Select the second item.



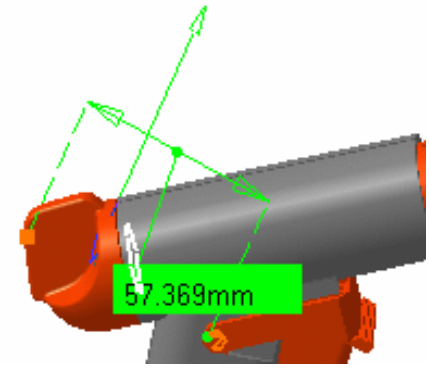
10. Select another item.

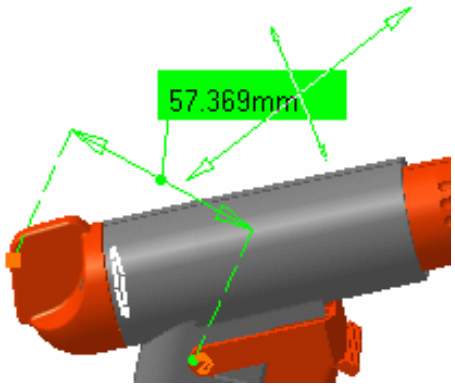


Using the Other Selection... command in the contextual menu, you can access the center of spheres.

11. If necessary, adjust the presentation of the measure:

You can move the lines and text of the measure.

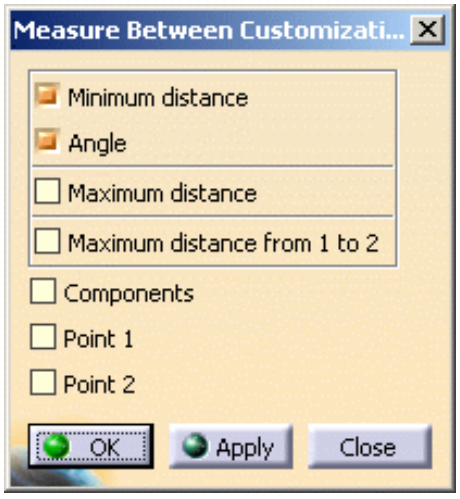




The Properties command (Graphics tab) lets you change the fill color and transparency as well as the color, linetype and thickness of measure lines.

Note: You cannot vary transparency properties, the current object is either the selected color or transparent.

Customizing Your Measure



You can, at any time, customize the display of the results in both the geometry area and the dialog box. To do so, click Customize... in the Measure Between dialog box and set your display in the Measure Between Customization dialog box.

Note: Measuring minimum distance, maxium distance and maximum distance from 1 to 2 are mutually exclusive options. Each time you change option, you must make your measure again.

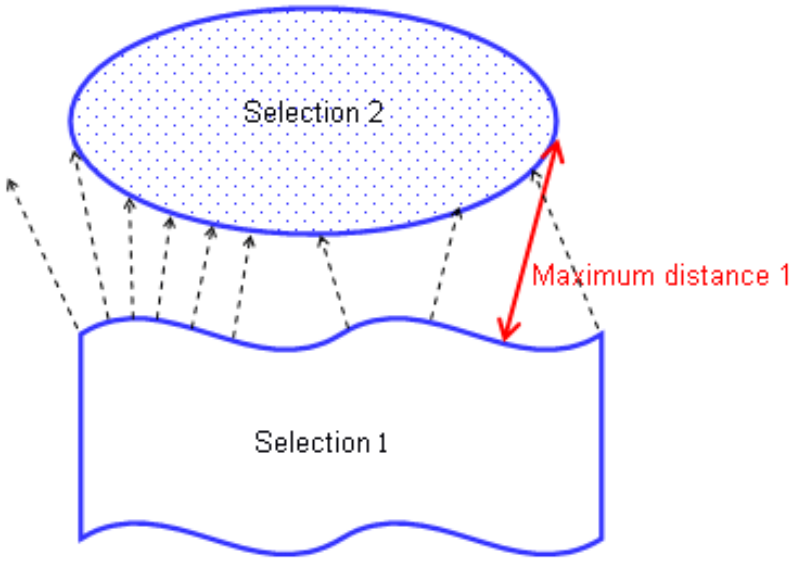
Point 1 and point 2 give the coordinates of the two points between which the distance is measured.

Measuring Maximum Distance

You can also measure the maximum distance between two surfaces, two volumes or a surface and a volume.

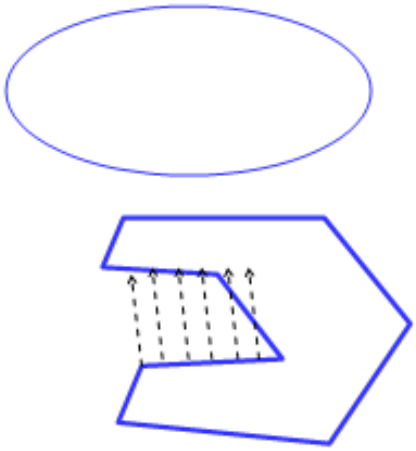
Distance is measured normal to the selection and is always approximate. Two choices are available:

- Maximum distance from 1 to 2: gives the maximum distance of all distances measured from selection 1.
  Note: This distance is, in general, not symmetrical.

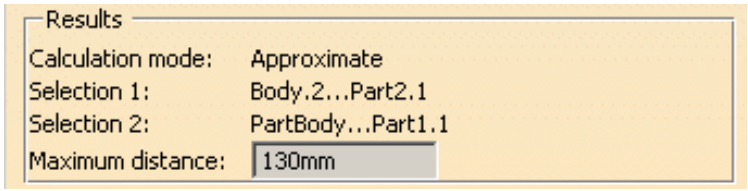


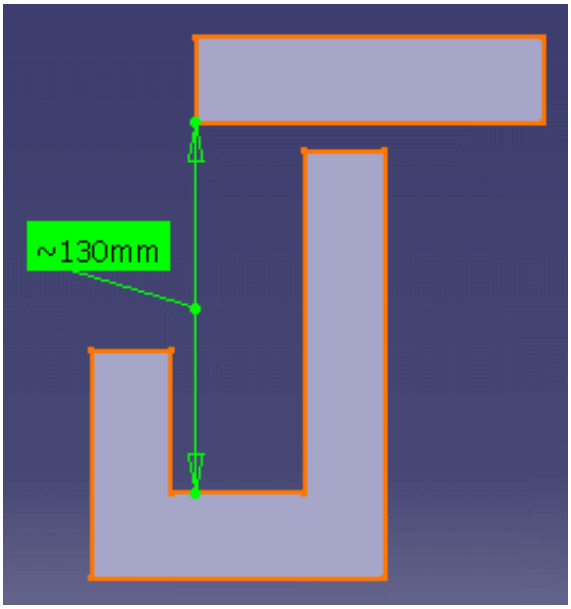
- Maximum distance: gives the highest maximum distance between the maximum distance measured from selection 1 and the maximum distance measured from selection 2.

Note: All selection 1 (or 2) normals intersecting selection 1 (or 2) are ignored.



12. Click Customize... and check the appropriate maximum distance option in the Measure Between Customization dialog box, then click **OK**.
13. Make your measure.

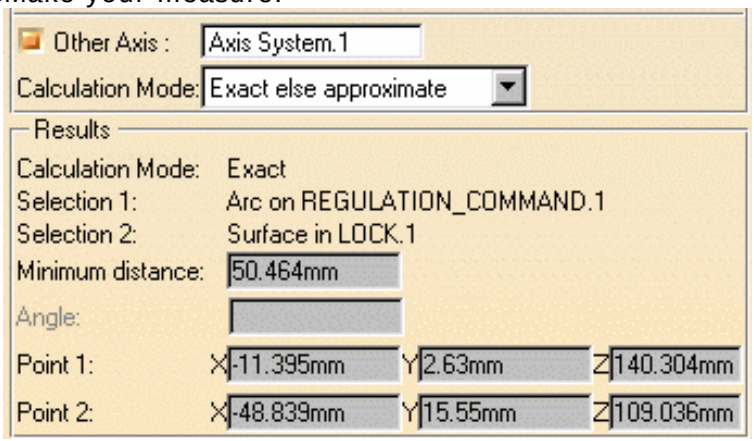




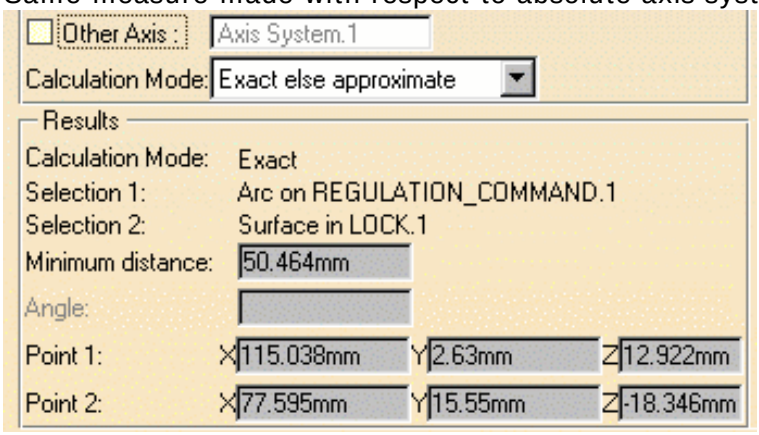
Measuring in a Local Axis System

For this part of the task, you will need a V5 axis system.

14. Click Customize... and check Minimum distance, Point 1 and Point 2 options in the Measure Between Customization dialog box, then click **OK**.
15. Check the Other Axis option in the dialog box.
16. Select a V5 axis system in the specification tree or geometry area.
17. Make your measure.

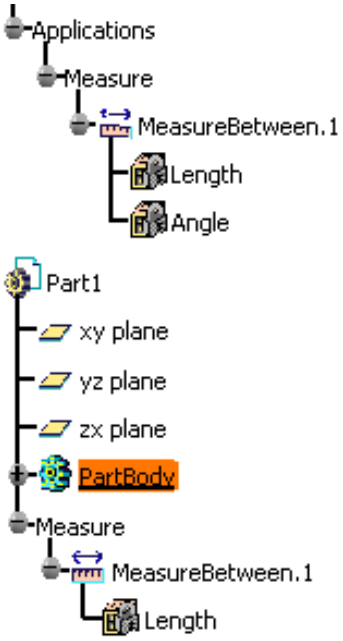


Same measure made with respect to absolute axis system:



- All subsequent measures are made with respect to the selected axis system. To change the axis system, click the Other Axis field and select another axis system. To return to the absolute axis system, uncheck the Other Axis option.
- This type of measure is associative: if you move the axis system, the measure is impacted and can be updated.

18. Click **OK** when done.



If you checked the Keep Measure option in the Measure Between dialog box, your measures are kept as features and your specification tree will look something like this if measures were made on the active product.
Or like this, if measures were made on the active part.

Note: If the product is active, any measures on parts are placed in No Show.

Some measures kept as features are associative. In Design Mode, if you modify a part or move a part in a product structure context and the measure is impacted, it will be identified as not up-to-date in the specification tree. You can then update it locally have it updated automatically.

When measures are used to valuate parameters, an associative link between the measure and parameter is created. Measures can also be used in formulas.

Editing Measures

In addition to editing the presentation of the measure, you can also edit the measure itself and change one of the selections on which it was based. This is particularly useful in design mode where you no longer have to redo your measure.
You can also change selections that no longer exist because they were deleted.

- Double-click the measure in the specification tree or geometry area.
- Make new selections.

  Notes:
  You can change selection modes when making new selections.
  For invalid measures where one selection has been deleted, you only have to replace the deleted selection.
  For all other measures, repeat all selections.

- Click **OK** when done.

Creating Geometry

A Create Geometry option in the dialog box lets you create the points and line corresponding to the minimum distance result in a part.

Sectioning Measure Between Results

Having made and kept your measure, select it then click the Sectioning icon to section measure results. The plane is created parallel to the direction defined by the measure and sections entities selected for the measure only. All section plane manipulations are available.

Note: You may need an appropriate license to access the Sectioning command.

Measuring Properties

This task explains how to measure the properties associated to a selected item (points, edges, surfaces and entire products).

This command lets you choose the selection mode, the calculation mode and axis system when measuring properties.

Note: In the No Show space, this command is not accessible.

Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model.

They are to be found in the online documentation filetree in the common functionalities sample folder cfysa\samples.

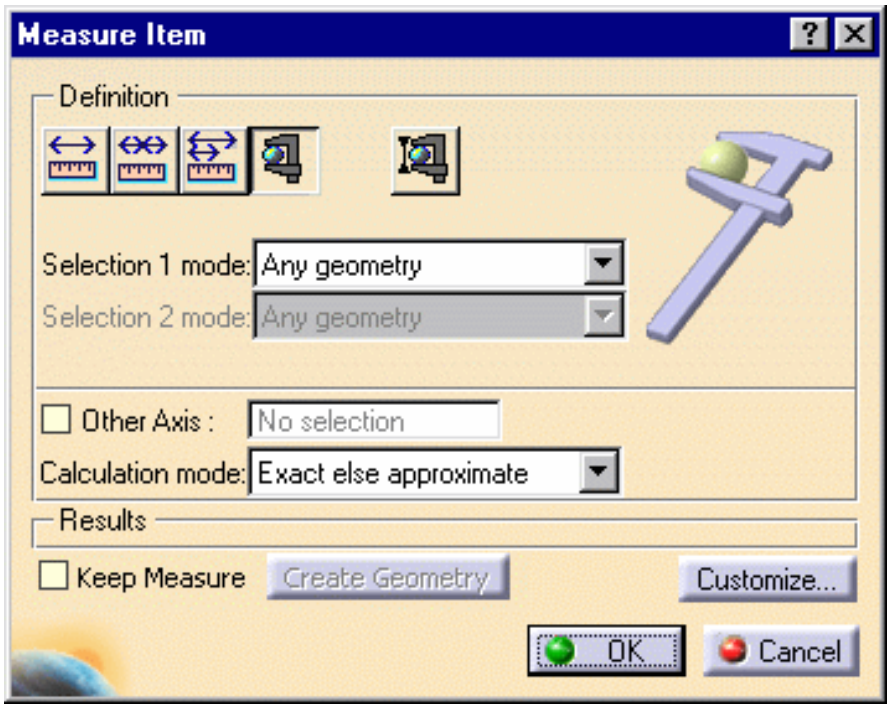
Restriction: Neither Visualization Mode nor cgr files permit selection of individual vertices.

1. Switch to Design Mode (**Edit ->Representations ->Design Mode**).
2. Set **View -> Render Style** to Shading with Edges.

   Note: You cannot use this command, if Shading only is selected.
3. Click the Measure Item  icon.

   In DMU, you can also select **Analyze -> Measure Item** from the menu bar.

   The Measure Item dialog box appears.



By default, properties of active products are measured with respect to the product axis system. Properties of active parts are measured with respect to the part axis system.
Note: This distinction is not valid for measures made prior to Version 5 Release 8 Service Pack 1 where all measures are made with respect to the absolute axis system.

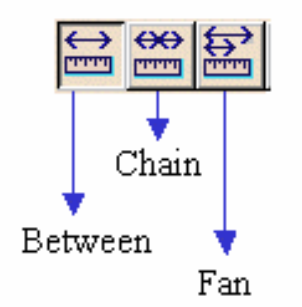
You can also measure properties with respect to a local V5 axis system.

The Keep Measure option lets you keep current and subsequent measures as features. This is useful if you want to keep measures as annotations for example.
Note: This option is not available in the Drafting workbench.

Some measures kept as features are associative and can be used to valuate parameters or in formulas.

In the Drafting workbench, measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

The Measure Between command is accessible from the Measure Item dialog box. Simply click one of the Measure Between icons in the Definition box to switch commands.



In DMU, the Measure Thickness command is also accessible from the Measure Item dialog box. For more information, see the appropriate task in the *DMU Space Analysis User's Guide*.

P1-Only Functionality

In P1, the Measure Tools toolbar appears.
This toolbar has two icons:



- Measure Dialogs : lets you show or hide the associated dialog box.
- Exit Measure : lets you exit the measure. This is useful when the dialog box is hidden.

4. Set the desired measure mode in the Selection 1 mode drop-down list box.

Defining the Selection 1 Mode

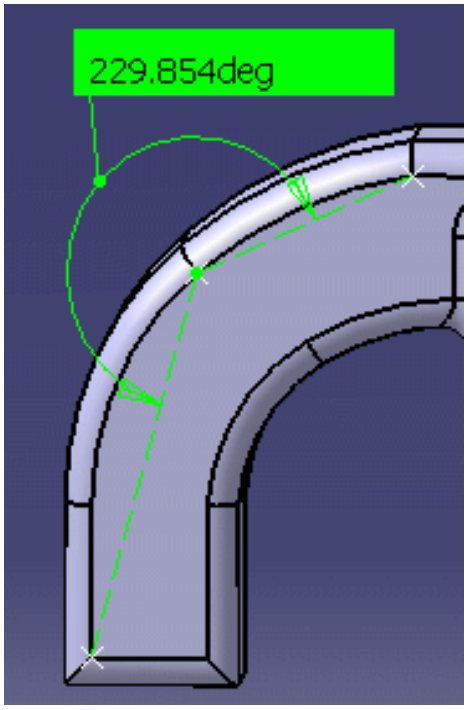
- Any geometry (default mode): measures the properties of the selected item (point, edge, surface or entire product).
- Point only: measures the properties of points. Dynamic highlighting is limited to points.
- Edge only: measures the properties of edges. All types of edge are supported.
- Surface only: measures the properties of surfaces.

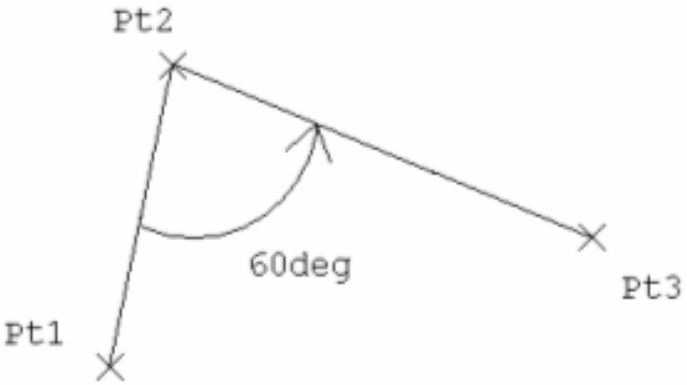
  In the last three modes, dynamic highlighting is limited to points, edges or surfaces depending on the mode selected, and is thus simplified compared to the Any geometry mode.

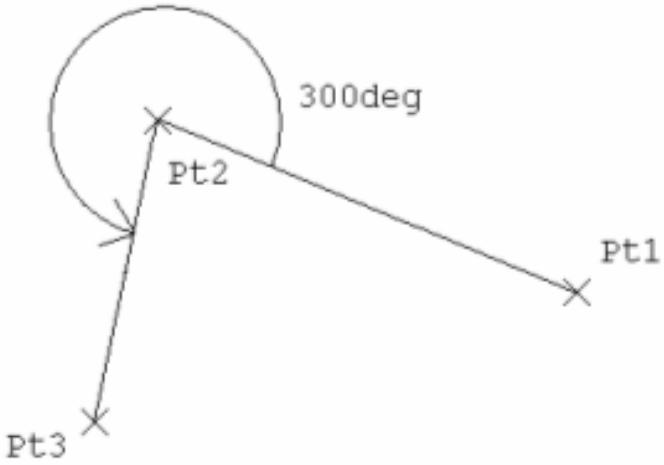
- Angle by 3 points: measures the angle between two lines themselves defined by three points.

  To define lines, select three existing points in the geometry area or in the specification tree.
  Note: You cannot select picking points. Smart selection is offered. This means that a sphere or circle, for example, are seen as points.

  The resulting angle is always positive. It is measured in a counterclockwise direction and depends on the order in which points were selected as well as your viewpoint (the normal to the plane is oriented towards you).







- Thickness (DMU only): measures the thickness of an item. For more information, see the appropriate task in the *DMU Space Analysis User's Guide.*

5. Set the desired calculation mode in the Calculation mode drop-down list box.

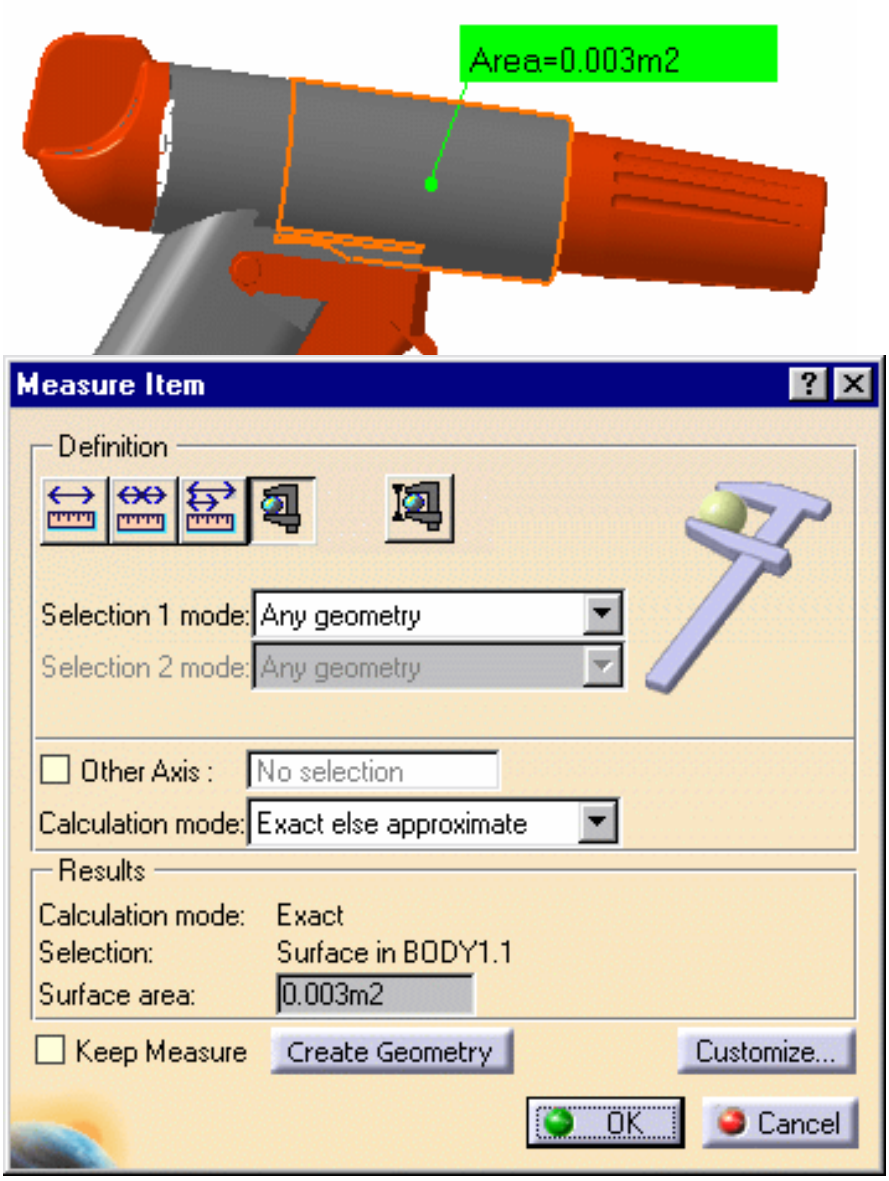
Defining the Calculation Mode

- Exact else approximate (default mode): measures access exact data and wherever possible true values are given. If exact values cannot be measured, approximate values are given (identified by a ~ sign).
- Exact: measures access exact data and true values are given. Note that you can only select exact items in the geometry area or specification tree.
  In certain cases, in particular if products are selected, a warning dialog box informs you that the exact measure could not be made.
- Approximate: measures are made on tessellated objects and approximate values are given (identified by a ~ sign).

Note: You can hide the ~ sign using the **Tools -> Options** command (**General ->Parameters and Measure ->Measure Tools**).

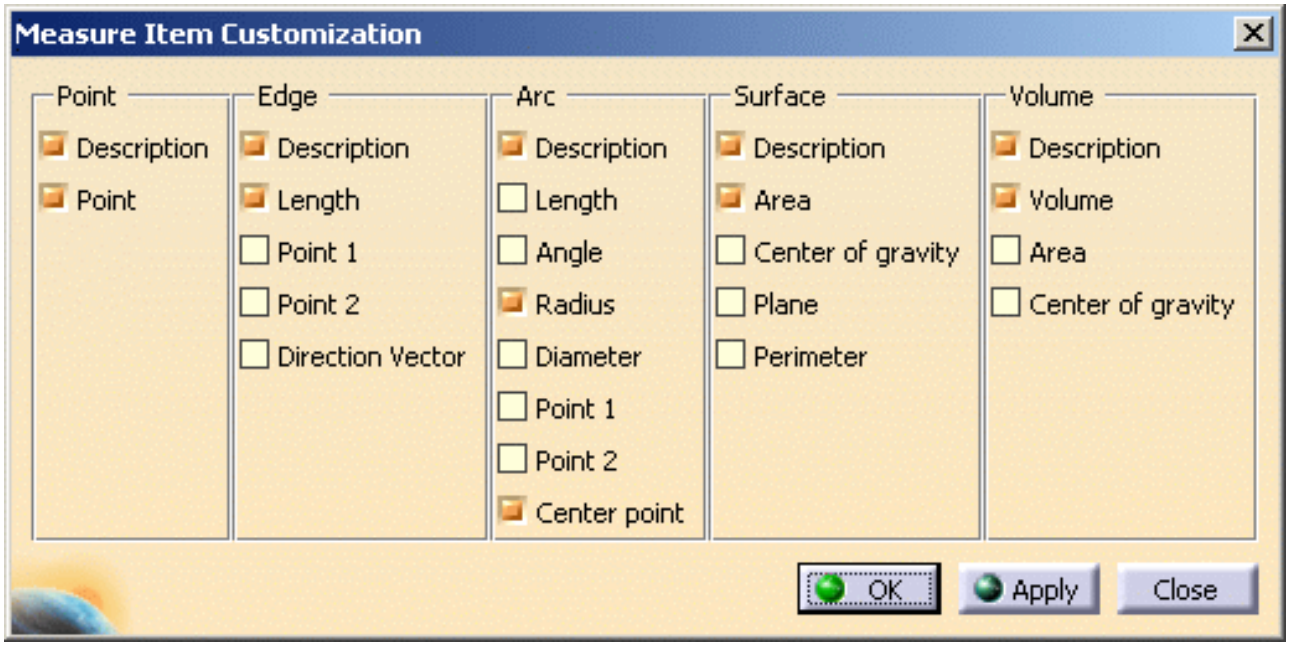
6. Click to select the desired item.

Note: The appearance of the cursor has changed to assist you.



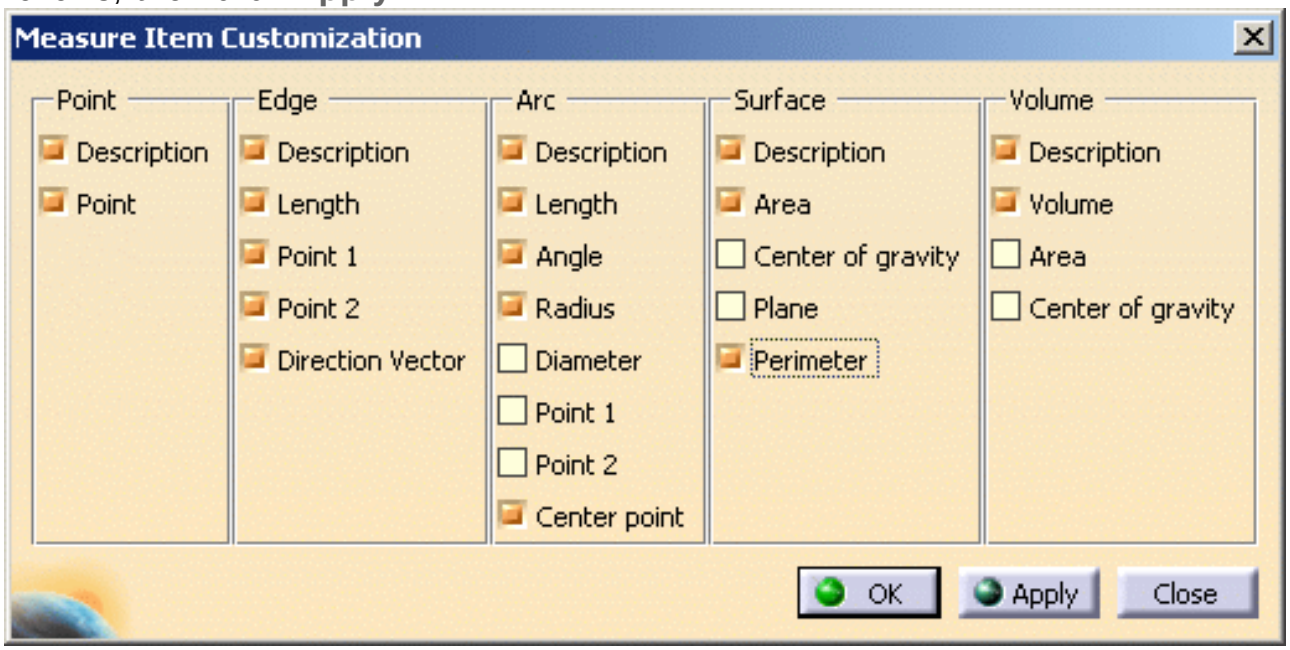
The dialog box gives information about the selected item, in our case a surface and indicates whether the result is an exact or approximate value. The surface area is also displayed in the geometry area.

7. Click Customize... in the Measure Item dialog box to see the properties the system can detect for the various types of item you can select. By default, you obtain:



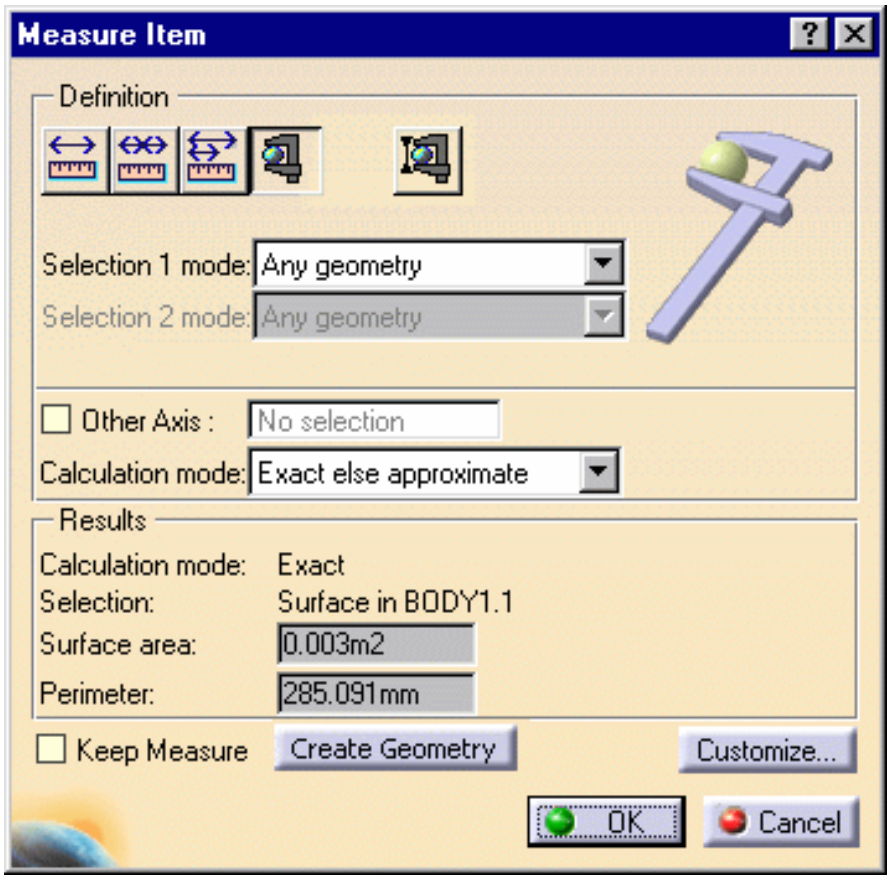
Note: The center of gravity of surfaces is visualized by a point. In the case of non planar surfaces, the center of gravity is attached to the surface over the minimum distance.

8. Set the display of results in both the geometry area and the Measure Item dialog box as follows, then click **Apply**.

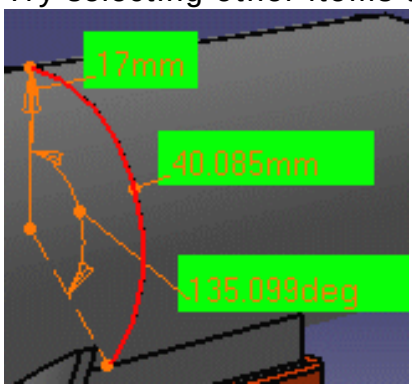


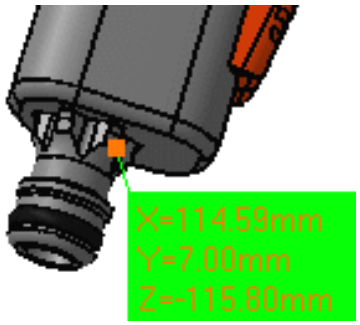
The Measure Item dialog box is updated and now gives the surface perimeter.

Note: Visualization mode does not permit the measure of surface perimeter.

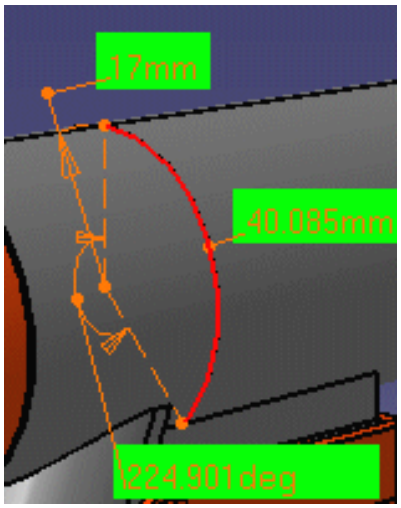


9. Try selecting other items to measure associated properties.





10. If necessary, adjust the presentation of the measure: You can move the lines and text of the measure.



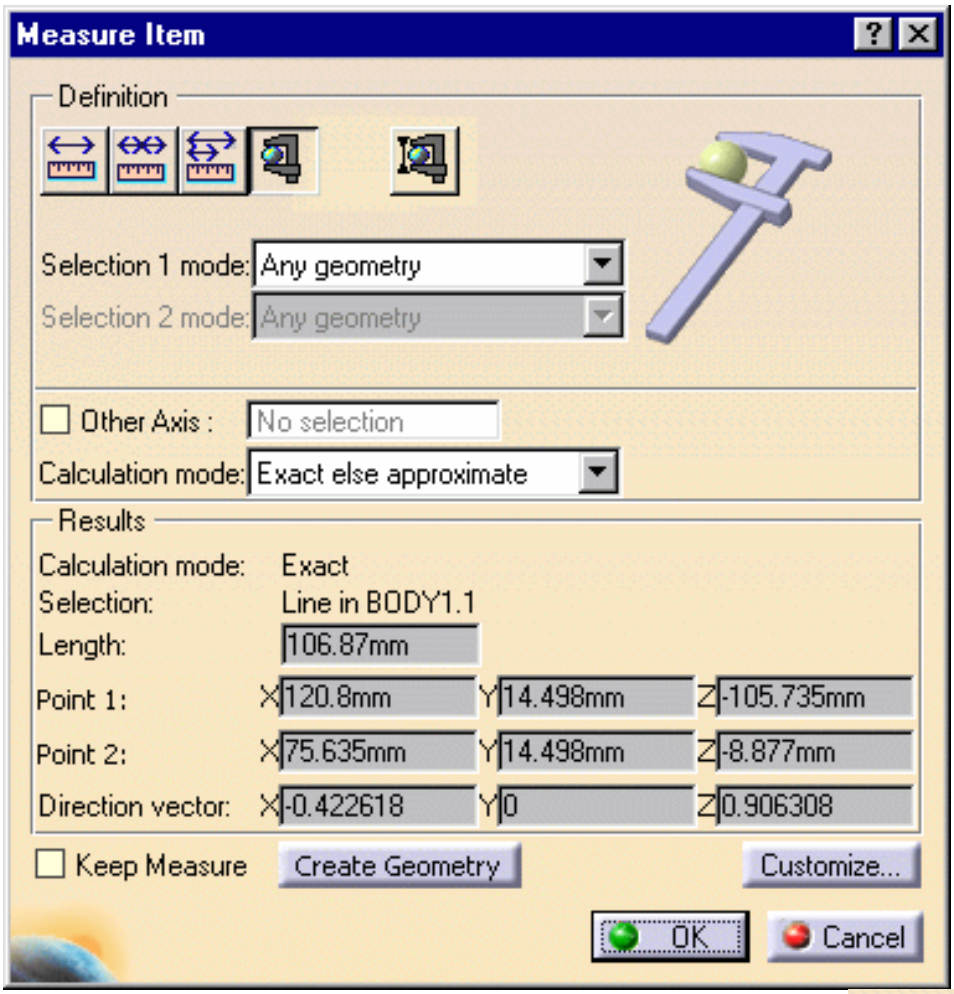
The Properties command (Graphics tab) lets you change the fill color and transparency as well as the color, linetype and thickness of measure lines.

Note: You cannot vary transparency properties, the current object is either the selected color or transparent.

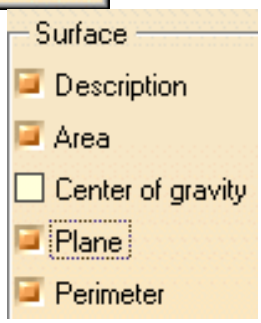
11. Click an edge:

The system detects whether the edge is a line, curve or arc, taking model accuracy into account and displays the properties as set in the Measure Item Customization dialog box.

Note: If the angle of an arc is less than 0.125 degrees, only the arc length is displayed in the geometry area. The angle and radius are not displayed.

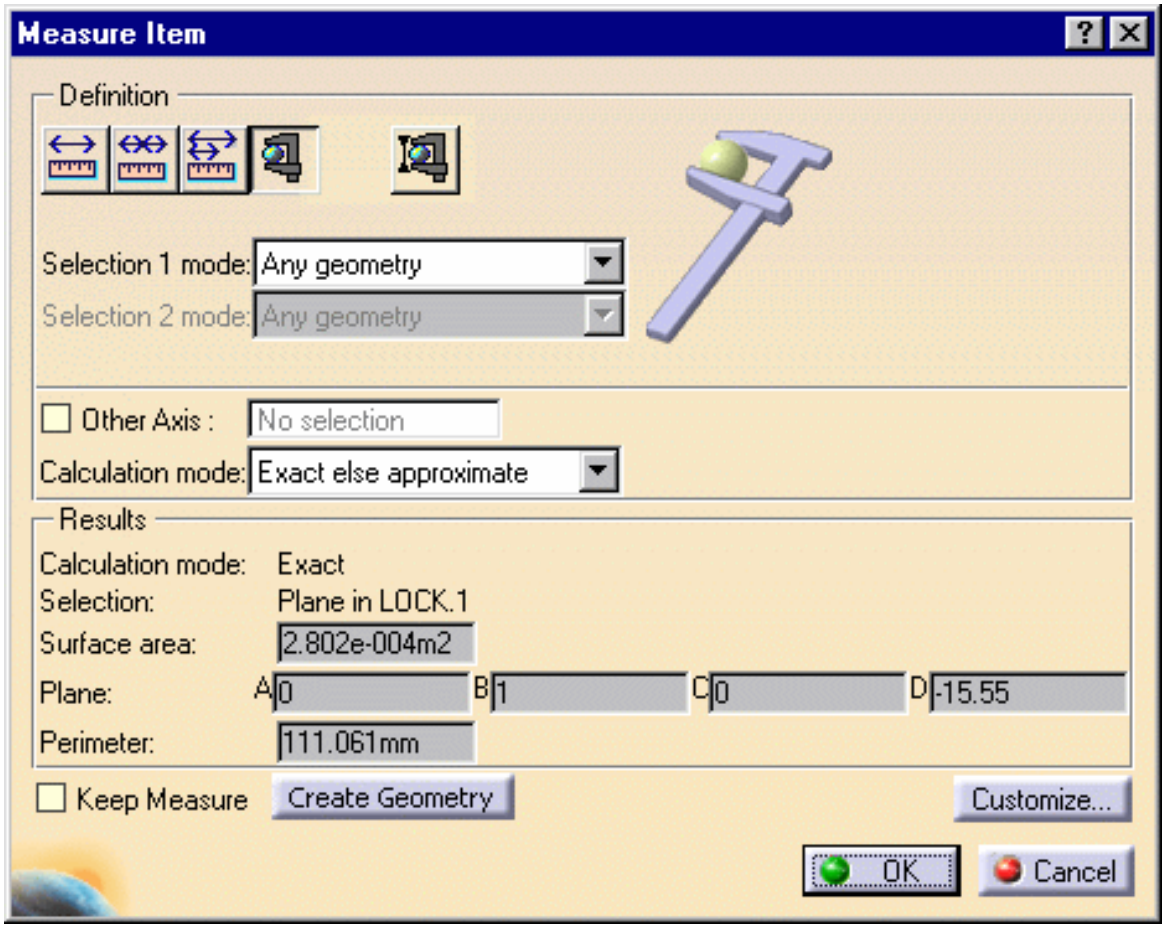


12. Click Plane (under Surface) in the Measure Item Customization dialog box, then click **OK**.



13. Click a planar face to obtain the equation of the plane.

A plane is recognized and is identified in the dialog box. The equation of a plane is: **Ax + By + Cz + D = 0**.



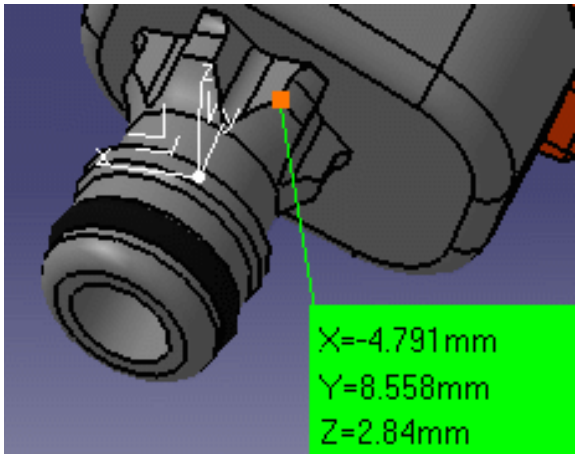
- The Measure Item command lets you access the radius of an exact cylinder or sphere.
- The Measure Item command also recognizes ellipse-type conic sections.
  Description: Ellipse in Part1.1
- Using the Other Selection... command in the contextual menu, you can access the axis of a cylinder as well as the center of a sphere to, for example, measure between two cylinder axes.
- The number of decimal places, the display of trailing zeros and limits for exponential notation is controlled by the Units tab in the Options dialog box (**Tools-> Options**, **General-> Parameters and Measure**). For more information, see the Infrastructure User's Guide.

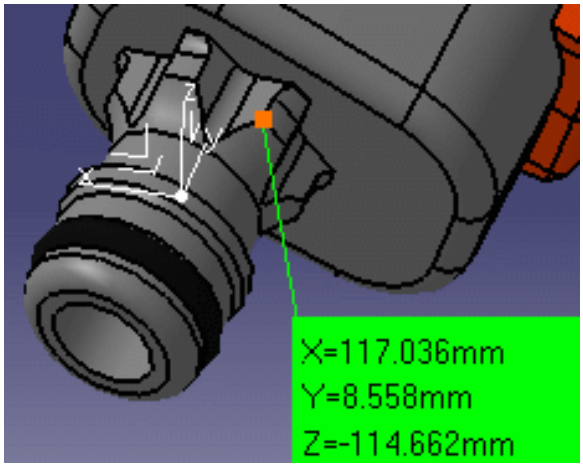
Measuring Properties in a Local Axis System

For this part of the task, you will need a V5 axis system.

14. Check the Other Axis option in the Measure Item dialog box.
15. Select a V5 axis system in the specification tree or geometry area.
16. Make your measure.

Measure made with respect to local axis system: Same measure made with respect to absolute axis system:

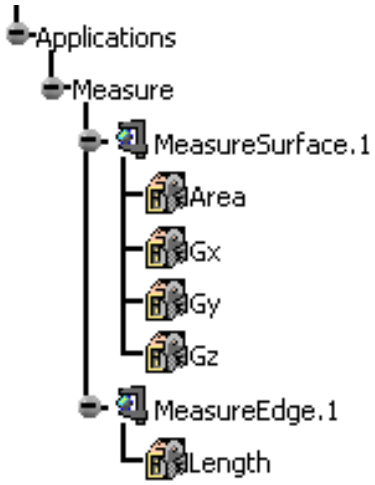




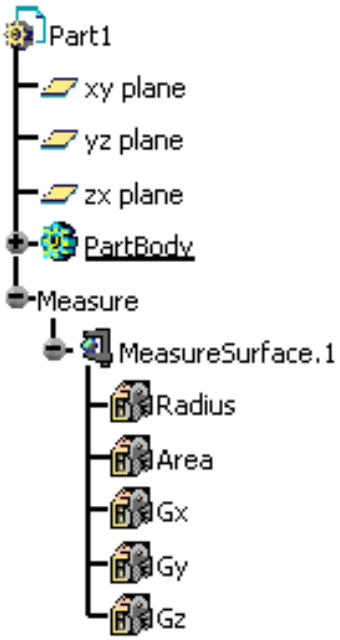
- All subsequent measures are made with respect to the selected axis system. To change the axis system, click the Other Axis field and select another axis system. To return to the main axis system, uncheck the Other Axis option.
- This type of measure is associative: if you move the axis system, the measure is impacted and can be updated.

17. Click **OK** when done.

If you checked the Keep Measure option in the Measure Item dialog box, your measures are kept as features and your specification tree will look something like this if properties of the active product were measured.



Or like this, if properties were those of the active part.



Note: If the product is active, any measures made on the active part are placed in No Show.

Some measures kept as features are associative. In Design Mode, if you modify a part or move a part in a product structure context and the measure is impacted, it will be identified as not up-to-date in the specification tree. You can then update it locally have it updated automatically.

When measures are used to valuate parameters, an associative link between the measure and parameter is created. Measures can also be used in formulas.

Editing Measures

In addition to editing the presentation of the measure, you can also edit the measure itself and change the selection on which it was based. This is particularly useful in design mode where you no longer have to redo your measure.
You can also change selections that no longer exist because they were deleted.

- Double-click the measure in the specification tree or geometry area.
- Make a new selection.

  Note:
  You cannot change the selection 1 mode. If you selected a curve, you must make a selection of the same type, i.e. another curve.

- Click **OK** when done.

Creating Geometry

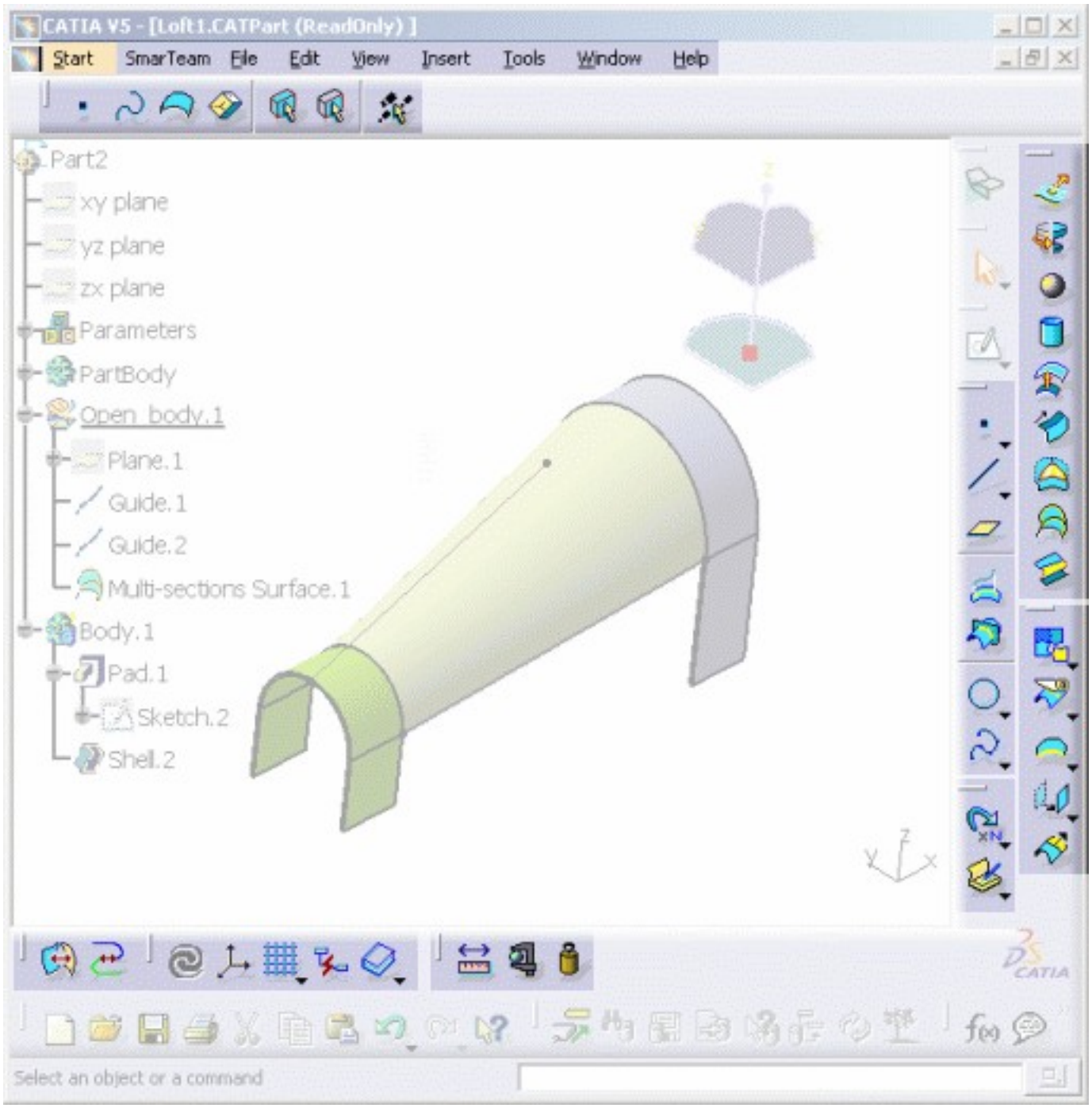
A Create Geometry option in the dialog box lets you create the points and line corresponding to the minimum distance result in a part.



Workbench Description

This section describes the menu and icon commands that are specific to the Wireframe and Surface workbench, which is shown below.

You can click the hotspots on this image to see the related documentation.



Menu Bar
Wireframe Toolbar

Surfaces Toolbar
Operations Toolbar
ReplicationToolbar
Tools Toolbar
Analysis Toolbar
Measure Toolbar
Selection Filter Toolbar
Specification Tree

Wireframe and Surface Menu Bar

Here we will present the various menus and menu commands that are specific to Wireframe and Surface Version 5.



Tasks corresponding to general menu commands are described in the *Infrastructure User's Guide*.

Edit

Please note that most of the Edit commands available here are common facilities offered with the Infrastructure.

The specific Wireframe and Surface Edit commands depend on the type of object being edited: Open Body or other entity.

Command...	Description...
Undo	Cancels the last action
Repeat	Repeats the last performed action
Update	See Updating Your Design
Cut **Copy** **Paste**	See Copying and Pasting
Paste Special...	See Using the Paste Special... Command
Delete	Deletes selected geometry
Search...	Allows searching and selecting objects



Selection Sets Define Selection Sets Find Owning Selection Sets
Allows to define and modify selected objects as sets

Links
Manages links to other documents

Properties
Allows displaying and editing object properties

Scan or Define in Work Object
See Scanning the Part



Edit Inputs...
Allows to edit the object inputs and parameters

Change Body...
Moving an Open Body to a new Open Body

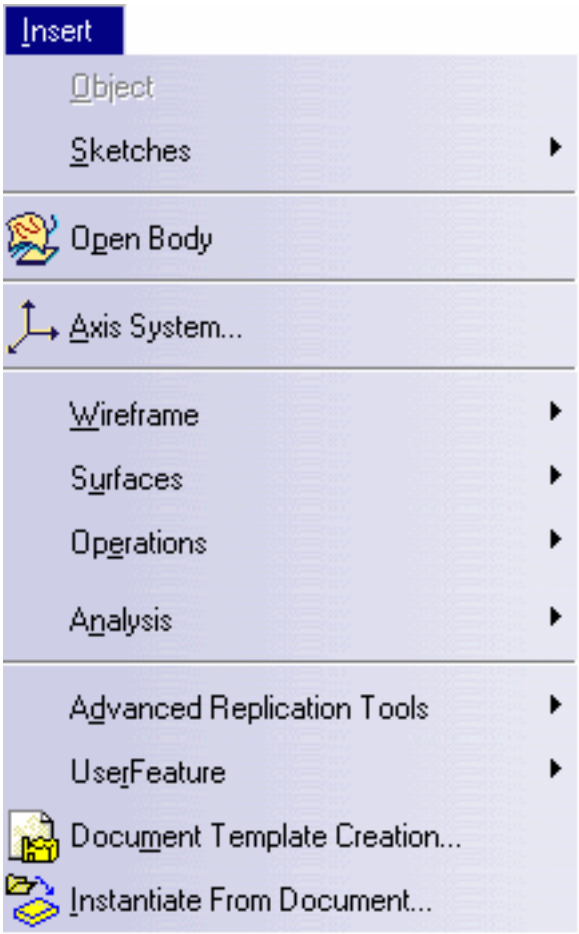
AutoSort
Allows to reorder the Open Body's children according to the logical construction order

Reorder Children
See Editing Definitions

Show Components
Hide Components
Hiding/Showing Open Bodies and Their Contents

Create Group	Allows creating groups. Refer to *Generative Shape Design User's Guide* - Basic Tasks - Managing Groups
Reset Properties	Allows resetting object properties

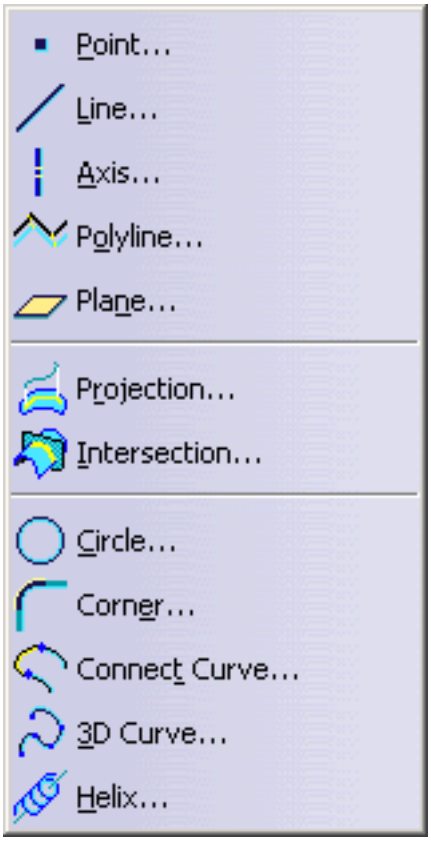
Insert



For...	See...
Sketcher...	Refer to the *Sketcher User's Guide*.
Open body	See Managing Open Bodies
Axis System	Allows the creation of local axis-system
Wireframe	Insert > Wireframe
Surfaces	Insert > Surfaces
Operations	Insert > Operations
Analysis	Insert > Analysis
Advanced Replication Tools	Insert > Advanced Replication Tools
Document Template Creation...	Allows the creation of part templates. Refer to the chapter Creating a Part Template in the *Product Knowledge Template User's Guide.*
Instantiate From Document...	Instantiating PowerCopies

Insert -> Wireframe

For...	See...
Point...	Creating Points
Line...	Creating Lines
Axis...	Creating an Axis
Polyline...	Creating Polylines
Plane...	Creating Planes



Projection...	Creating Projections
Intersection...	Creating Intersections
Circle...	Creating Circles
Corner...	Creating Corners
Connect Curve...	Creating Connect Curves
Spline...	Creating Splines
Helix...	Creating Helices

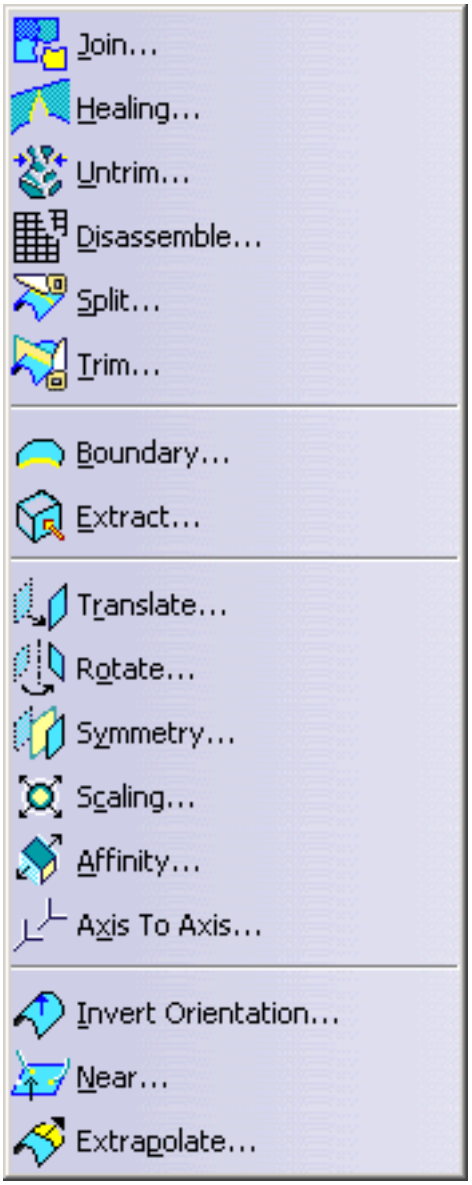
Insert -> Surfaces



For...	See...
Extrude...	Creating Extruded Surfaces
Revolve...	Creating Revolution Surfaces
Sphere...	Creating Spherical Surfaces
Cylinder...	Creating Cylindrical Surfaces
Offset...	Creating Offset Surfaces
Sweep...	Creating Swept Surfaces
Fill...	Creating Filling Surfaces
Loft...	Creating Lofted Surfaces
Blend...	Creating Blend Surfaces

Insert -> Operations

For...	See...
Join...	Joining Geometric Elements
Healing...	Healing Geometry



Untrim...	
Disassemble...	Disassembling Surfaces
Split...	Splitting Geometry
Trim...	Trimming Geometry
Boundary...	Creating Boundary Curves
Extract...	Extracting Geometry
Translate...	Translating Geometry
Rotate...	Rotating Geometry
Symmetry...	Performing Symmetry on Geometry
Scaling...	Transforming Geometry by Scaling
Affinity...	Transforming Geometry by Affinity
Axis To Axis...	Transforming Elements from an Axis to Another
Invert Orientation	Inverting the Orientation of Geometry
Near...	Creating Nearest Entity of a Multiple Element
Extrapolate...	Extrapolating Curves and Extrapolating Surfaces

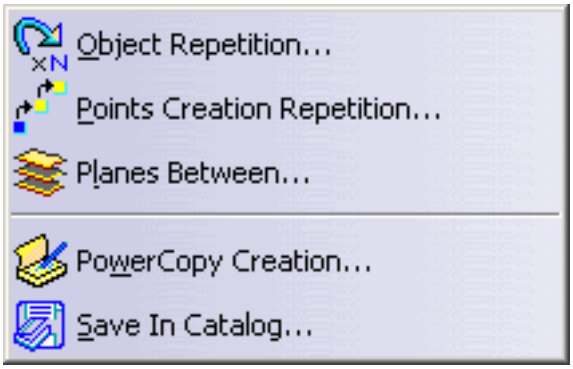
Insert -> Analysis



For...	See...
Connect Checker	Checking Connections Between Surfaces
Curve Connect Checker	Checking Connections Between Curves

Insert -> Advanced Replication Tools

For...	See...
Object Repetition...	Repeating Objects



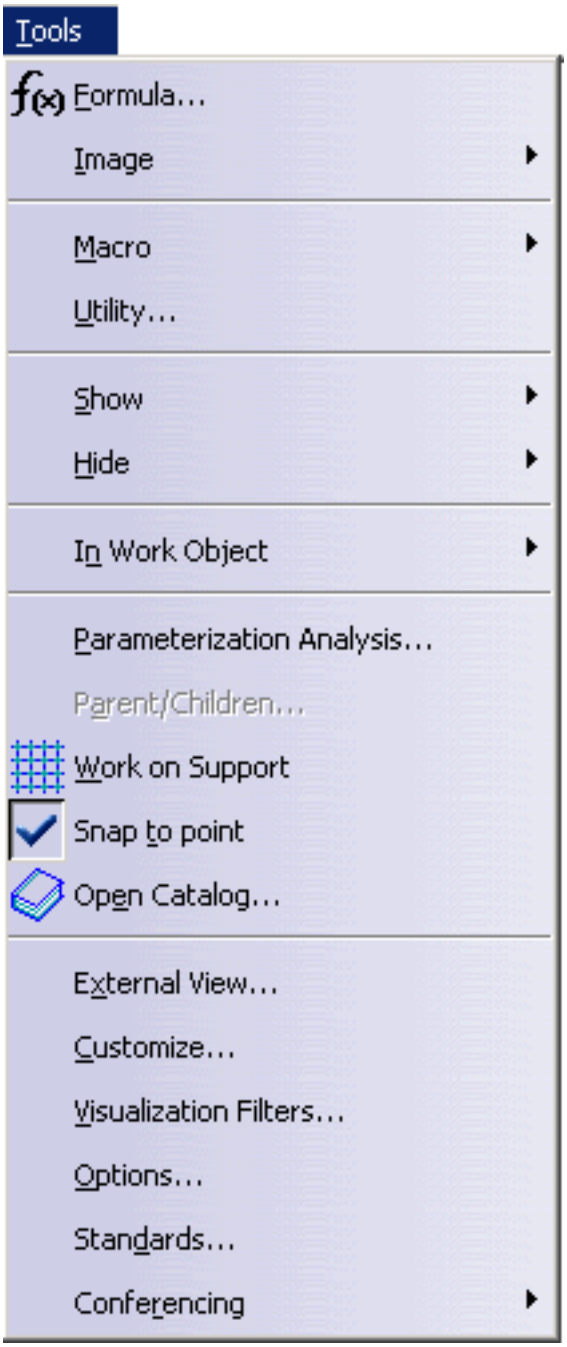
Points Creation Repetition...	Multiple Points
Planes Between...	Creating Planes Between Other Planes
PowerCopy Creation...	Creating PowerCopies
Save In Catalog...	Saves powercopies into a catalog

Tools

Please note that most of the Tools commands available here are common facilities offered with the Infrastructure.

Specific Wireframe and Surface Tools commands are described in the present document.

Command...	Description...
Formula...	Allows editing parameters and formula.
Image	Allows capturing images.
Macro	Allows recording, running and editing macros.
Show Hide	Hiding/showing open bodies and their contents
In Work Object	See Scanning the Part
Parameterization Analysis	See Analyzing Using Parameterization
Parent/ Children...	Allows viewing the parents and children of a selected object.
Work on Support	See Working with a Support
Snap to point	See Working with a Support
Open Catalog...	Allows the opening of catalogs, for PowerCopies for example



External View... Allows specifying a feature as a reference for other products/applications. Refer to the chapter Managing Open Bodies in the *Generative Shape Design User's Guide*

Customize... Allows customizing the workbench.

Visualization Filters... Allows to manage layer filters.

Options Allows customizing settings.

Up Next

Wireframe Toolbar

This toolbar contains the following tools for creating wireframe geometry.






See Points


See Multiple Points


See Lines


See Axis


See Polylines


See Planes


See Projections


See Intersections


See Circles


See Corners


See Connect curves


See Splines


See Helix

Back Up Next

Surfaces Toolbar

This toolbar contains the following tools for creating surface geometry.



 See Extruded Surfaces

 See Surfaces of Revolution

 See Spherical Surfaces

 See Cylindrical Surfaces

 See Offset Surfaces

 See Swept Surfaces

 See Filled Surfaces

 See Lofted Surfaces

 See Blend Surfaces

Back Up Next

Operations Toolbar

This toolbar contains the following tools for performing operations on surface and wireframe elements.



 See Joining Geometry

 See Healing Geometry

 See Restoring a Surface

 See Disassembling Surfaces

 See Splitting Geometry

 See Trimming Geometry

 See Boundary Curves

 See Extracting Geometry

 See Translating Geometry

 See Rotating Geometry

 See Performing a Symmetry on Geometry

 See Transforming Geometry by Scaling

 See Transforming Geometry by Affinity

 See Transforming elements into a new axis-system

 See Extrapolating Curves and Extrapolating Surfaces

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Replication Toolbar

This toolbar contains the following tools to help you model your part designs.



 See Repeating Objects

 See Multiple Points

 See Creating Planes Between Other Planes

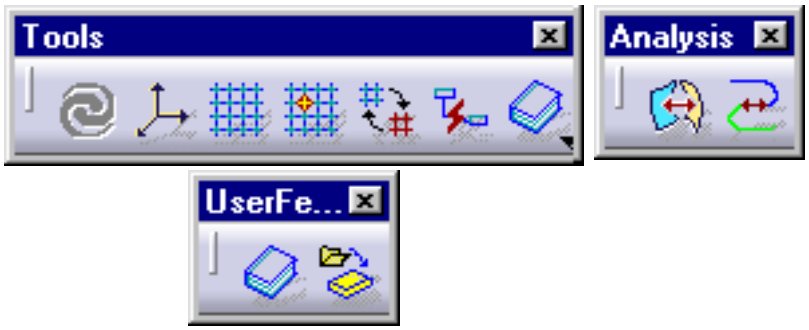
 See Creating PowerCopies

 See Instantiating PowerCopy

Back Up Next

Generic Tools Toolbar

These toolbars contain the following tools to help you design and analyze your parts.



 See Updating Your Design

 See Axis-System

 See Working with a Support

 See Working with a Support

 See Working with a Support

 See Creating Datums

 See Instantiating PowerCopies

 See Instantiating PowerCopies

 See Checking Connections Between Surfaces

 See Checking Connections Between Curves

 See Inverting the Orientation of Geometry

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Analysis Toolbar

These toolbars contain the following tools to help you apply materials onto surfaces for analysis purposes.





See Applying Materials Onto Surfaces

Back Up Next

Measure Toolbar

This toolbar contain the following tools to help you create a persistent and associative link between a measure and a parameter.




See Measuring Minimum Distances and Angles


See Measuring Properties


See Measuring Inertia

Back Up Next

Selection Filter Toolbar

This toolbar contains the following tools to help you manage sub-geometry selection.



See Selecting using A Filter

Back Up Next

Specification Tree

Within the Wireframe And Surface workbench, you can generate a number of elements that are identified in the specification tree by the following icons.

Further information on general symbols in the specification tree are available in Symbols Used in the Specification Tree.

 Sketch

 Point

 Multiple Points

 Line

 Axis

 Polyline

 Plane

 Projection

 Intersection

 Circle

 Corner

 Connect Curve

 Spline

 Helix

 Extrude

 Revolve

 Sphere

 Join

 Healing

 Surface

 Split

 Trim

 Boundary

 Extract

 Translate

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 Symmetry

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 Affinity

 Axis To Axis

 Near

 Extrapolate

 Inverse

 Multiple Planes


Cylinder


Multi-Output


Offset


Open Body


Sweep


Power Copy


Fill


Working support


Multi-sections Surfaces


Surface Connection Analysis


Blend


Curve Connection Analysis

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Glossary

A C E F J L O P R S T W

A

affinity An operation in which an element is transformed by applying X, Y, Z affinity ratios with respect to a reference axis system.

C

child A status defining the hierarchical relation between a feature or element and another feature or element.

constraint A geometric or dimension relation between two elements.

E

extruded surface A surface that is obtained by extruding a profile along a specified direction.

F

feature A component of a part.

J

join An operation in which adjacent curves or adjacent curves can be joined.

L

lofted surface A surface that is obtained by sweeping one or more planar section curves along a spine, which may be automatically computed or user-defined. The surface can be made to follow one or more guide curves.

O

offset surface A surface that is obtained by offsetting an existing surface a specified distance.

P

parent A status defining the hierarchical relation between a feature or element and another feature or element.

part A 3D entity obtained by combining different features. It is the content of a CATPart document.

part body A component of a part made of one or several features.

profile An open or closed shape including arcs and lines.

R

revolution surface A surface that is obtained by revolving a profile around an axis.

rotate An operation in which an element is rotated by a specified angle about an given axis.

S

scaling An operation that resizes an element to a percentage of its initial size.

sketch A set of geometric elements created in the Sketcher workbench. For instance, a sketch may include a profile, construction lines and points.

split An operation in which one element is cut by another element.

swept surface A surface obtained by sweeping a profile in planes normal to a spine curve while taking other user-defined parameters (such as guide curves and reference elements) into account.

symmetry An operation in which an element is transformed by means of a mirror symmetry with respect to a reference plane, line or point.

T

translate An operation in which an element is displaced a specified distance along a given direction.

trim An operation in which two element cut each other mutually.

W

wireframe element Elements such as points, lines or curves that can be used to represent the outline of a 3D object.

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SheetMetal Design



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- More Information

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- Entering the Workbench
- Defining the Parameters
- Creating the First Wall
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Preface

The Sheet Metal Design workbench is a new generation product offering an intuitive and flexible user interface. It provides an associative feature-based modeling making it possible to design sheet metal parts in concurrent engineering between the unfolded or folded part representation.

Sheet Metal Design offers the following main functions:

- Associative and dedicated Sheet Metal feature based modeling
- Concurrent engineering between the unfolded or folded part representation
- Access to company defined standards tables
- Dedicated drawing capability including unfolded view and specific settings.

All sheetmetal specifications can be re-used by the Knowledge Advisor workbench to capture corporate knowledge and increase the quality of designs.

Natively integrated, Sheet Metal Design offers the same ease of use and user interface consistency as all V5 applications.

As a scalable product, Sheet Metal Design can be used in cooperation with other current or future companion products in the next CATIA generation such as Assembly Design and Generative Drafting. The widest application portfolio in the industry is also accessible through interoperability with CATIA Solutions Version 4 to enable support of the full product development process from initial concept to product in operation.

Using This Product
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Preface

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Using This Product

This guide is intended for the user who needs to become quickly familiar with the CATIA Version 5 Sheet Metal Design product. The user should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars.

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

The next sections deal with the handling of more detailed capabilities of the product.

This guide also includes a guide intended for the user of the New Sheet Metal Design product. You will find more information in the Working with the New SheetMetal Design Workbench section.

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Where to Find More Information

Prior to reading this book, we recommend that you read the *Infrastructure User's Guide.*

The *Part Design User's Guide*, the *Assembly Design User's Guide* and the *Generative Drafting User's Guide* may prove useful.

Finally, you can read the *Sheet Metal Production User's Guide* to find out more about that product and to fully use the interoperability between the two products.

Conventions

What's New?

This table identifies what new or improved capabilities have been documented in Version 5 Release 12 of the Sheet Metal Design and the NEW Sheet Metal Design workbenches.

There is no new or improved capabilities in Version 5 Release 12 of the CATIA Sheet Metal Design workbench.

NEW Sheet Metal Design

New Functionalities

Creating Tangent Walls

Creating Rolled Walls

Creating Bends From a Line

Aggregating Bodies

Displaying Swept Walls Axes

Creating a Louver

Saving As DXF

Integration With Weld Design

Enhanced Functionalities

Editing the Sheet and Tool Parameters

You can now set the bend radius value with a radius=0 using the DIN standard

Creating Rectangular Patterns
Creating Circular Patterns
Creating User-Defined Patterns

You can now duplicate a stamp without fillet

Manually Creating Bends from Walls

You can now create a bend on a flange or on a stamp

You can now create a bend with a radius=0

Creating Walls From an Edge

You can now create a wall from an edge on a flange and on a stamp

Getting Started

Before getting into the detailed instructions for using Version 5 CATIA - Sheet Metal Design, the following tutorial provides a step-by-step scenario demonstrating how to use key functionalities.

The main tasks proposed in this section are:

Entering the Workbench
Defining the Parameters
Creating the First Wall
Creating the Side Walls
Creating a Cutout
Creating Automatic Bends
Extracting Drawings from the Sheet Metal Part



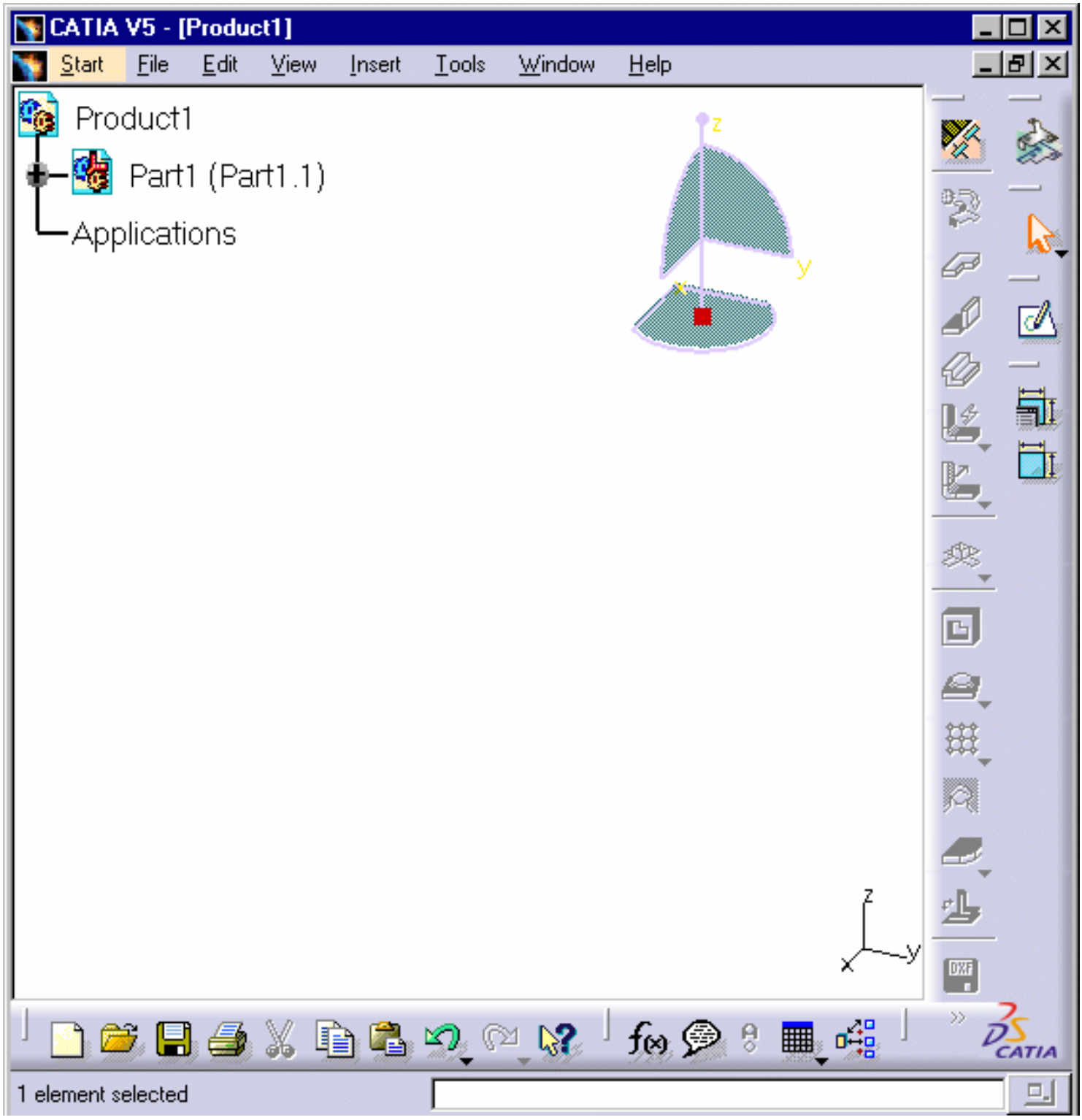
All together, these tasks should take about 15 minutes to complete.

Entering the Sheet Metal Workbench

The Sheet Metal Design functions are available when you are in the Part environment. Several functions are integrated from the Part Design workbench.

This task shows how to enter the workbench.

Choose the **Mechanical Design -> Sheet Metal Design** item from the **Start** menu. The Sheet Metal toolbar is displayed and ready to use.





You may add the **Sheet Metal Design** workbench to your Favorites, using the **Tools -> Customize** item. For more information, refer to the *Infrastructure User's Guide.*



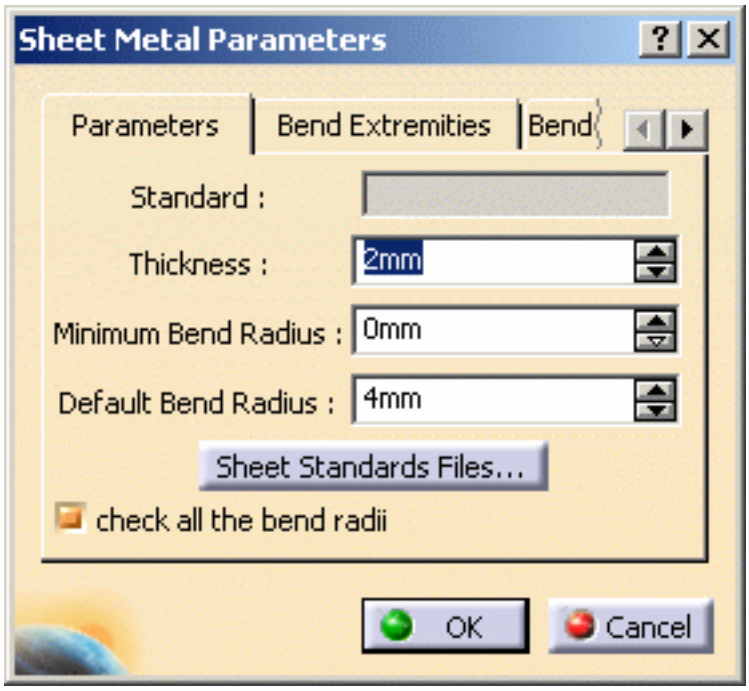
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Defining the Sheet Metal Parameters

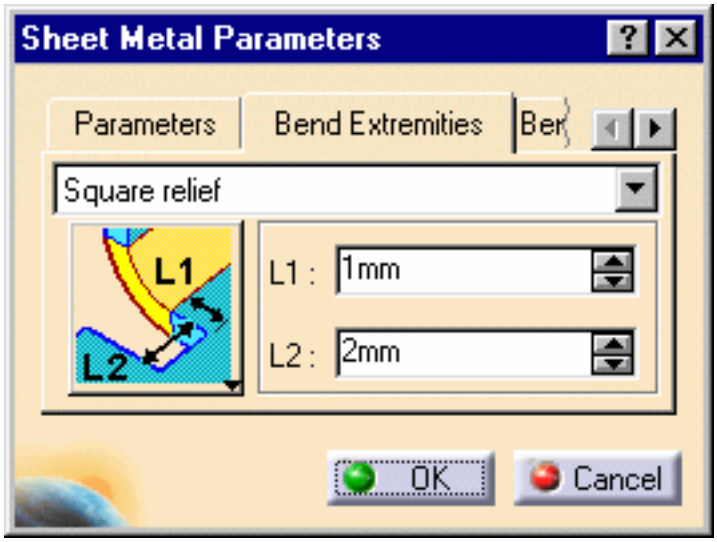
This task shows you how to configure the sheet metal parameters.

1. Click the **Sheet Metal Parameters** icon .

   The Sheet Metal Parameters dialog box is displayed.



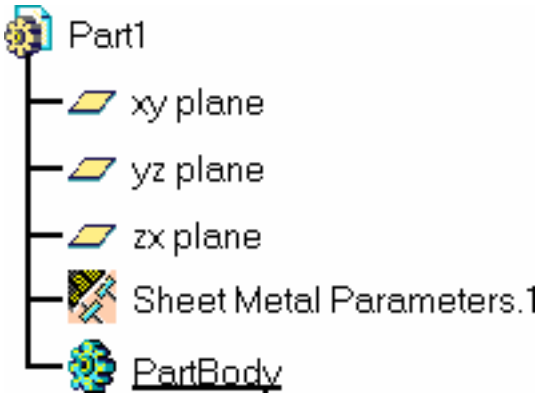
2. Enter 1mm in the **Thickness** field.
3. Enter 5mm in the **Default Bend Radius** field.
4. Select the **Bend Extremities** tab.



5. Select **Tangent** in the Bend Extremities combo list.

An alternative is to select the bend type in the graphical combo list.

6. Click **OK** to validate the parameters and close the dialog box.
   The **Sheet Metal Parameters** feature is added in the specification tree.



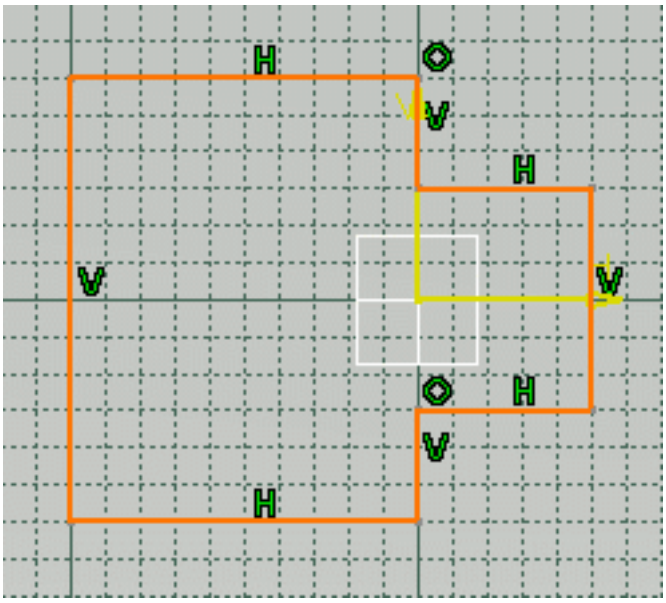
The other two tabs are not used in this scenario.

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Creating the First Wall

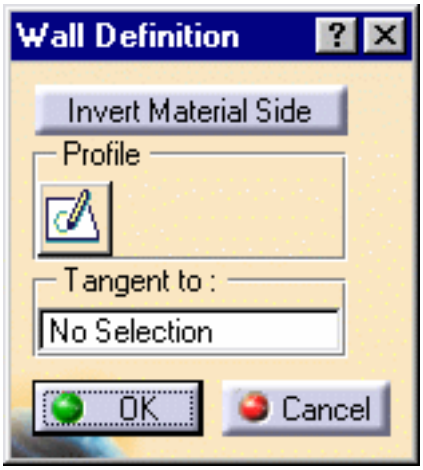
This task shows how to create the first wall of the Sheet Metal Part.

1. Click the **Sketcher** icon then select the **xy plane**.
2. Select the **Profile** icon .
3. Sketch the contour as shown below:

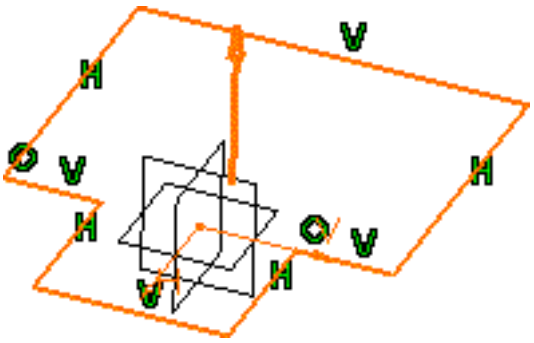


4. Click the **Exit workbench** icon to return to the 3D world.
   The sketch remains selected.

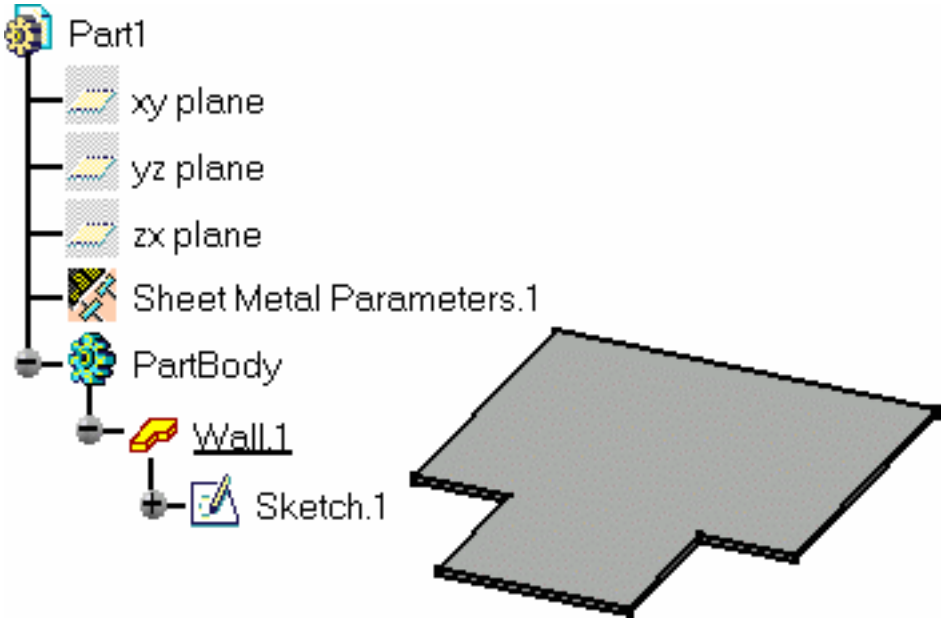
5. Click the **Wall** icon .
   The Wall Definition dialog box opens.



By default, the Material Side is set to the top.



6. Click OK.
   The **Wall.1** feature is added in the specification tree



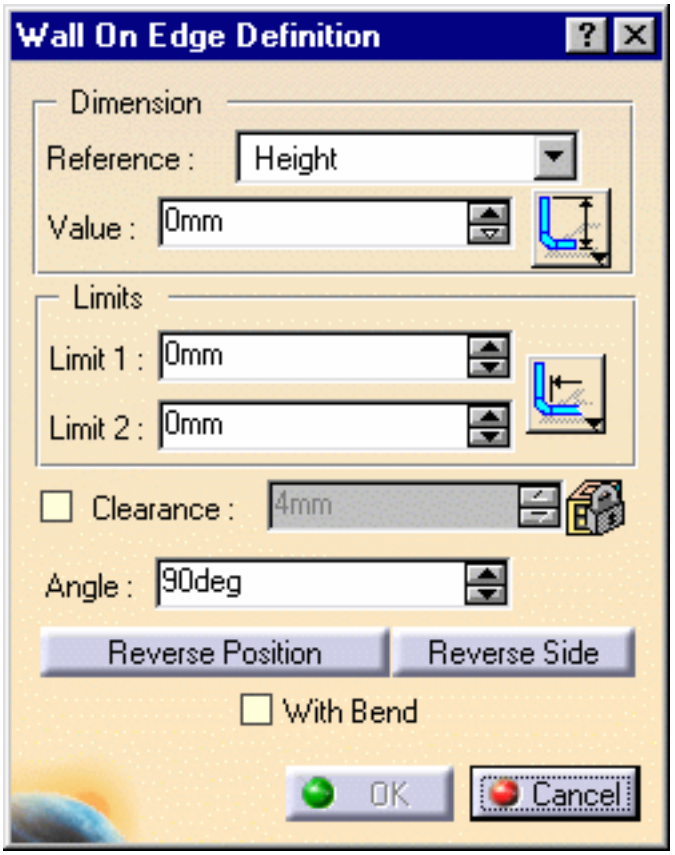
The first wall of the Sheet Metal Part is known as the Reference wall.

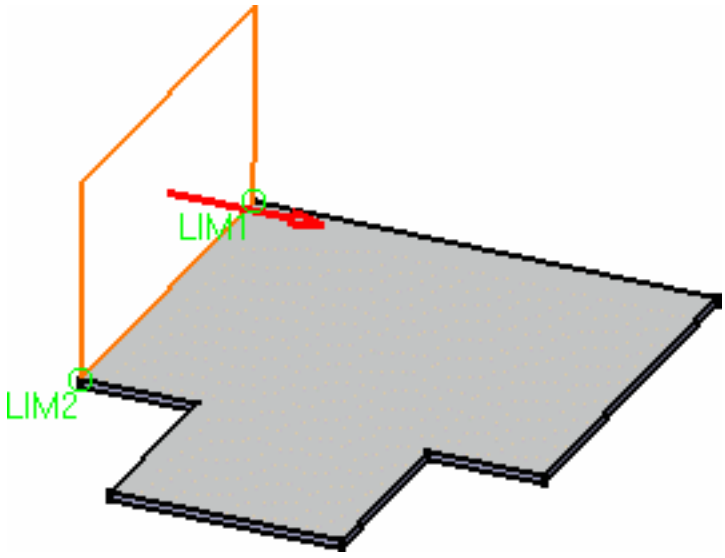
Back Up Next

Creating the Side Walls

This task shows you how to add other walls to the Sheet Metal part.

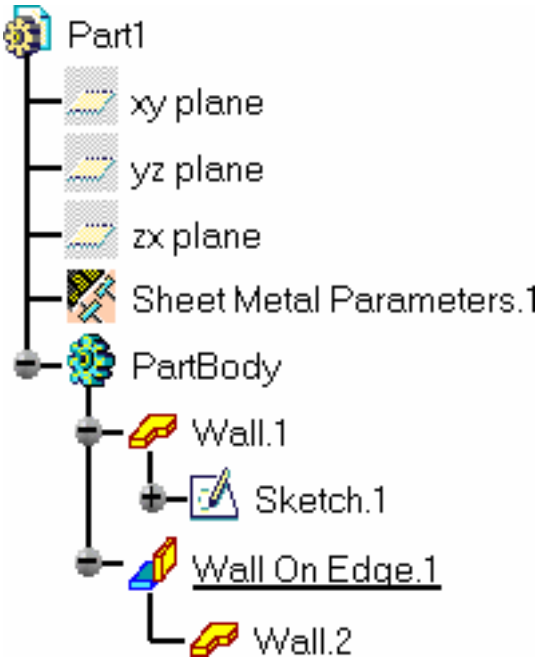
1. Click the **Wall on Edge** icon .
   The Wall On Edge Definition dialog box opens.
2. Select the left edge.
3. Enter 50mm in the **Value** field.
   The application previews the wall.

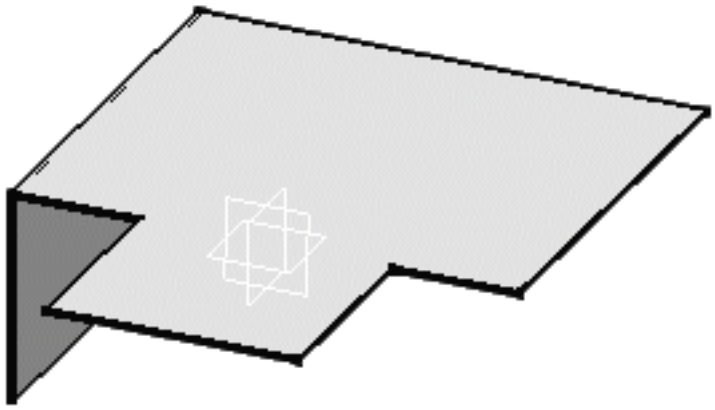




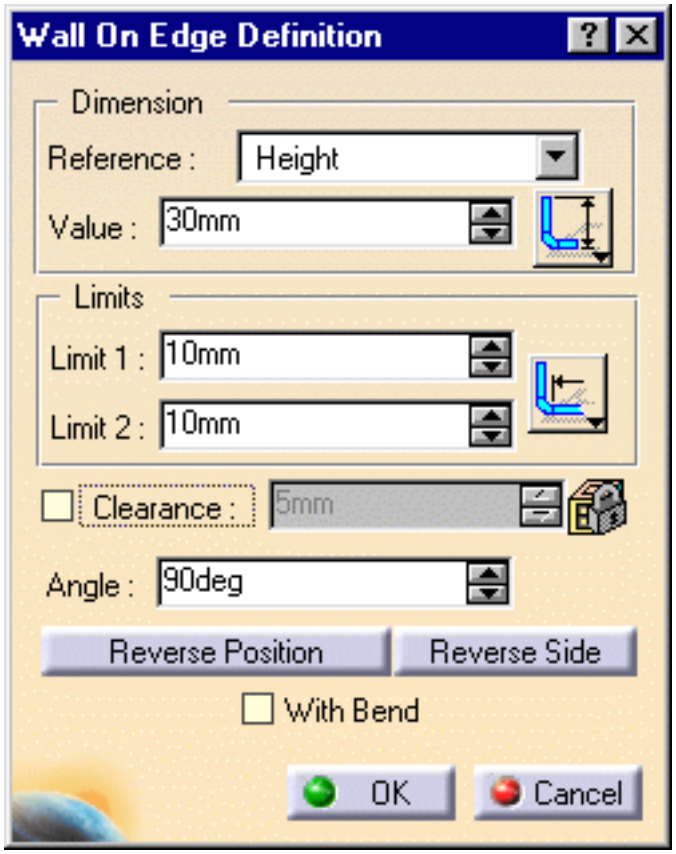
By default, the material side is such that it ensures a continuity with the reference profile. If needed, invert it using the **Reverse side** button, or clicking the arrow.

4. Click the **Reverse Position** button to Invert the sketch profile.
5. Click **OK**.
   The wall is created and the **Wall On Edge.1** feature is displayed in the specification tree:

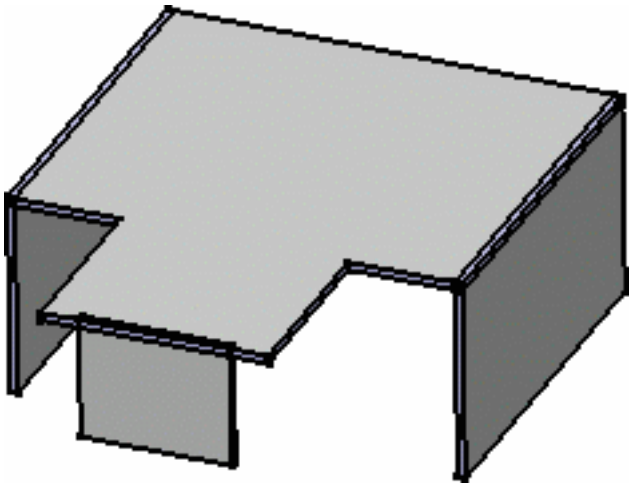




6. Select the right edge.
7. Click the **Wall on Edge** icon  again.
   The Wall On Edge Definition dialog box opens with the parameters previously selected.
8. Invert the sketch profile and click **OK** to validate.
9. Click the **Wall on Edge** icon  again.
10. Select the front edge.
    The Wall Definition dialog box opens with the parameters previously selected.
11. Enter 30mm in the **Value** field and 10mm in the **Limit1** and **Limit2** fields, then invert the sketch profile.
12. Press **OK** to validate.



The final part looks like this:





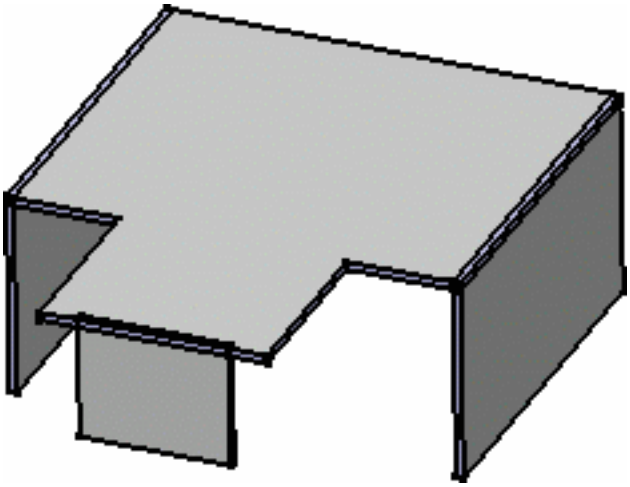
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Creating a Cutout

In this task, you will learn how to:

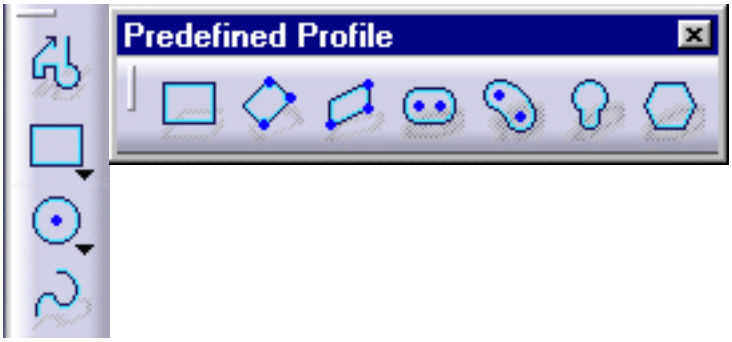
- open a sketch on an existing face
- define a contour in order to create a cutout.

1. Select **Wall On Edge.2** from the geometry area to define the working plane.
2. Click the Sketcher icon .

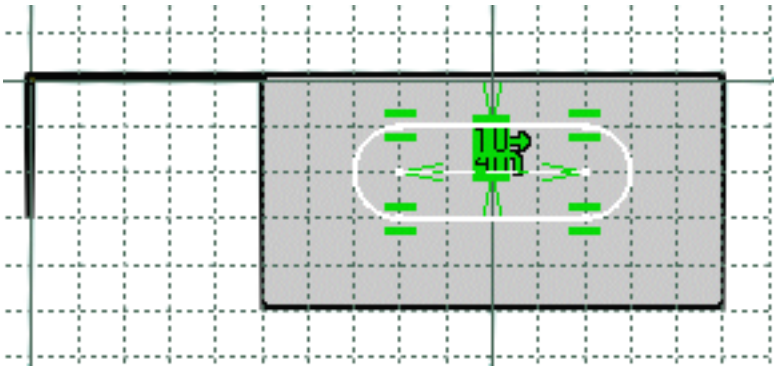


3. Click the **Elongated Hole** icon  to create the contour.

To access the oblong profile, click the black triangle on the **Rectangle** icon.
It displays a secondary toolbar.



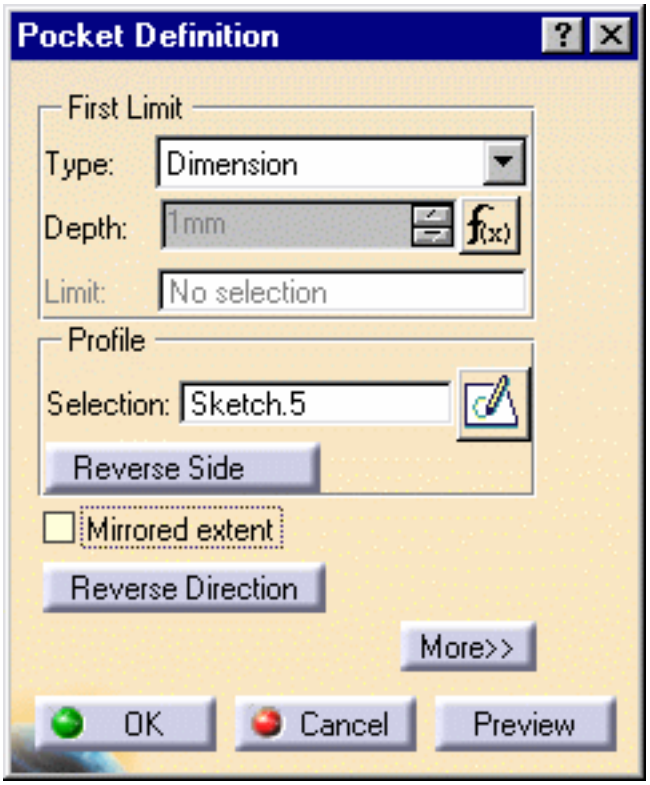
4. Click to create the first point and drag the cursor.
5. Click to create the second point.
   The first semi-axis of the profile is created.
6. Drag the cursor and click to create the third point.
   The second semi-axis is created and the oblong profile is displayed.



7. Click the **Exit workbench** icon  to return to the 3D world.

8. Select the **Cutout** icon .

   The **Pocket Definition** dialog box is displayed and a cutout is previewed with default parameters.

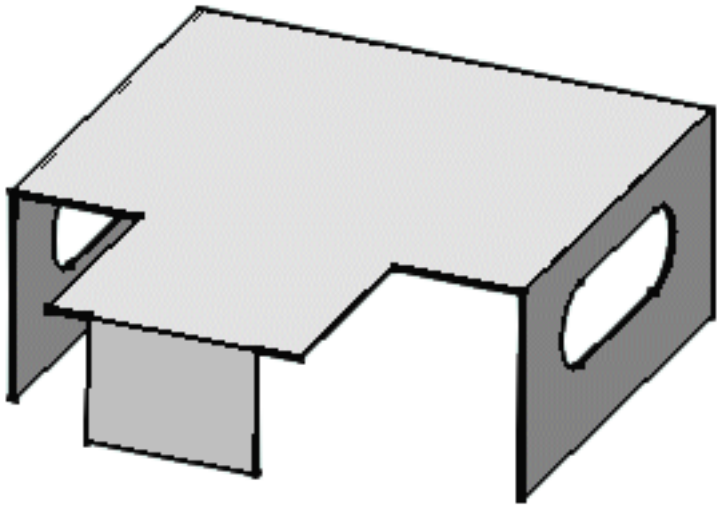


9. Set the Type to **Up to last** option to define the limit of your cutout.

   This means that the application will limit the cutout onto the last possible face, that is the opposite wall.

10. Click **OK**.

    This is your cutout:





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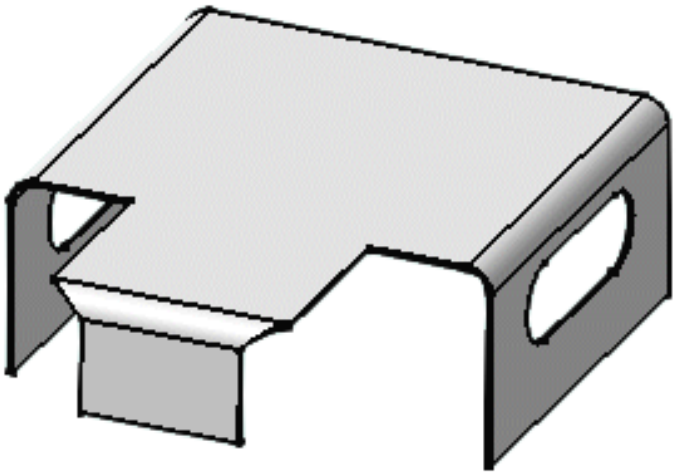
Creating the Bends Automatically

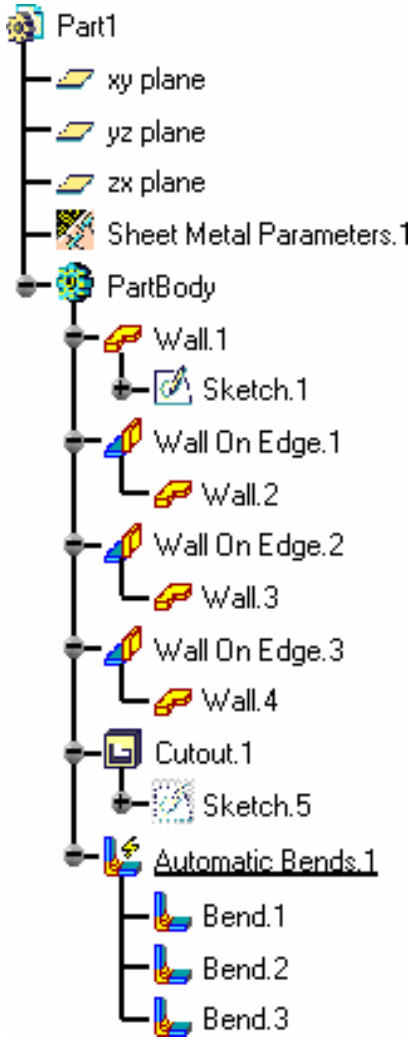
This task shows how to create the bends automatically.

1. Click the **Automatic Bends** icon .
   The bends are created.

CATIA displays the bends creation in the specification tree: **Automatic Bends.1**

The Sheet Metal part looks like this:





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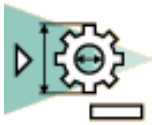
Extracting Drawings from the Sheet Metal Part

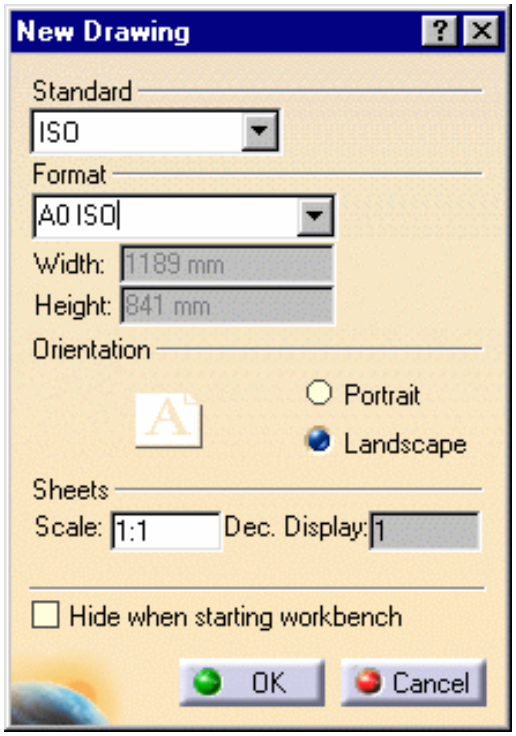
This task shows how to create the Sheet Metal Part views in the Generative Drafting workbench.

The Sheet Metal part is displayed.

1. Click  or select **File -> New...**
2. Select the **Drawing** type and click OK.

   The Generative Drafting workbench is launched.
   The New Drawing dialog box opens.





3. Click **OK**.

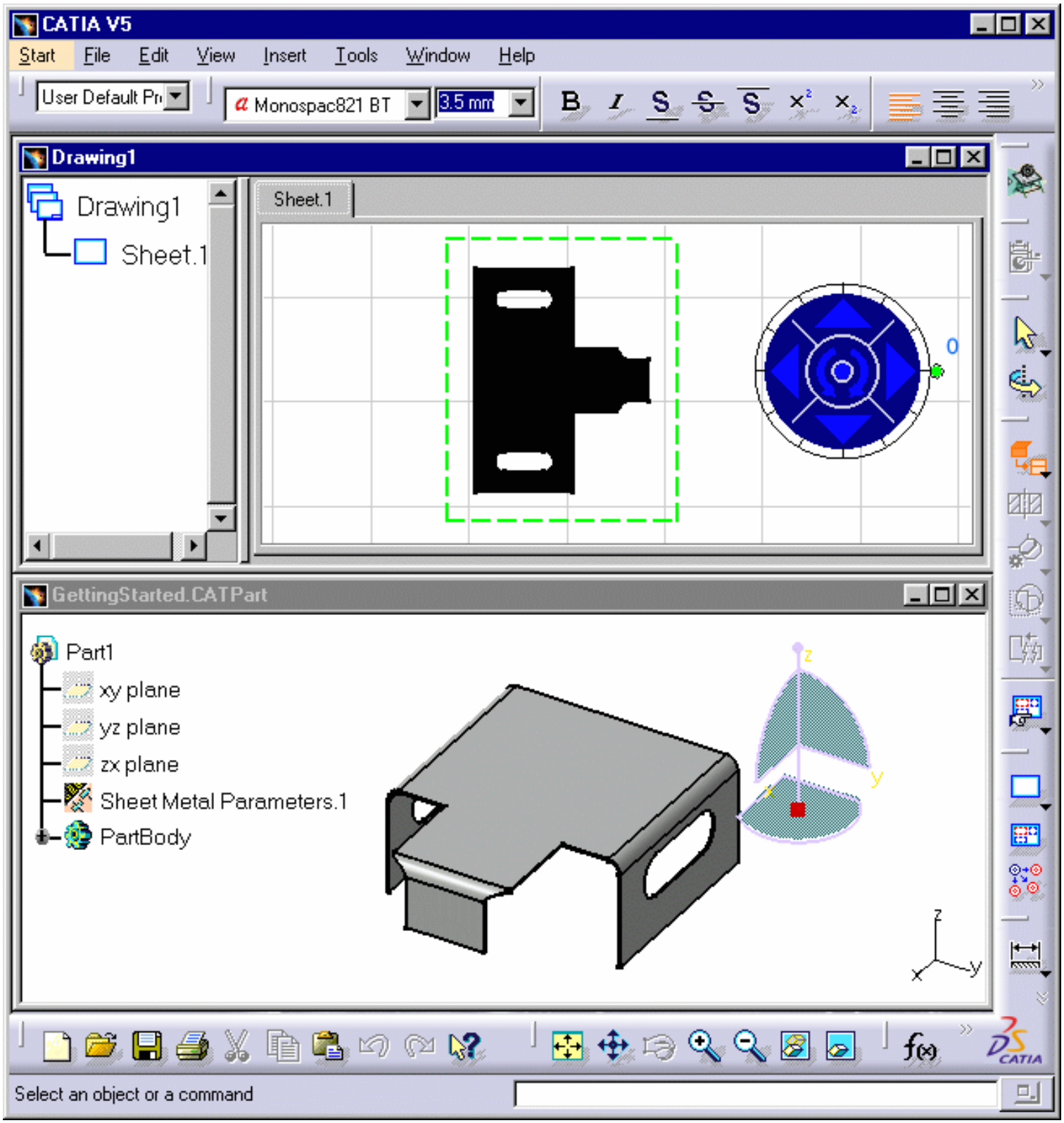
   For more information about this workbench, refer to *Generative Drafting User's Guide*.

4. The drawing sheet appears.
5. Tile the windows horizontally using the **Window -> Tile Horizontally** menu item.
6. Select the Unfolded View icon  in the Projections toolbar from Generative Drafting Workbench.

This icon is added to the Projections toolbar provided the Sheet Metal workbench is present.

7. Choose the **xy plane** in the Sheet Metal specification tree.
The unfolded view is displayed with the bends axes and limits.

Eventually, the Drafting sheet looks like this:





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Basic Tasks

The Basic Tasks section explains how to create and modify various kinds of features.

Managing the Default Parameters
Creating walls
Extruding
Isolating Walls
Creating Rolled Walls
Creating Bends
Creating Swept Walls
Unfolding
Pockets
Stamping
Patterning
Corner Relief
Creating Corners
Creating Chamfers
Mapping Elements
Saving As DXF
Reference Elements

Managing the Default Parameters

This section explains and illustrates how to use or modify various kinds of features. The table below lists the information you will find.



Using Sheet Metal Design assumes that you are in a CATPart document.



Edit the parameters: select the **Parameters** tab et the wall thickness and bend radius values.



Modify the bend extremities : select the **Bend Extremities** tab and choose a predefined bend type.



Define the bend corner relief: select the **Bend Corner Relief** tab and choose a predefined corner relief type.



Define the bend allowance: select the **Bend Allowance** tab and define the allowance value (K factor).

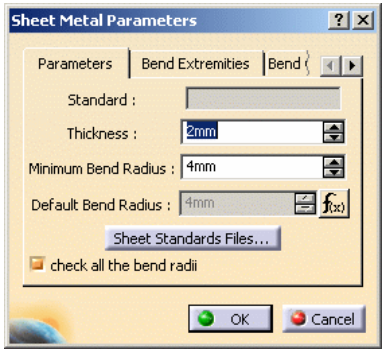
Up Next

Editing the Sheet and Tool Parameters

This section explains how to change the different sheet metal parameters needed to create your first feature.

1. Click the **Sheet Metal Parameters** icon .

   The Sheet Metal Parameters dialog box is displayed.



2. Change the **Thickness** if needed.

3. Change the **Minimum Bend Radius** if needed.
   The Minimum Bend radius defines the minimum internal radius allowing the creation of a bend. If the bend has a radius smaller than the one defined here, it will not be created.

You can set the value to 0 to create bend with no radius. If using the DIN standard, the KFactor automatically sets to 0 as well.

This option only relevant with the NEW SheetMetal Design workbench.

4. Change the **Default Bend Radius** if needed.
   The Default Bend Radius corresponds to the internal radius and is linked by default to the creation of the bends.

Convention dictates that the inner angle between the two elements is used to define the bend.
It can vary from 0° to 180° exclusive. This angle is constant and the bend axis is rectilinear.

This option is only relevant with the NEW SheetMetal Design workbench.

5. Click **OK** to validate the Sheet Metal Parameters.

The Standard field displays the Standard to use with the part, if implemented. The name of this standard file is defined in a Design Table.

When the **Check all the bend radii** button is checked, and you click OK in the Sheet Metal Parameters dialog box, existing bend radii are checked and a list displays flanges or bends that do not use the minimum Bend Radius value as defined in step 3. Therefore, they will not be modified.

Parameters can be defined in a Design Table. To do so, press the **Sheet Standards Files...** button to access to the company defined standards, if need be. For more information, refer to the Customizing Standard Files section.

All parameters hereafter, or only some of them, can be defined in this Design Table:

Sheet Metal Parameters	**Column associated in the Design Table**	**Definition**
Standard in Sheet Metal Parameters	SheetMetalStandard	sheet reference name
Thickness	Thickness	sheet thickness
Minimum Bend Radius	MinimumBendRadius	minimum bend radius
Default Bend Radius	DefaultBendRadius	default bend radius
K Factor	KFactor	neutral fiber position
Radius Table	RadiusTable	path to the file with all available radii

Whenever both Radius Table and Default Bend Radius are defined in the Design Table, only the Radius Table will be taken into account for the bend creation.

Standard Names For Holes	**Column associated in the Design Table**	**Definition**
Clearance Hole	ClearanceHoleStd	path to the Clearance Hole Standard file
Index Hole	IndexHoleStd	path to the Index Hole Standard file
Manufacturing Hole	ManufacturingHoleStd	path to the Manufacturing Hole Standard file
Fastener Hole	FastenerHoleStd	path to the Fastener Hole Standard file
Standard Names For Stamps	**Column associated in the Design Table**	**Definition**
Flanged Hole	ExtrudedHoleStd	path to the Flanged Hole Standard file
Bead	BeadStd	path to the Bead Standard file
Circular Stamp	CircularStampStd	path to the Circular Stamp Standard file
Surface Stamp	SurfaceStampStd	path to the Surface Stamp Standard file
Flanged CutOut	FlangedCutoutStd	path to the Flanged CutOut Standard file
Curve Stamp	CurveStampStd	path to the Curve Stamp Standard file

When a parameter refers to a path, another sub-Design Table will be associated to the corresponding feature.

1. Here is an example for the use of a bend allowance table:

Main Sheet Metal Parameters Design Table

	A	B	C	D
1	SheetMetalStandard	Thickness (mm)	RadiusTable	
2	AG 3412	2	RadiusTableForThickness2.xls	
3	AG 3824	4	RadiusTableForThickness4.xls	
4				
5				

Radius Table For Thickness 2

This table defines available all bend radii for a thickness of 2 mm. A design table will be created on the Default Bend Radius of the Sheet Metal Parameters and on the Radius of each bend.

	A	B
1	BendRadius (mm)	BendTable
2	1	BendTableT2R1.xls
3	2	BendTableT2R2.xls
4	4	BendTableT2R4.xls
5		

Bend Table for Thickness 2 and Bend Radius 1

Whenever a bend is created, a radius table will be associated. If the configuration "Bend Radius = 1mm" is selected, a new design table (the Bend Table) will be created from BendTableT2R1.xls in order to compute the bend allowance.

According to the open angle, the bend deduction will be read in the Allowance column or interpolated if necessary.

	A	B	C
1	OpenAngle (deg)	Allowance (mm)	
2	25	-1.942	
3	90	-3.644	
4	160	-0.534	
5			

2. Here is an example for the use of a hole standard file:

Main Sheet Metal Parameters Design Table

	A	B	C	D	E	F	G	H	I
1	SheetMetalStandard	Thickness (mm)	MinimumBendRadius (mm)	DefaultBendRadius (mm)	KFactor	ClearanceHoleStd	FastenerHoleStd	IndexHoleStd	ManufacturingHoleStd
2	AG 3412	2	0	4	0.36	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls
3	ST 5123	3	1	5	0.27	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls
4									
5									

Hole Standard

Whenever a hole is created, a design table will associate its radius with a standard name.

	A	B
1	StandardName	Diameter (in)
2	M1	0.39
3	M2	0.65
4	M3	0.89
5	M4	0.25
6	M5	0.56
7		

3. Here is an example for the use of a stamp standard file:

Main Sheet Metal Parameters Design Table

	A	B	C	D	E	F	G	H	I
1	SheetMetalStandard	SurfaceStampStd	CurveStampStd	CircularStampStd	BeadStd	BridgeStd	FlangedCutoutStd	ExtrudedHoleStd	StiffeningRibStd
2	AG 3412	SurfaceStampAG3412.xls	CurveStampAG3412.xls	CircularStampAG3412.xls	BeadAG3412.xls	BridgeAG3412.xls	FlangedCutoutAG3412.xls	ExtrudedHoleAG3412.xls	StiffeningRibAG3412.xls
3	ST 5123	SurfaceStamp5123.xls	CurveStampST5123.xls	CircularStampST5123.xls	BeadST5123.xls	BridgeST5123.xls	FlangedCutoutST5123.xls	ExtrudedHoleST5123.xls	StiffeningRibST5123.xls
4									

Whenever a stamp is created, a design table will associate its dimension with a standard name.

- **Surface Stamp**

	A	B	C	D	E
1	StandardName	Height (mm)	Angle (deg)	Radius1 (mm)	Radius2 (mm)
2	S1	6	80	2	2
3	S2	8	75	1	1
4					

- **Curve Stamp**

	A	B	C	D	E	F
1	StandardName	Height (mm)	Length (mm)	Angle (deg)	Radius2 (mm)	Radius1 (mm)
2	C1	4	6	75	1	1
3	C2	5	7	80	1	1
4						

- **Circular Stamp**

	A	B	C	D	E	F
1	StandardName	Diameter (mm)	Height (mm)	Angle (deg)	Radius1 (mm)	Radius2 (mm)
2	C1	10	6	80	2	2
3	C2	20	5	85	1	1
4						

- **Bead**

	A	B	C	D	E
1	StandardName	SectionRadius(mm)	EndRadius(mm)	Height(mm)	Radius1 (mm)
2	Bead04	4	6	4	2
3	Bead09	9	10	5	3
4					

- **Bridge**

	A	B	C	D	E	F	G	H
1	StandardName	Angle (deg)	PositioningAngle (deg)	Length (mm)	Radius1 (mm)	Radius2 (mm)	Height (mm)	Width (mm)
2	B1	80	5	10	2	2	6	5
3	B2	75	4	12	1	1	8	6
4								

- **Flanged Cutout**

	A	B	C	D
1	StandardName	Height (mm)	Angle (deg)	Radius1 (mm)
2	F1	6	80	2
3	F2	8	75	1
4				

- **Extruded Hole**
  (or **Flanged Hole** in the NEW SheetMetal Design workbench)

	A	B	C	D
1	StandardName	Diameter (mm)	Height (mm)	Angle (deg)
2	D20	20	6	90
3	D15	15	6	70
4				

- **Stiffening Rib**

	A	B	C	D	E
1	StandardName	Angle (deg)	Radius2 (mm)	Length (mm)	Radius1 (mm)
2	S1	80	2	30	2
3	S2	75	1	35	2
4					



Modifying the Bend Extremities

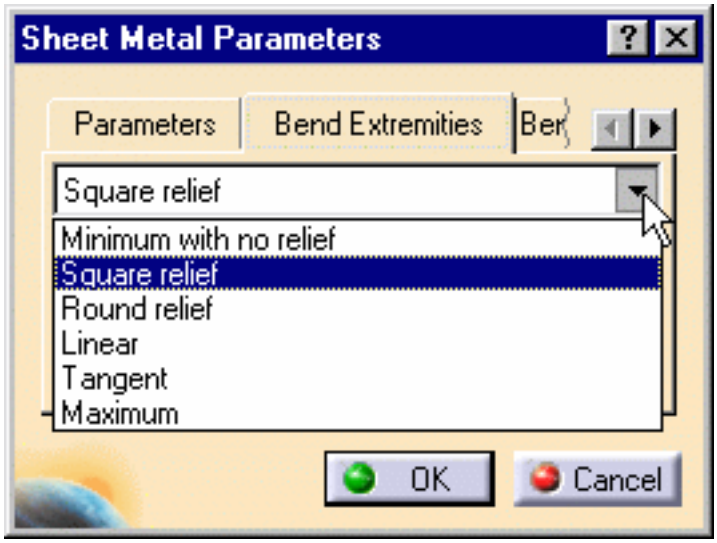
This section is only available with the SheetMetal Design products.

This section explains how to change the bend extremities.

1. Click the **Sheet Metal Parameters** icon .
   The Sheet Metal Parameters dialog box is displayed.

   The second tab concerns the bend extremities.

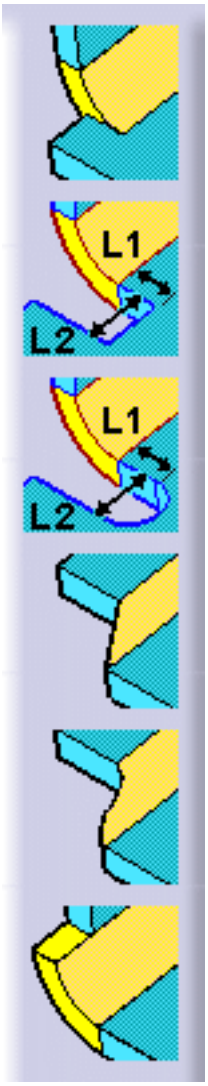
A combo box displays the six possible axial relimitations for the straight bend:



These options can also be accessed through the pop-up button:

- **Minimum with no relief**: the bend corresponds to the common area of the supporting walls along the bend axis.
- **Square relief**: a square relief is added to the bend extremity. The L1 and L2 parameters can be modified if need be.
- **Round relief**: a round relief is added to the bend extremity. The L1 and L2 parameters can be modified if need be.
- **Linear**: the unfolded bend is split by two planes going through the corresponding limit points (obtained by projection of the bend axis onto the edges of the supporting walls).

- **Tangent**: the edges of the bend are tangent to the edges of the supporting walls.
- **Maximum**: the bend is calculated between the furthest opposite edges of the supporting walls.



Defining the Bend Corner Relief

This section is only available with the SheetMetal Design products.

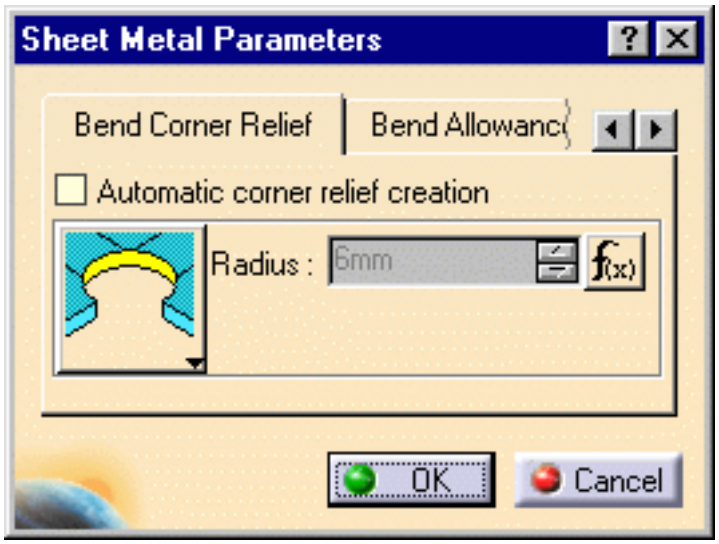
This section explains how to change the bend corner relief.

Open the CornerRelief01.CAPTPart model from the samples directory.
Within the **Tools -> Options -> General -> Parameters -> Knowledge** tab, check the **Load Extended Language Libraries** option.

1. Click the **Sheet Metal Parameters** icon .

   The Sheet Metal Parameters dialog box is displayed.

   The third tab concerns the bend corner relief.



By default, no corner relief is created when a bend is created. Check the **Automatic corner relief creation** option to activate this creation every time a bend is created.

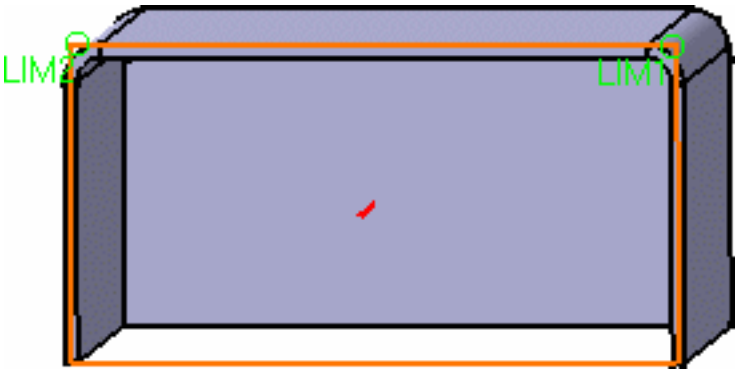
Three corner relief types are available. Select the icon corresponding to the requested type:

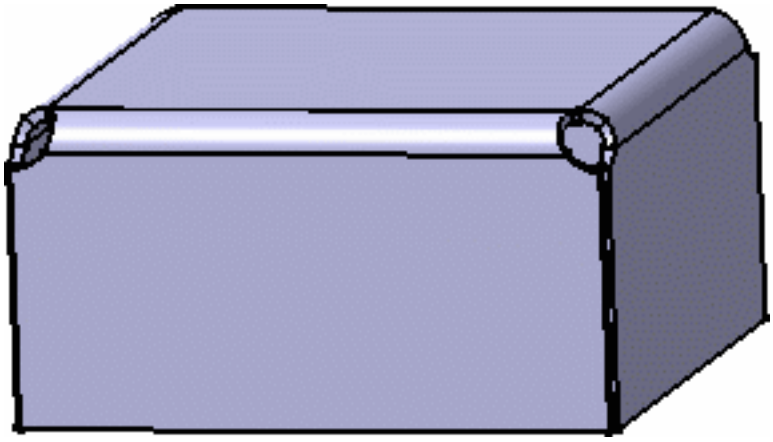
-  : **square**: the square corner relief is created using the bend limits. Its dimensions are defined by the width of the unfolded bends.
-  : **circular**: its center is located at the intersection of the bend axes. For that option, a radius is proposed by default. It is equal to the bend radius + the thickness. To change it:
  - Select **Formula -> Deactivate** from the contextual menu of the input field and enter a new value,
  - click on the f(x) button and entering a new formula.

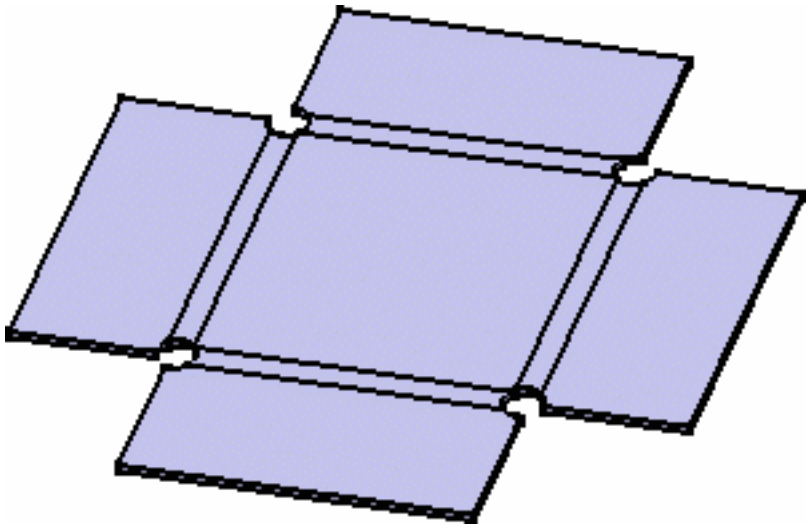
-  : **triangular**: the triangular relief is created from the intersection point of the inner bend limits towards the intersection points of the outer bend limits with each wall.

The corner relief is not previewed during its creation.

The corner relief is taken into account in the unfolded view.







- For better result, you should select the Maximum Bend Extremities option when creating corner relief.
- These parameters are applied to each corner relief created or to be created, except to those with that have been redefined, or the locally defined corner relieves.



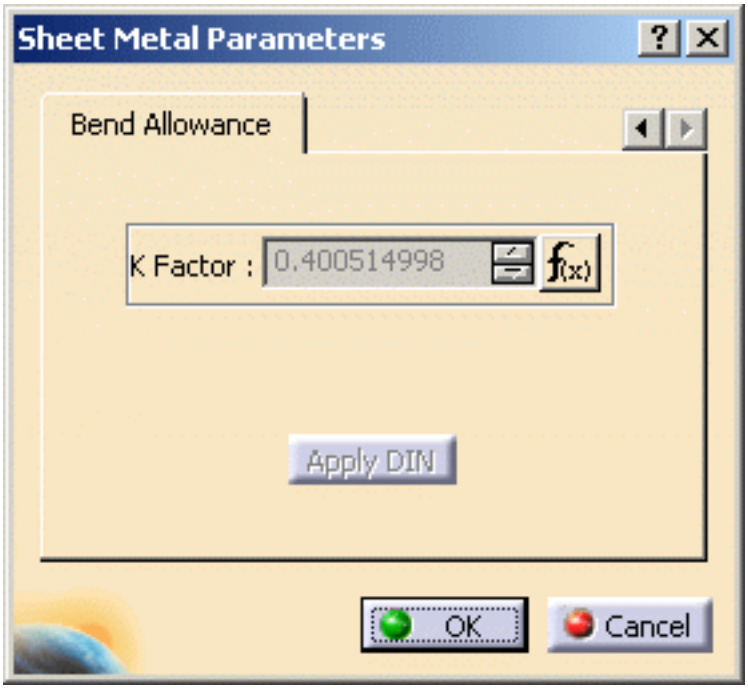
Computing the Bend Allowance

This section explains the calculations related to folding/unfolding operations.

1. Click the **SheetMetal Parameters** icon .

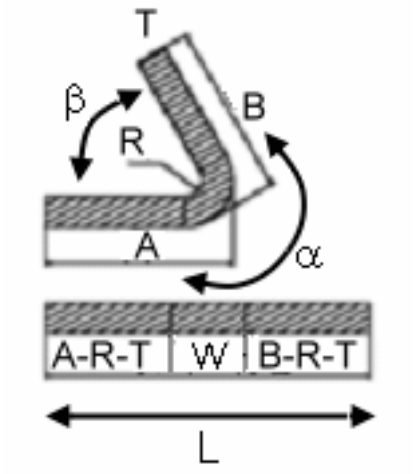
   The Sheet Metal Parameters dialog box is displayed.

   The fourth tab concerns the bend allowance.

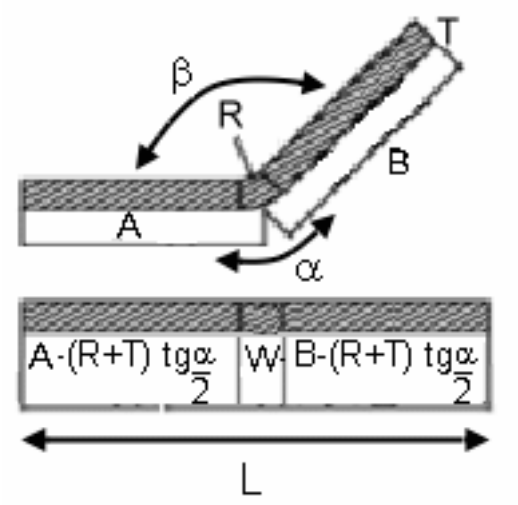


- **Bend Allowance**

The bend allowance corresponds to the unfolded bend width.



bend < 90°



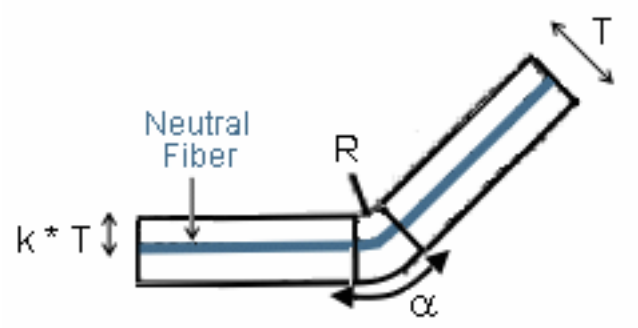
bend > 90°

L is the total unfolded length
A and **B** the dimensioning lengths as defined on the above figure. They are similar to the DIN definition.

- **K Factor**

Physically, the neutral fiber represents the limit between the material compressed area inside the bend and the extended area outside the bend. Ideally, it is represented by an arc located inside the thickness and centered on the bend axis.

The K factor defines the neutral fiber position:



W = α * (R + k * T)

where:

W is the bend allowance
R the inner bend radius
T the sheet metal thickness
α the inner bend angle in radians.

If β is the opening bend angle in degrees:

α = π * (180 - β) / 180

When you define the sheet metal parameters, a literal feature defines the default K Factor and a formula is applied to implement the **DIN** standard. This standard is defined for thin steel parts. Therefore the K Factor value ranges between 0 and 0.5.

The DIN definition for the K factor slightly differs.

W = α * (R + k' * T/2)
Therefore **k' = 2 * k** and ranges from 0 to 1.

This formula can be deactivated or modified by right-clicking in the K factor field and choosing an option from the contextual menu. It can be re-activated by clicking the Apply DIN button. Moreover, the limit values can also be modified.

When a bend is created, its own K Factor literal is created.
Two cases may then occur:

a. If the Sheet Metal K Factor has an activated formula using the default bend radius as input parameter, the same formula is activated on the bend K Factor replacing the default bend radius by the local bend radius as input.
b. In all other cases, a formula "equal to the Sheet Metal K Factor" is activated on the local bend K Factor.
   This formula can also be deactivated or modified.

- **Bend Deduction**

When the bend is unfolded, the sheet metal deformation is thus represented by the bend deduction **V,** defined by the formula:

$$L = A + B + V$$

(refer to the previous definitions).

Therefore the bend deduction is related to the K factor using the following formula:

$$V = \alpha * (R + k * T) - 2 * (R + T) * \tan (\min(\pi/2,\alpha) / 2)$$

This formula is used by default. However, it is possible to define bend tables on the sheet metal parameters. These tables define samples: thickness, bend radius, open angle, and bend deduction. In this case, the bend deduction is located in the appropriate bend table, matching thickness, bend radius, and open angle. If no accurate open angle is found, an interpolation will be performed.
When updating the bend, the bend deduction is first computed using the previously defined rules. Then the bend allowance is deduced using the following formula:

$$W = V + 2 * (R + T) * \tan (\min(\pi/2,\alpha) / 2)$$


When the bend deduction is read in the bend table, the K factor is not used.



Creating Walls

This section explains and illustrates different methods to create walls.


Create walls from an existing Part: click the icon, and select a face of a wall


Create a wall from a sketch: use the sketcher to define the profile, and set the material side.


Create a wall tangent to another one: select a profile coincident with an existing wall, and select the wall to which it should be tangent


Create a wall from an edge: select a wall edge, set the height, limits, angle, then the sketch and material sides

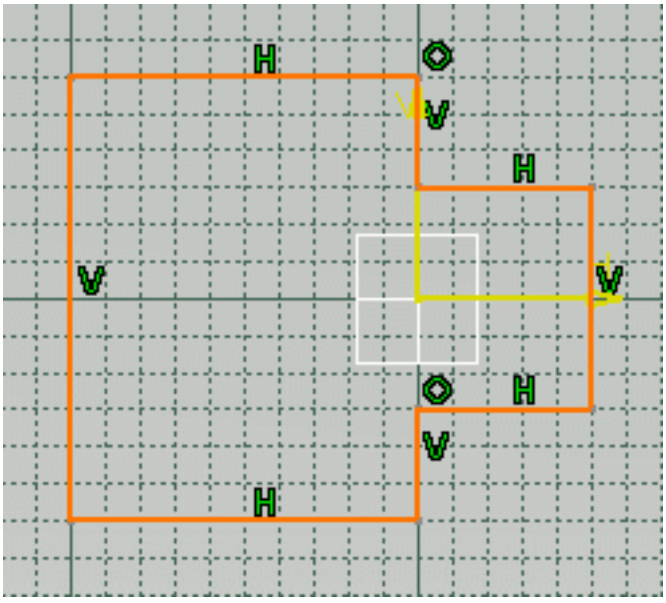
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Creating Walls from a Sketch

This task shows how to create a wall from a sketch.

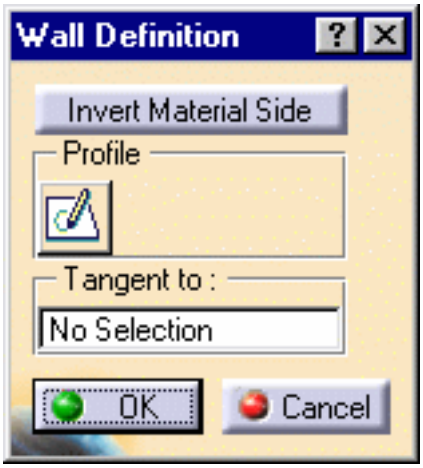
You must be in the Sheet Metal Workbench, with a .CATPart document open, and you must have defined the Sheet Metal parameters.
Set the sketcher grid to **H = 100mm** and **V = 100mm**, using the **Tools -> Options**, **Mechanical Design -> Sketcher**, **Sketcher** tab.

1. Click the **Sketcher** icon  then select the **xy plane**.
2. Select the **Profile** icon .
3. Sketch the contour as shown below:



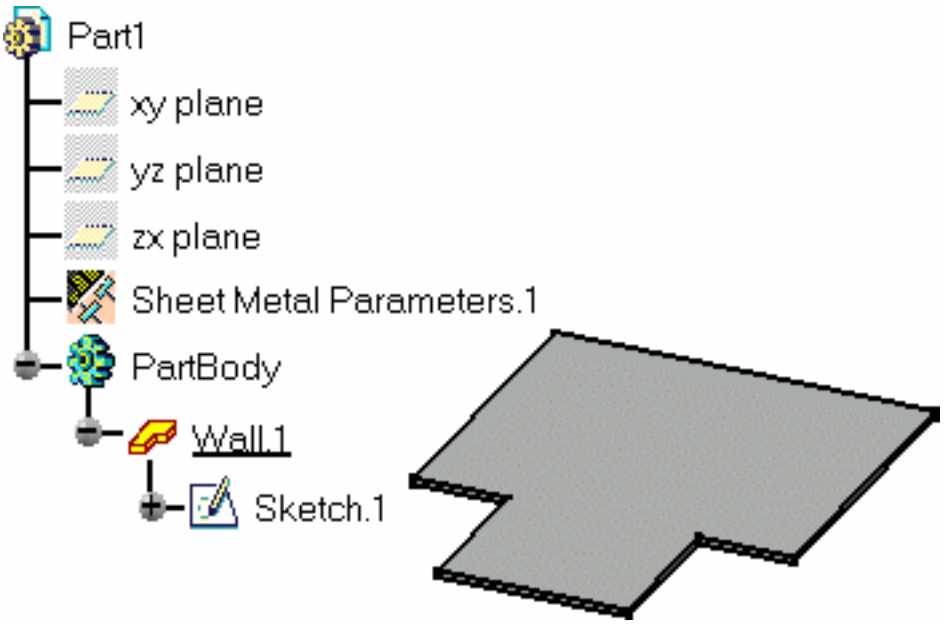
4. Click the **Exit workbench** icon  to return to the 3D world.

5. Click the **Wall** icon .
The Wall Definition dialog box opens.



By default, the Material Side is set to the top.

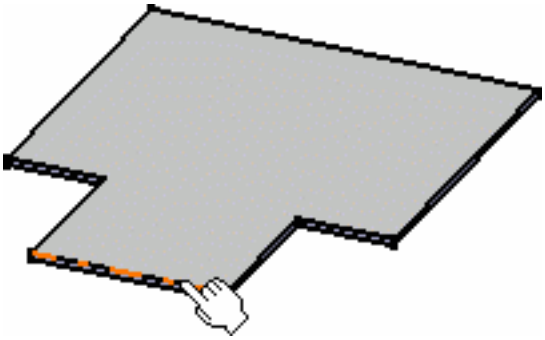
6. Click OK.
The **Wall.1** feature is added in the specification tree.



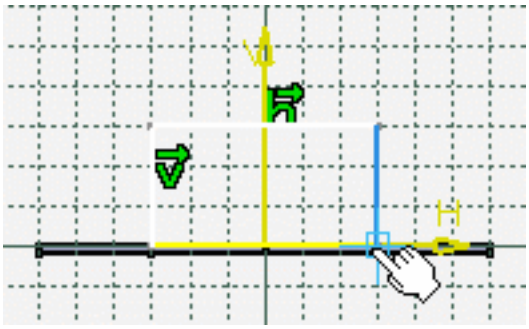
The first wall of the Sheet Metal Part is known as the Reference wall.

- Click the sketcher icon from the Wall Definition dialog box, if you wish to directly edit the selected sketch. When exiting the sketcher, you then go back to the wall creation step, without having to reactivate the Wall icon.

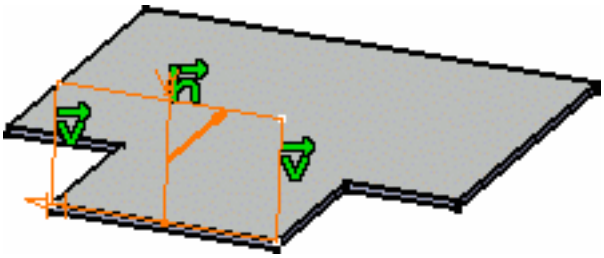
This is also very useful if you have selected an edge from a wall and clicked the **Wall** icon .



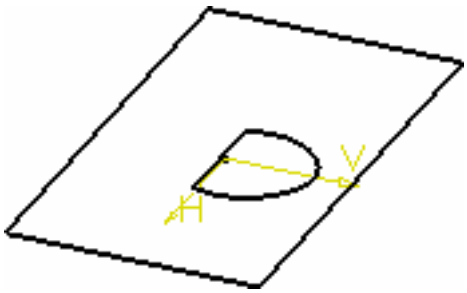
In this case, the sketcher is automatically activated and the plane defined as being the selected edge's plane.



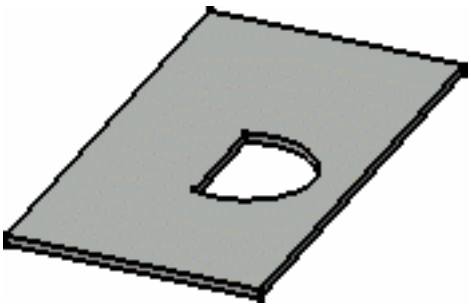
You can then directly draw a sketch, then exit the sketcher and return to the wall creation step.



- You can directly create a wall with a hole, by selecting a sketch with an inner contour (the contours must not intersect):



Sketch with inner contour



Resulting wall

Note however, that the emptied area is part of the wall and is not a separate cutout that can be edited.



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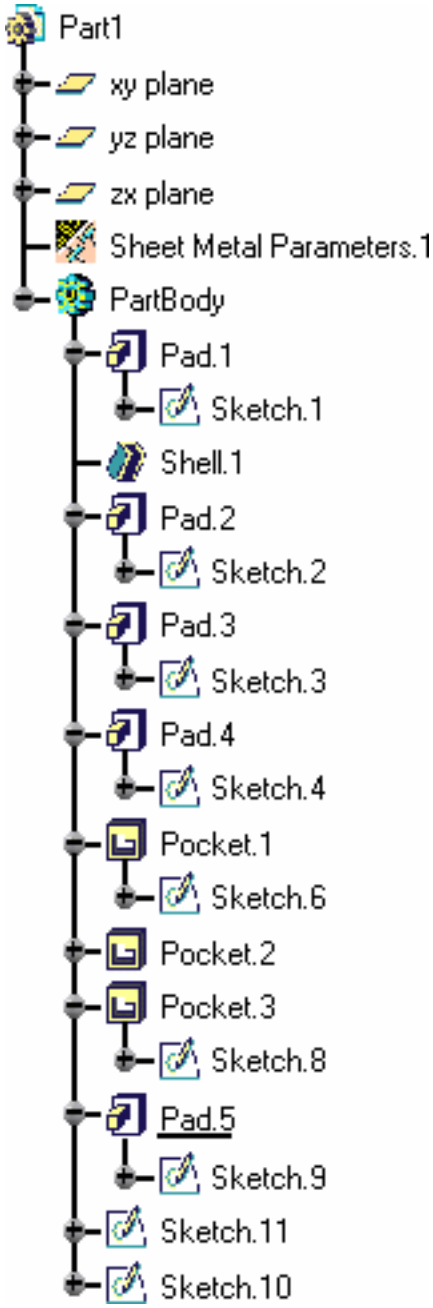
Creating Walls From an Existing Part

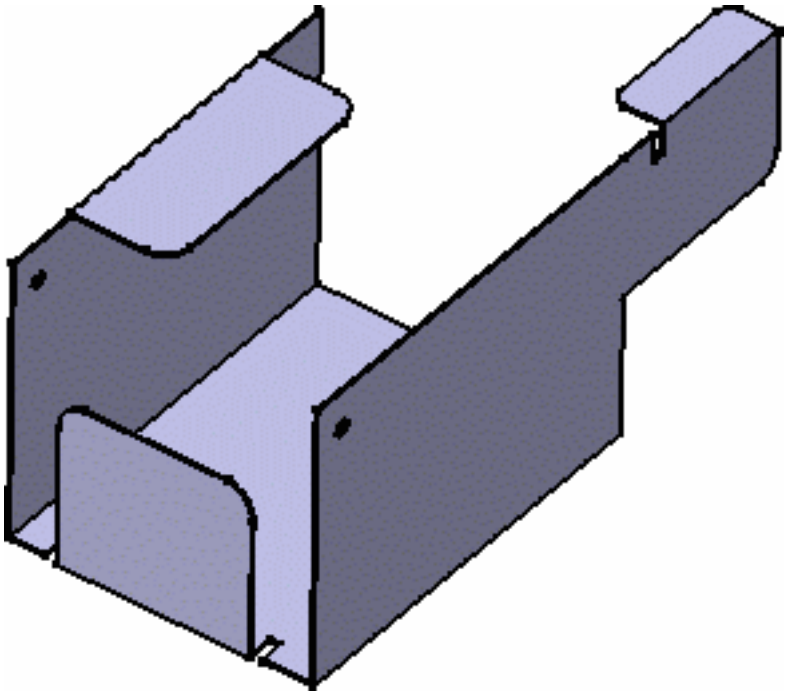
This functionality is only available with SheetMetal Design.

This task illustrates how to create a Sheet Metal part using an existing Part, that is recognizing the thin part shapes of the Part as created using the Part Design workbench or from a CATIA Version 4 Solid for example.

Open the WallScenario1.CATPart document from the samples directory.

The document contains a Part created in the Part Design workbench and it looks like this:

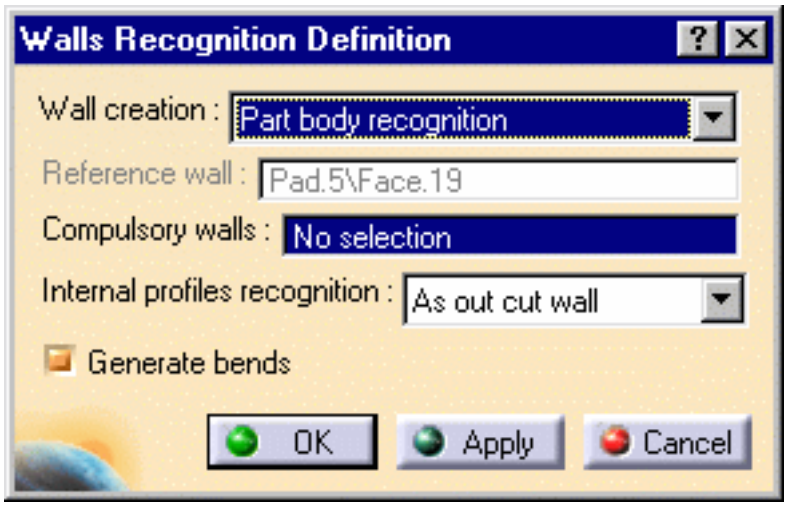




1. Click the **Walls Recognition** icon .

2. Click any face to be the reference wall.

The Walls Recognition Definition dialog box is displayed.



3. Choose the **Wall creation** mode:

- **Part body recognition**: the whole solid is processed and walls are created wherever possible
- **Only selected faces**: only explicitly selected faces of the solid are processed and the corresponding walls are created.

The **Reference wall** is indicated in the Walls Recognition Definition dialog box for information only (it is grayed out).

4. Select faces as the **Compulsory walls**.
   These are faces from which the walls are to be generated when there might be an ambiguity. For example, if the initial part is a box, you will need to select two opposite inner faces and outer faces on the other two sides of the box, in order to avoid overlapping when generating the walls.

5. Set the Internal profiles recognition mode:

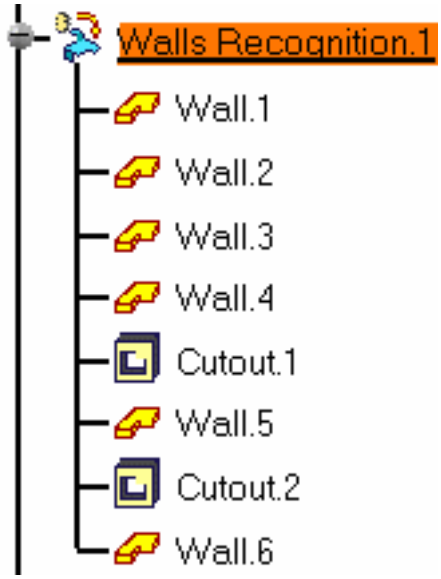
- **As cut out wall**: generates walls with inner contours (no cutout feature is generated)
- **One cut out by wall**: regardless of how many pockets there are on a face of the solid, only one cutout feature is generated per wall
- **One cut out by profile**: for each inner contour on the sketch-based solid, a cutout feature is generated
- **None**: whether there are pockets on the solid faces, or not, no cutout feature is created in the resulting SheetMetal features.

The **Generate Bends** check button allows the automatic creation of bends as the walls are being created, wherever applicable.

6. Click Apply.

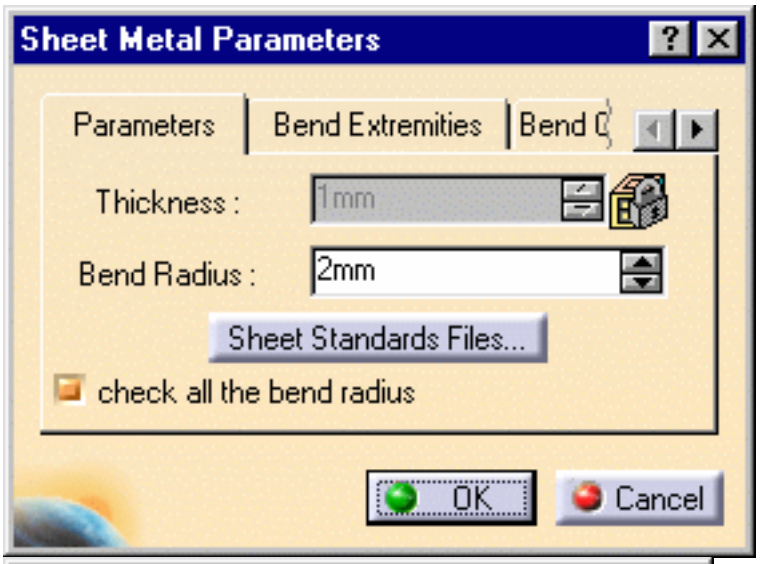
   The walls are generated from the Part Design geometry.

   The **Walls Recognition.1** feature is added to the tree view.



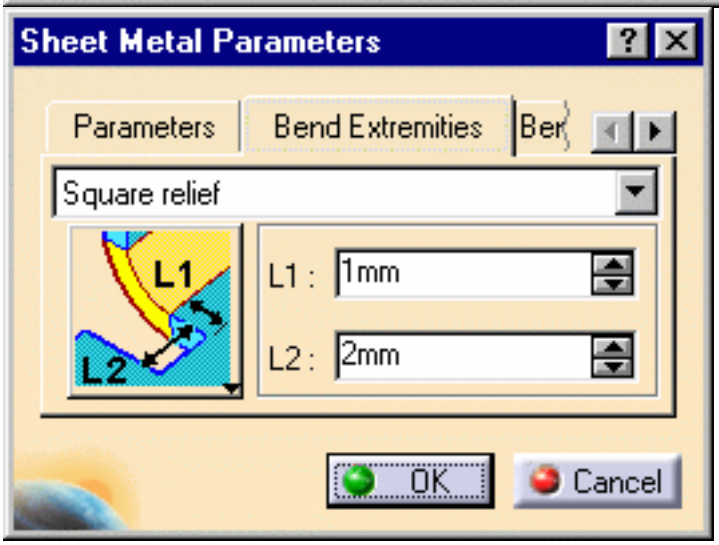
At the same time, the Sheet Metal parameters are created, deduced from the Part geometry.

7. Select the icon  to display the sheet metal parameters:



- the **Thickness** is equal to 1mm
- the **Bend radius** is twice the thickness value

- the **Bend Extremities** field is set to Square relief.



You can modify a few of these parameters
The **Thickness** parameter cannot be modified because it is based, like the bend extremities and radius, on the initial solid geometry . However you can modify these parameters (bend radius and bend extremities) to be taken into account for sheet metal features other than the "recognized" ones.
The bend allowance, being used to unfold the part, and the corner relief affect all features, and therefore can be edited even for "recognized" features.

8. Click OK in the Sheet Metal Parameters when all parameters have been redefined where needed.

The solid is now a Sheet Metal part. All the features are displayed in the specification tree.

- Once the solid has been converted to a Sheet Metal part, you can create bends as with any other Sheet Metal part, or modify/add Sheet Metal features to complete the design.
- Uncheck the **Generate Bends** button, if you do not wish bends to be created automatically.



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Creating Tangent Walls

This task shows how to create a wall tangent to a planar part of another wall or of a flange. This capability will then allow this tangent wall to be seen when unfolding the part, even though there is no bend linking it to its tangent support, provided this support is unfoldable too.

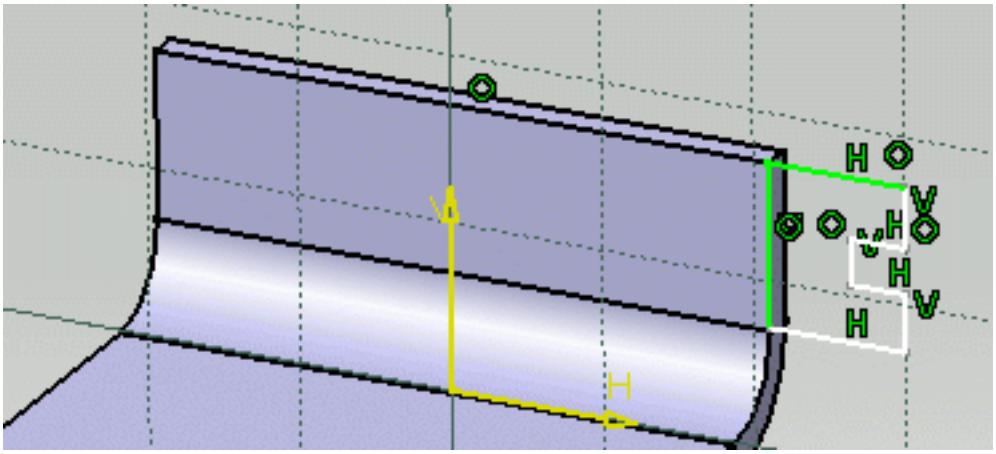
Open the TangentWall1.CATPart document from the samples directory.

1. Select a face of an existing wall and click the Sketcher icon .
   Here we selected the flange's planar face.

2. Select the **Profile** icon and sketch the contour as shown below.

You can also select Sketch.2 from the specification tree.

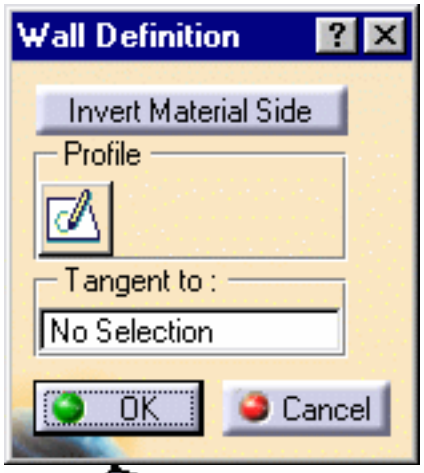
3. Using the Constraint Defined in Dialog box icon, set coincidence constraints between the edges where the support and sketch are to coincide.



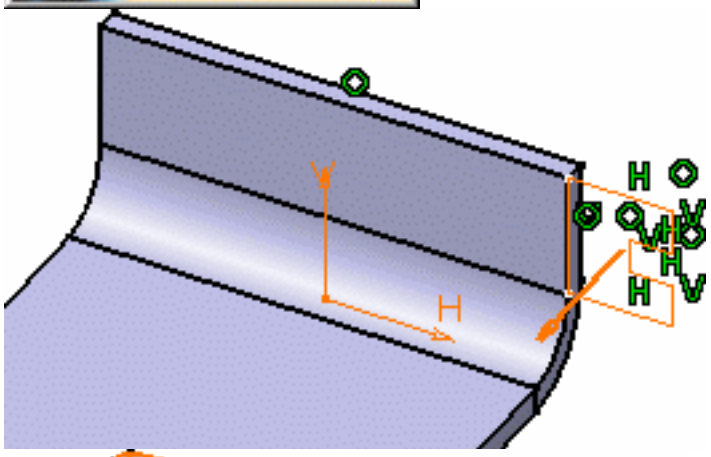
4. Click the **Exit workbench** icon to return to the 3D world.

5. Make sure the sketch is still active, then click the **Wall** icon .

   The Wall Definition dialog box opens.

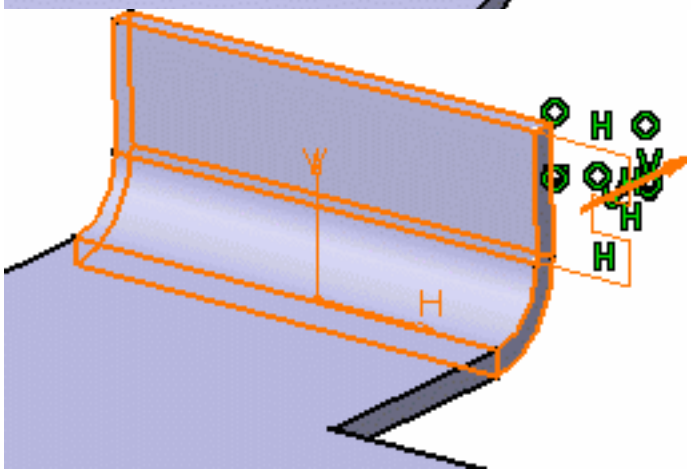


Note the orientation of the wall to be created.



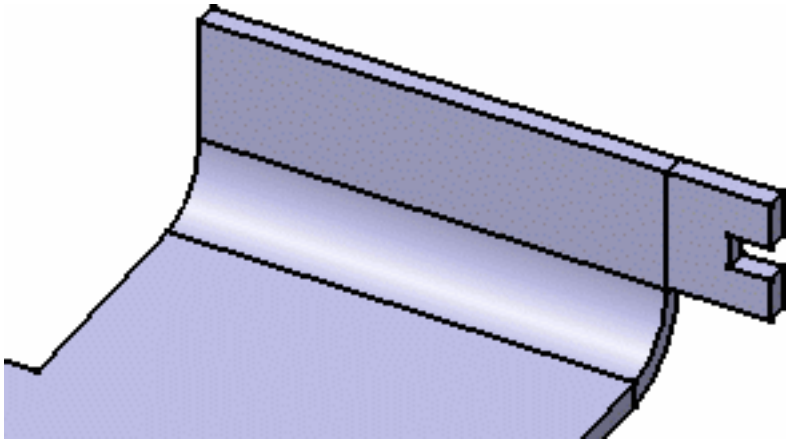
6. Click inside the **Tangent to** field, then select the wall to which the new wall has to be tangent.

   Here, you need to select the planar face of the flange.

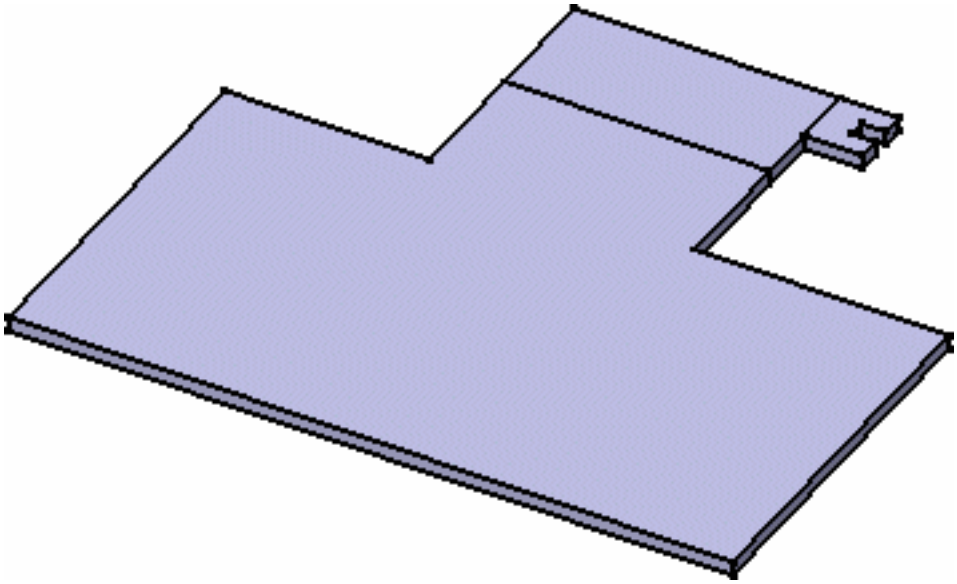


Note that the orientation automatically changes to conform to the material orientation already defined on the support wall.

7. Click **OK**.
   The wall is created and a **Wall.xxx** feature is added in the specification tree.



8. Click the **Unfold** icon .
   The tangent wall is unfolded as a wall linked by a bend to another wall, would be.



If you want to create the wall on a wall that cannot be unfolded, the system issues a warning and prevents you from creating the tangent wall.

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Creating Walls From An Edge

This task shows how to create walls from edges of an existing wall. This function is used to create a box in an easy and quick way from an existing reference wall.

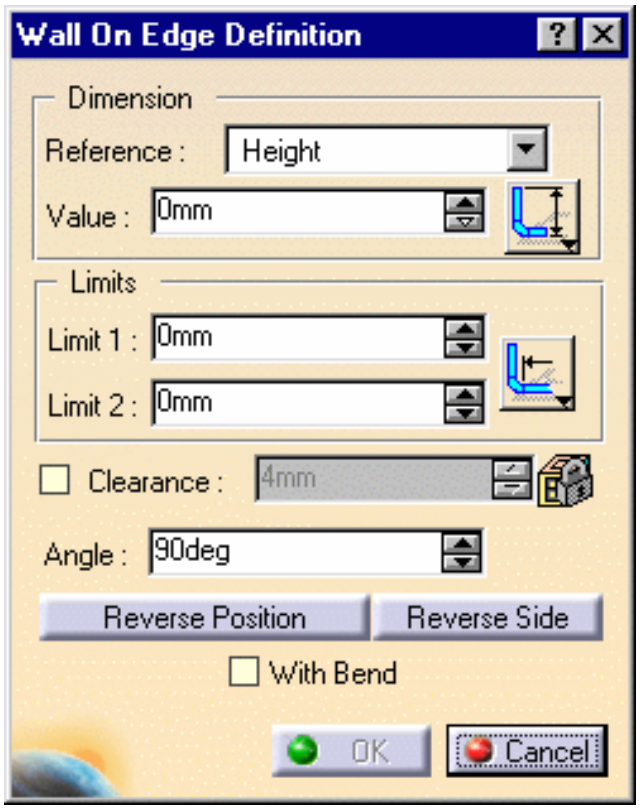
At least one wall must already exist.

Open the Wall1.CATPart document from the samples directory.

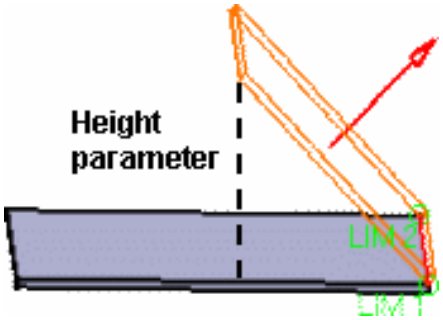
1. Click the **Wall on Edge** icon and select an edge of an existing wall.

   The Wall On Edge Definition dialog box is displayed together with a preview of the wall.

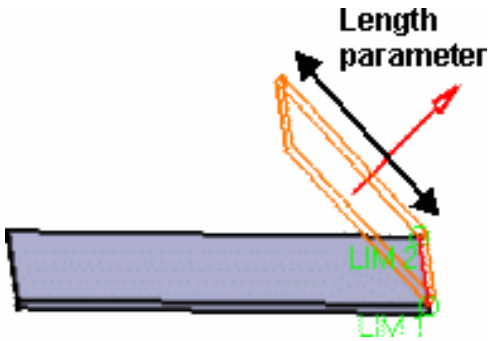
2. Define the type of wall to be created by specifying the Reference, that is:

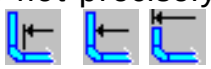


- the **Height** of the wall: that is the orthogonal projection from the top of the wall on edge to the reference wall.
  Select the icon to define the height of the wall from the bottom of the reference wall or the icon to define the height of the wall from the top of the reference wall.



- the **Length** of the wall: that is the absolute value of the wall on edge without bend.

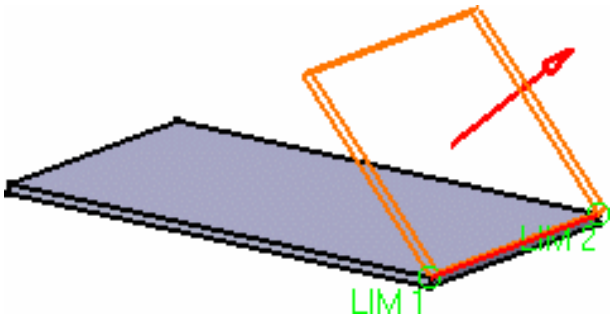


- the limits of the wall: Limit 1 and Limit 2. These texts only indicate on which side a given limit is. They are not precisely on the limit spots. The actual locations of the limits are defined with the  icons and an input distance that is taken into account respectively from the inner side of an existing bend, the inner side of an existing wall or the outer side of an existing wall.

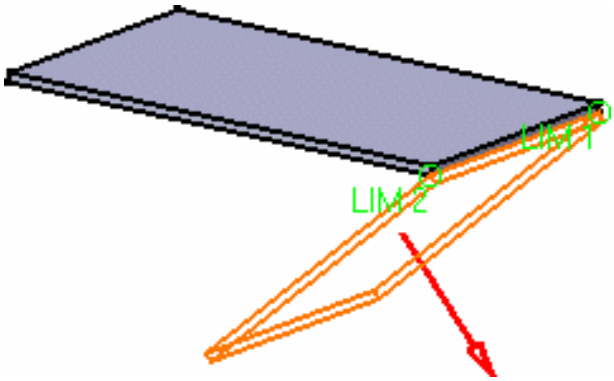
- the angle of the wall: by default it is perpendicular to the plane containing the edge. You may modify it according to your needs. it is updated dynamically on the screen.

This preview gives information about:

- the Sketch Profile: by default, if you pick an edge on the top of the reference wall, the direction of the wall is upwards, if you pick an edge on the bottom of the reference wall the direction of the wall is downwards.

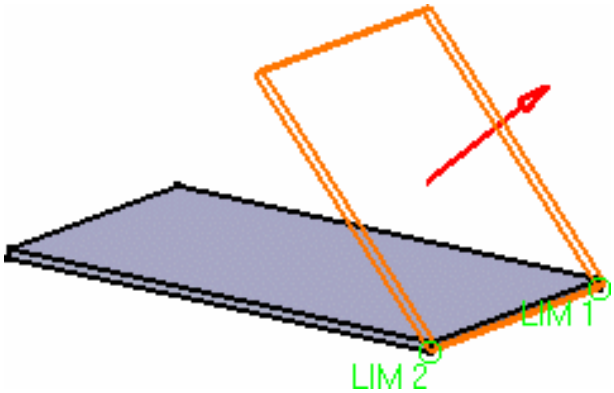


Preview with top edge selected

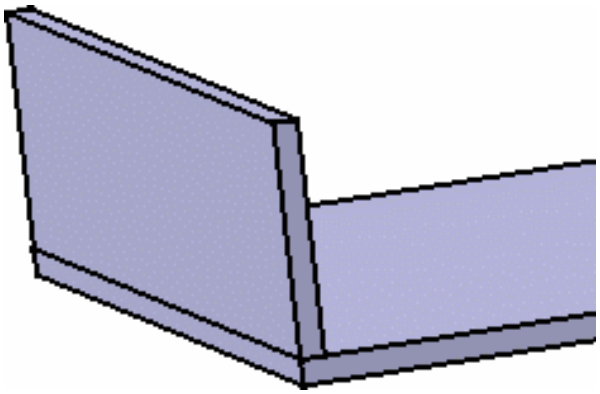


Preview with bottom edge selected

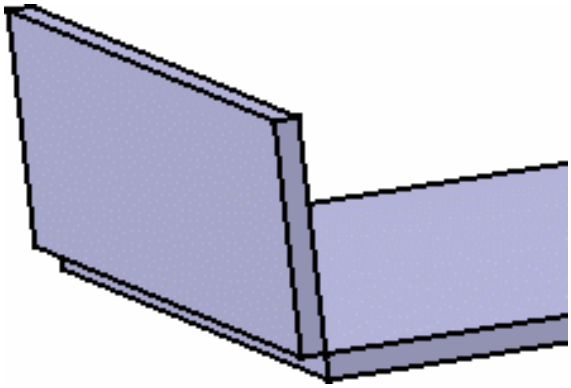
You can invert the sketch's position, and therefore the wall's, using the **Reverse Position** button.



Preview with bottom edge selected and sketch profile inverted



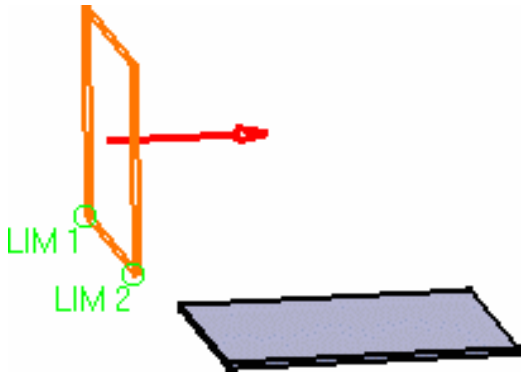
- the material side: by default the material side is proposed so as to keep a continuity with the reference profile.

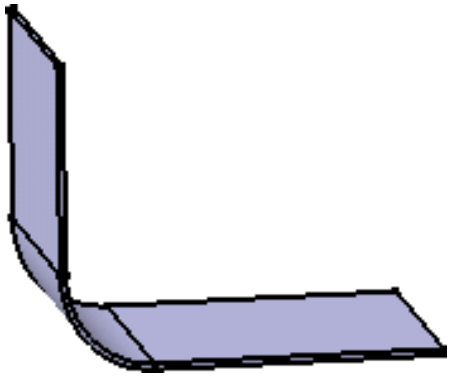


However, you can change it by clicking the red arrow or the **Reverse Side** button.

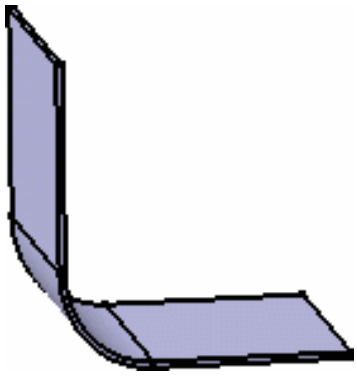
3. Check the **Clearance** option to offset the wall on edge from the selected edge.

   The entered value is the radius of the bend on this edge.





Wall on edge with clearance



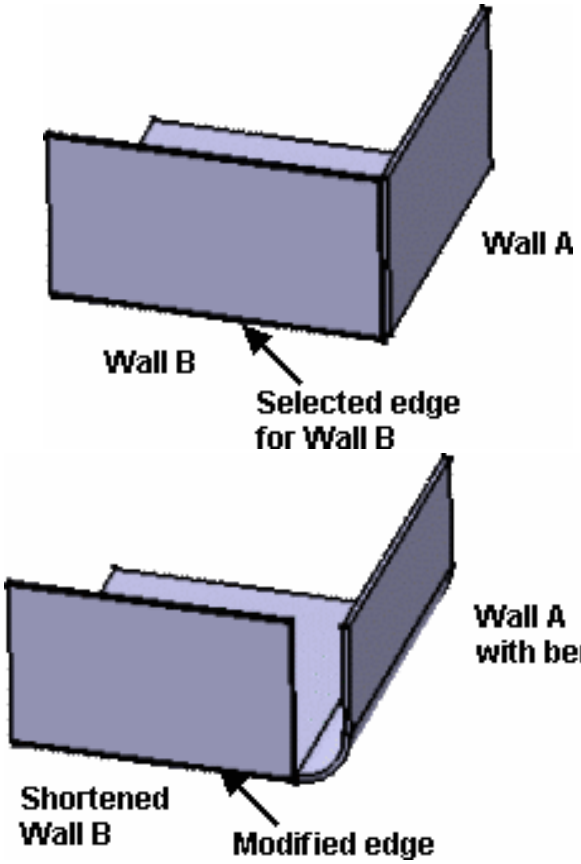
Wall on edge without clearance

Note that the reference wall remains unchanged when changing the bend radius value. It is the Wall on Edge's length that is affected.

4. You can also choose to create the wall with or without a bend by checking the **With Bend** option.

- If there is no wall from which a limit can be computed, the reference element is the edge of the reference wall.

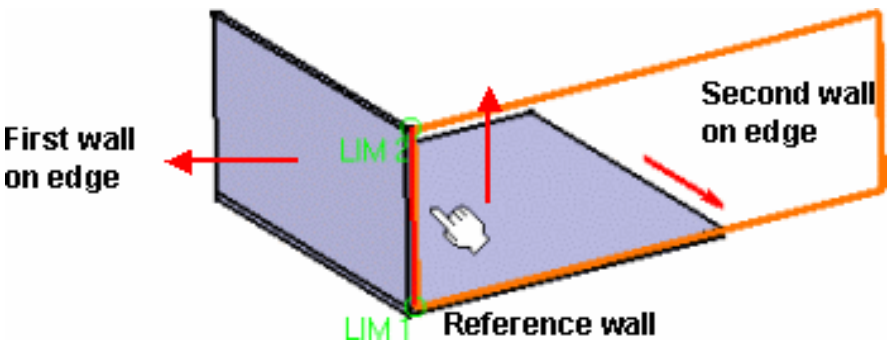
- A wall on edge is defined by the selected edge (reference edge). When the reference edge is modified, by adding any feature that shortens the edge (a bend to an adjacent wall on edge or a cutout for example) the wall on edge based on this reference edge is recomputed.
  To avoid this you may:
  - create the bend manually on the wall modifying the edge used as the reference to create the other wall
  - reorder the creation of walls to postpone the creation of the modifying feature



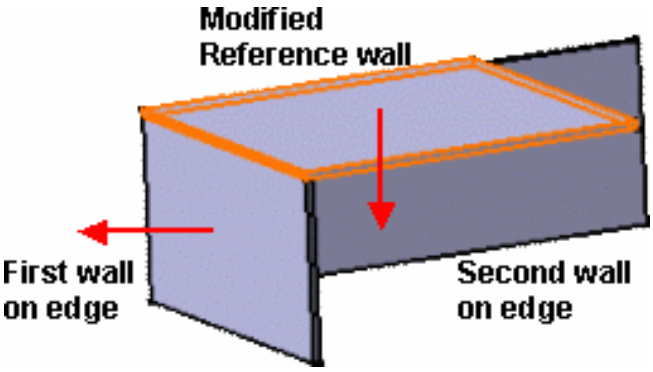
- Both limits are computed with the same reference icon.
- The bend is not previewed, even if the option **With Bend** is checked. However it will be created.
- The selected options are modal and will be proposed to create the next wall.

- Walls on edge being contextual features, if you break the profiles continuity by inverting the material side of a wall, you may have to manually re-specify all features based upon the modified one, even if they are not directly connected to the modified wall, in order to update the part afterwards.

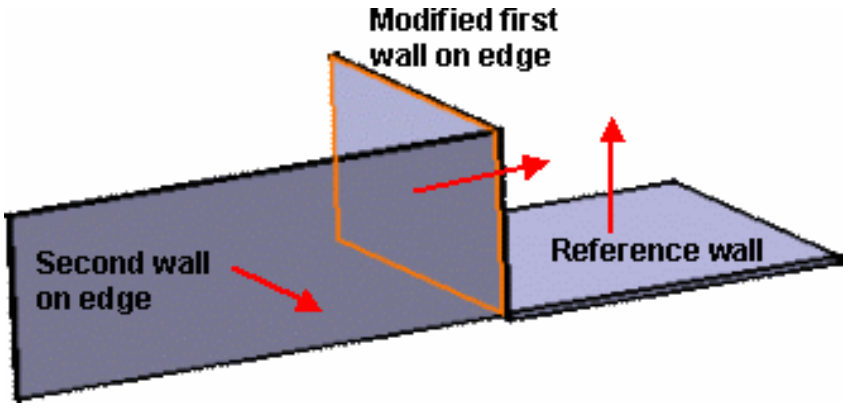
This is the case when creating a wall on edge from another wall on edge, for example:



- If you invert the material side of the reference wall on which the first wall on edge has been created, both walls on edge are relocated (as if you were flipping the geometry):



- If you invert the material side of the first wall on edge (and not the material side of the reference wall), the second wall on edge is relocated. Indeed, its specification being relative to the first wall on edge, when its input data (i.e. the edge selected on the first wall on edge) is modified, the second wall on edge is rebuilt at a new location.



5. Click OK in the Wall On Edge Definition dialog box.

A WallOnEdge.x element is created in the specification tree.

- A WallOnEdge.x element can be edited: double-click it in the graphic area or in the specification tree to display its creation dialog box and modify the parameters described above, including the edge from which it is created. However, the sketch of a wall on edge cannot be edited directly.
- Would you need to edit the sketch of a wall on edge, you have to isolate it first. See Isolating Walls.
- You can cut or copy and paste a wall on edge.
- If you cut and paste a wall on edge with children elements, these children elements are lost. This may result in update errors.
- You cannot undo an **Isolate** action after having modified the wall.
- Isolating a wall on edge erases all updating data.

Back Up

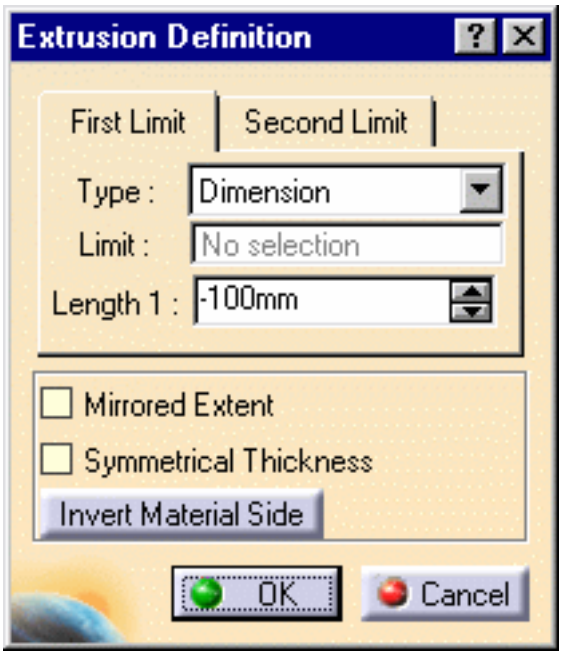
Extruding

This task explains how to create a wall by extrusion.

You must have defined the Sheet Metal parameters.
A model is available in the Extrude1.CATPart from the samples directory.

1. Click the **Extrusion** icon .
2. Select a sketch.

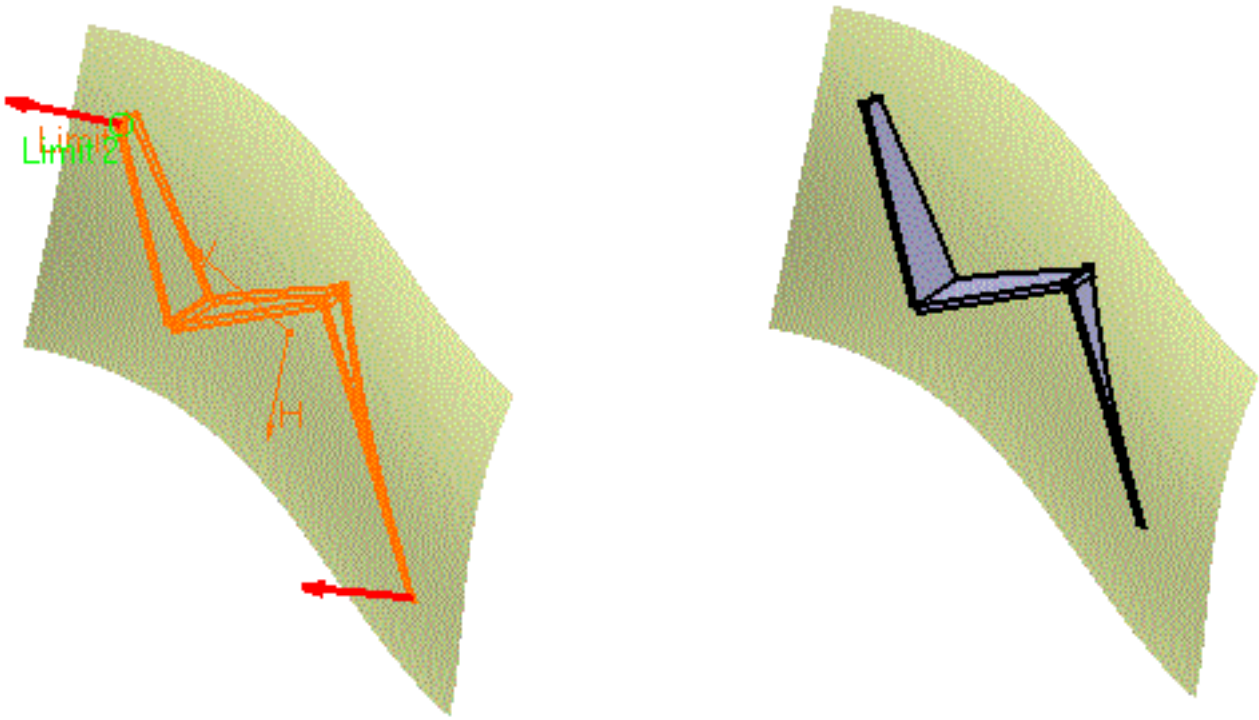
   The Extrusion Definition dialog box is displayed.



Several types of extrusion are available:

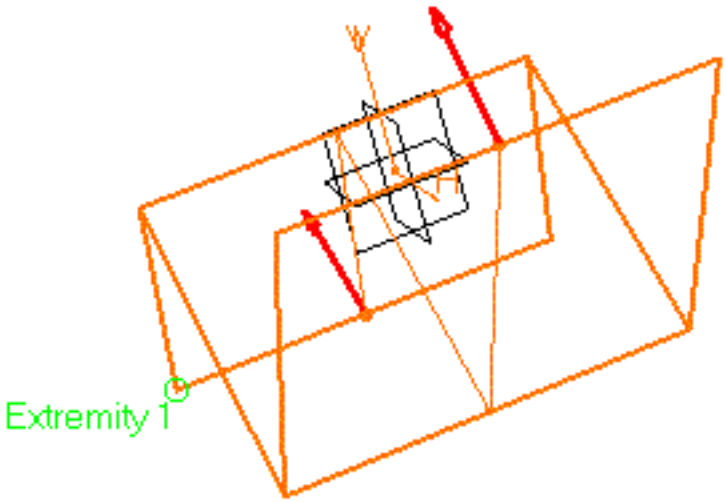
- **Dimension** : the requested input data are a sketch and a dimension,
- **Up to plane** or **Up to surface**: a plane or a surface are input as limit to the extrusion. These functions are used to create walls that are not rectangular.

3. Edit the **Length1** and **Length2** to set both extremities, for option **Dimension**.



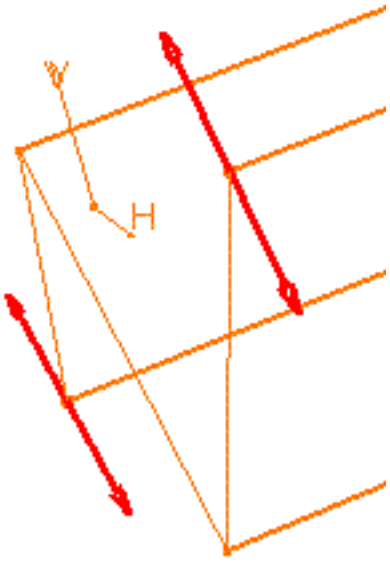
4. Define the options as needed:

- Check the option **Mirrored extent** to extent the material on both sides of the sketch. In that case, only **Length1** can be edited.

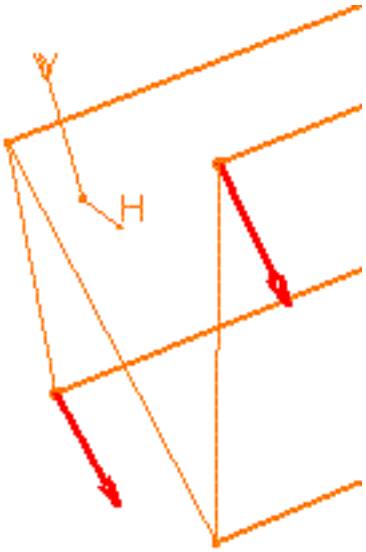


This option is only available if the type is set to **Dimension**.

- Check **Symmetrical thickness** to create thickness inside and outside the sketch.



- Push **Invert Material Side** to invert the direction of the creation of the material.

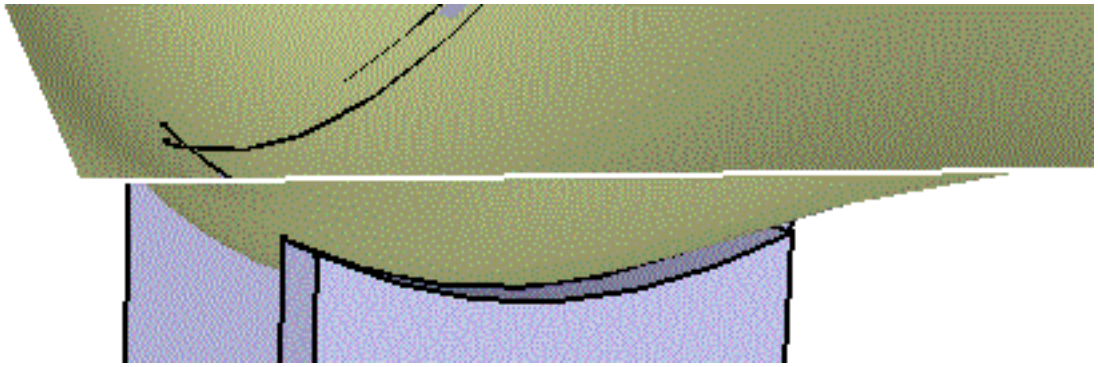


5. Click OK.

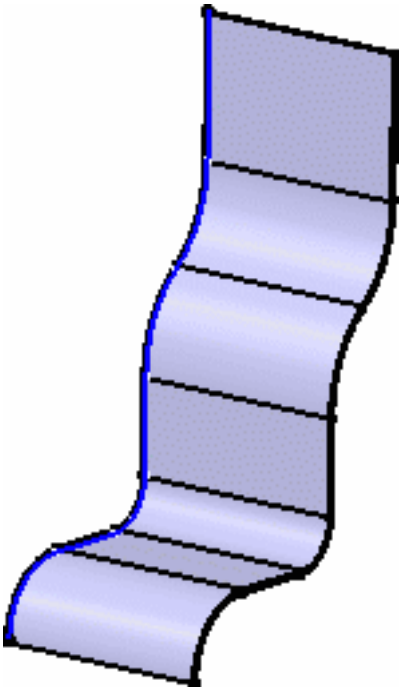
   The walls corresponding to the selected sketch are created according to the specified options, and added to the specification tree.

- When the extrusion is the first Sheet Metal feature of the Part, the reference wall is the first wall created based on the first segment of the sketch.

- For option **Up to Surface**, while the wall end that is limited by the surface has the shape of the surface, its thickness does not fit the surface. It is a "rectangular" polygon defined by the first edge that comes into contact with the surface.



- Such an extrusion can also be performed on a sketch made of lines and arcs of circle, provided there are no tangency discontinuities between the different elements.
  However, in this case:
  - the **Up to plane** or **Up to surface** capabilities are not available,
  - you cannot isolate such an extrusion,
  - if the element of the extruded sketch connected to the part is an arc of circle, the extrusion will not display in the unfolded view. To avoid this, prefer to create a User Defined Flange or remove the arc of circle of the extruded sketch and create a bend to connect the extrusion to the part.



- Extrusion walls can be edited and/or isolated.

The sketch may not be closed.

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Isolating Walls

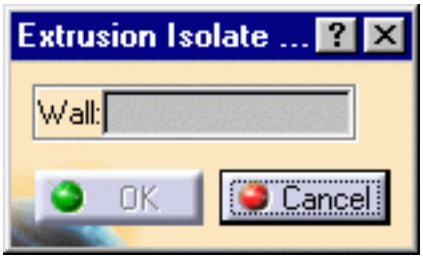
This task explains how to isolate a wall. This is possible in two cases:

1. after having created walls by extrusion (see Extruding in SheetMetal Design or Extruding in New SheetMetal Design)
2. after having created a wall on edge (see Creating Walls from an Edge).

You must have defined the Sheet Metal parameters.
A model is available in the Extrude2.CATPart from the samples directory.
If you use the New SheetMetal Design, open the NEWExtrude2.CATPart document.

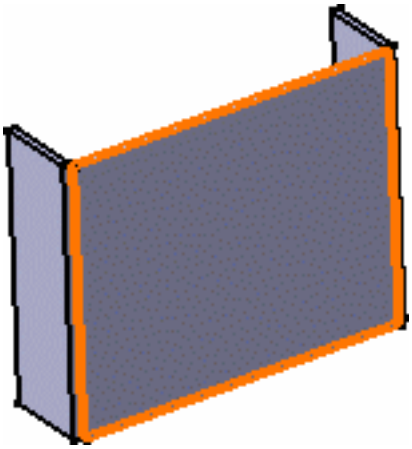
Isolating Extruded Walls

1. Right-click the Extrusion.1 feature and choose the **Extrusion.1 object -> Isolate** contextual menu item.



The Extrusion Isolate dialog box is displayed.

2. Select one of the wall of the extrusion to be isolated.

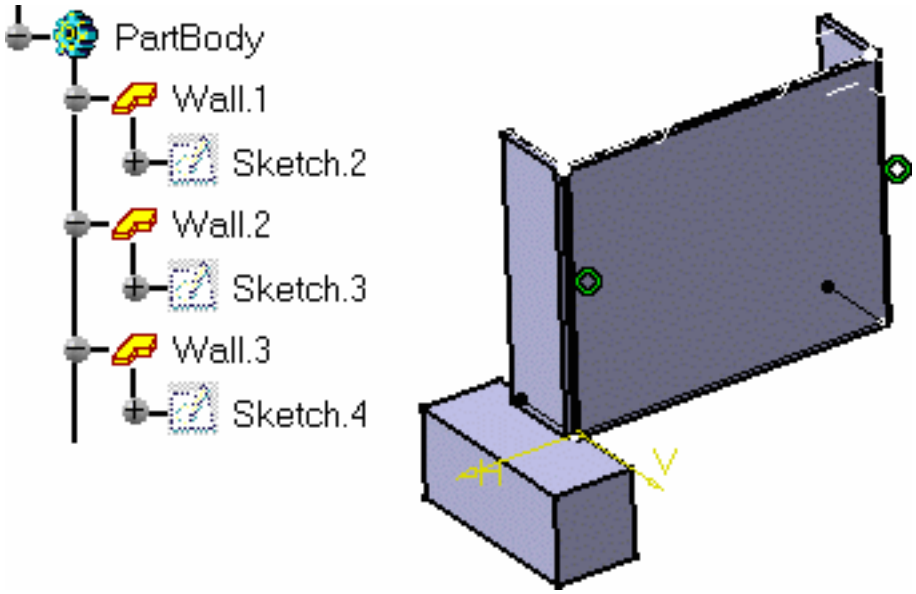


The selected wall is highlighted in the geometry.
This wall is the reference wall, meaning that it can be modified and the other walls will take the modification into account. On the other hand if the other walls are modified the reference wall is an anchoring wall, and modifications will be made around it.

The Extrusion Isolate dialog box is updated.

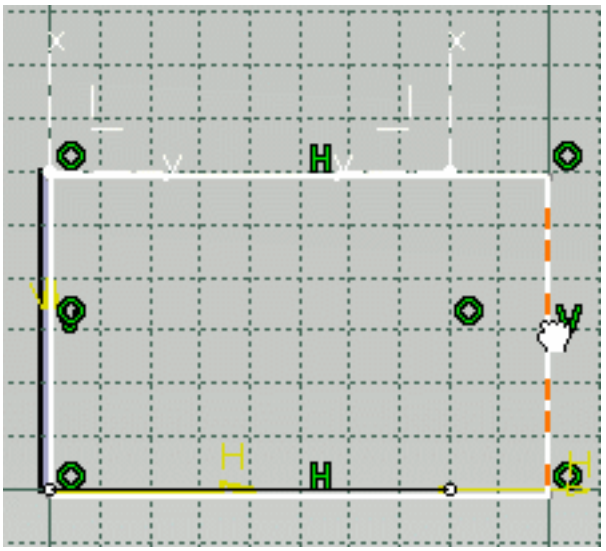
3. Click OK in the dialog box.

   The walls of the extrusion have been isolated, each with its own sketch. Yet coincidence constraints are automatically generated between the isolated walls.

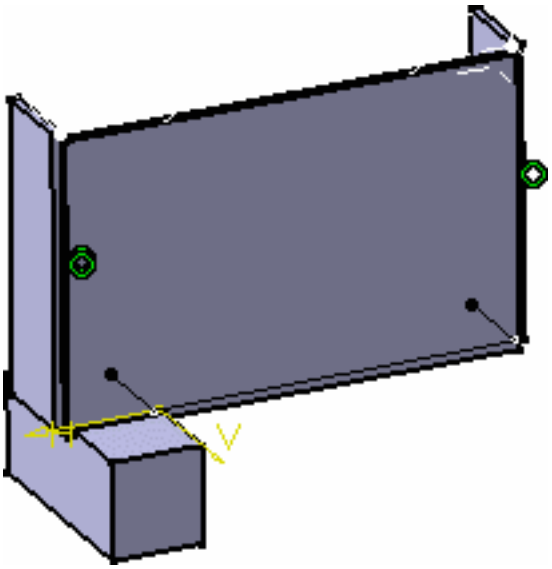


The extrusion's initial sketch is retained (Sketch.1 in the example above).

4. Double-click the sketch of the reference wall (here Sketch.3) and modify it by increasing its length.



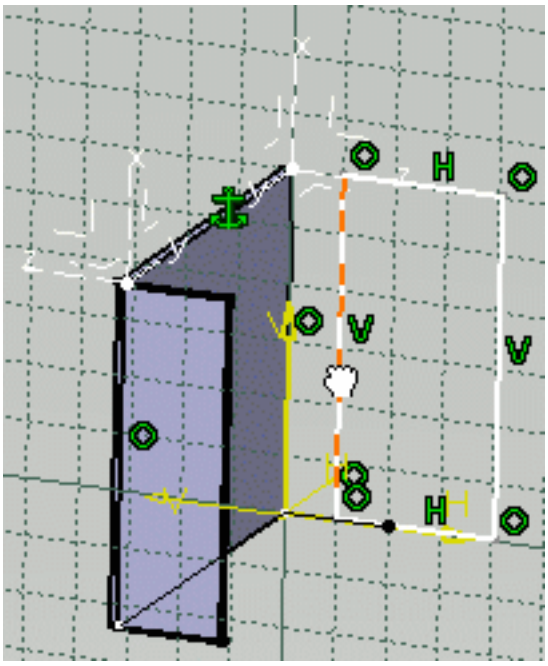
5. Exit the Sketcher using the **Exit** icon .



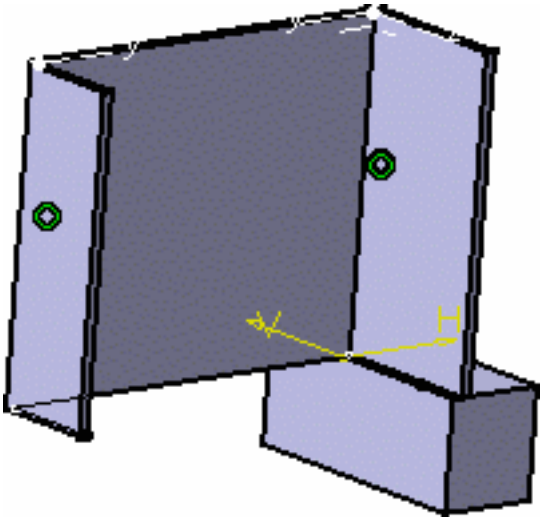
The Part is updated.

You can note that the wall that was adjacent to the pad, now lies on it, as it is the reference wall that was modified.

However, had you modified the sketch of the wall lying on the pad (Sketch.4), moving it further away from Wall.2 as shown here to the right, the updated pad would not take the gap between the walls into account.



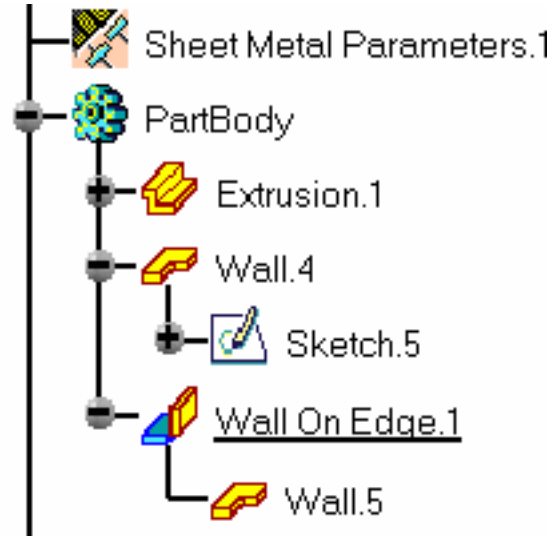
The resulting part looks like this (Wall.3 has been modified but still coincides with Wall.2):

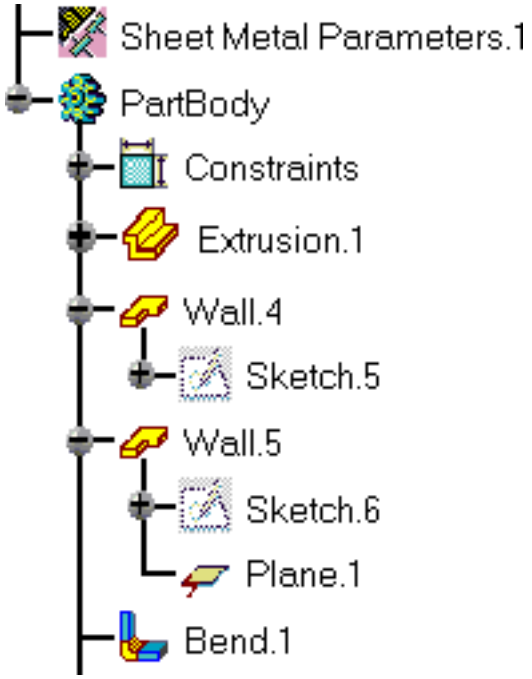


Isolating Walls on Edge

1. Right-click the wall on edge and choose the **Wall On Edge** contextual menu item.

   The wall on edge is then changed to a standard wall, as you can see from the specification tree.

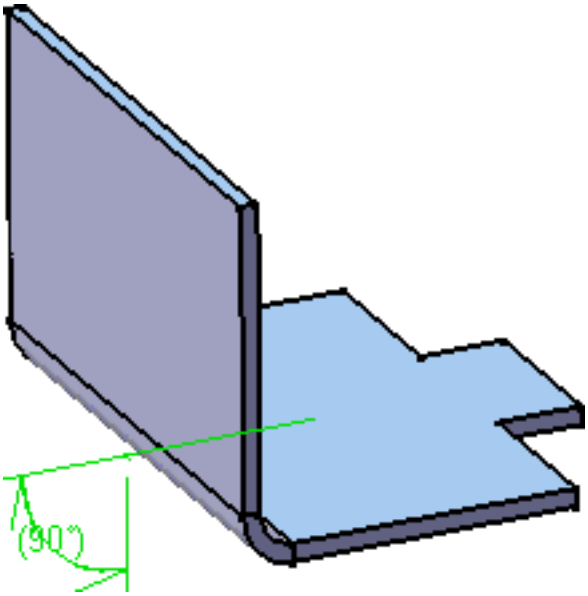




   You can then edit its sketch if needed.

   In the present case, the wall on edge had been created with a bend. Therefore when isolating this wall from the reference wall, the bend is created as a separate feature that can be edited as well.

   The angle value between the two walls is displayed for edition.



- You cannot undo an **Isolate** action after having modified the wall.
- Isolating a wall on edge erases all updating data.



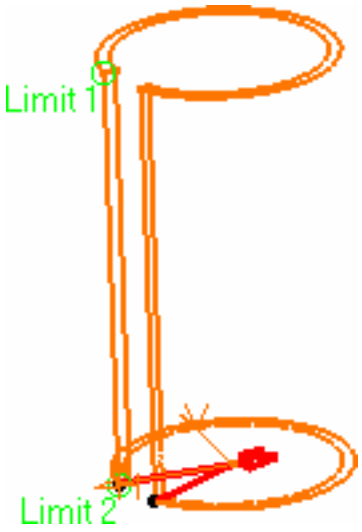
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Creating Rolled Walls

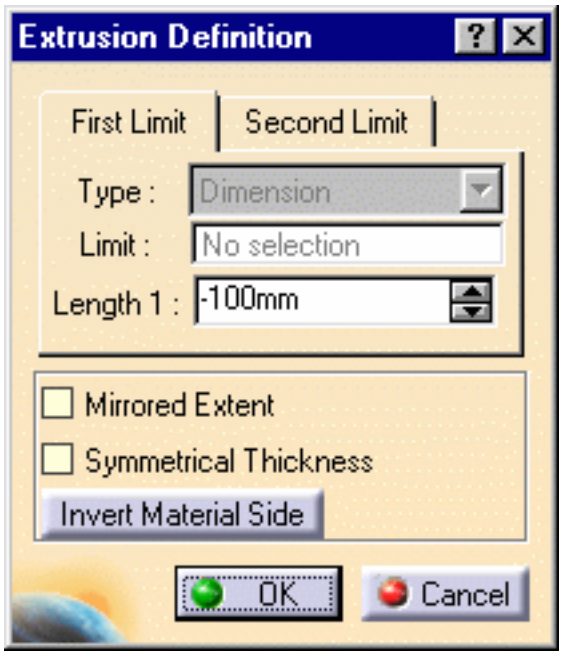
This task shows how to create rolled walls (such as pipes, open pipes with flange, etc.).

You must have defined the Sheet Metal parameters, and have a sketch available, in the form of an circular arc.
The part is available in the Rolledwall1.CATPart from the samples directory.

1. Click the **Extrusion** icon .
2. Select the circular sketch.

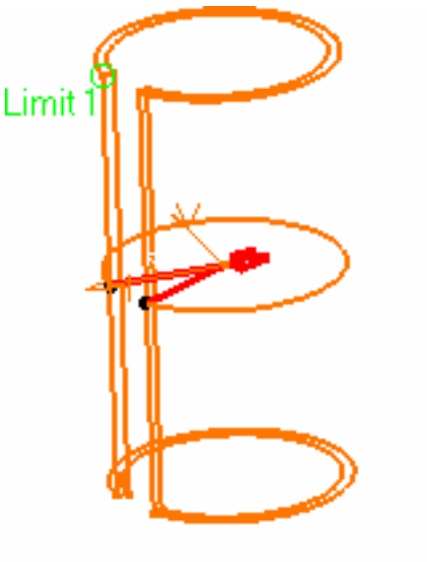


3. Make sure the type is set to **Dimension**.
4. **Length 1** and **Length 2** indicate the location of **Limit 1** and **Limit 2**.

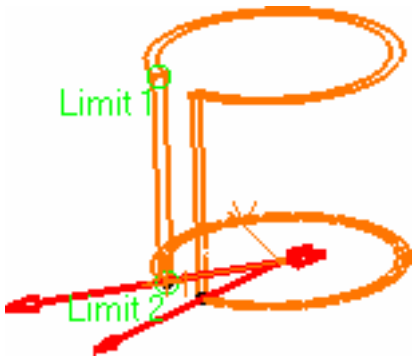


5. Define the options as needed (the length being down to -50 mm):

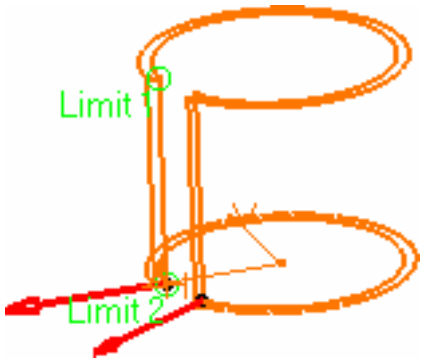
- Check the option **Mirrored extent** to extend the material on both sides of the sketch. In that case, only **Length1** can be edited.



- Check **Symmetrical thickness** to create thickness inside and outside the sketch.



- Push **Invert Material Side** to invert the direction of the creation of the material.



6. Click OK in the dialog box to create the rolled wall:



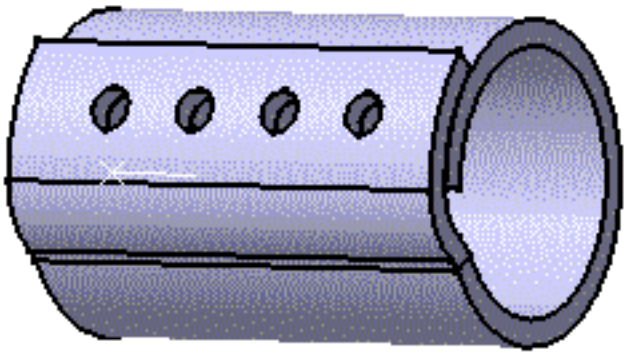
The rolled wall is a particular extrusion:

- the input sketch is either a circular arc or a closed circle,
- the creation type is always **Dimension**.

The sketch may be open. In that case, you can define where the opening should be.

The sketch may be closed. In that case, you have no control on the opening location.

The only operations you can combine with a rolled wall in a Sheet Metal model are flanges and cutouts.



No other elements (standard wall, bend,...) are allowed.

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Creating Bends on Walls

This section explains and illustrates different methods to create bends on walls. Bends can only be created between walls, whether manually or automatically, and not between any other Sheet Metal features, such as stamps for example.


Create bends from wall: select the two walls, set the bend radius value, the bend extremities, and specify the use of corner relief


Generate bends automatically: select the part, then a reference wall


Create conical bends: select the part, and choose a reference wall


Create flat bends: select a sketch, set the creation mode and limiting option, set the radius and angle in relation to the selected sketch.

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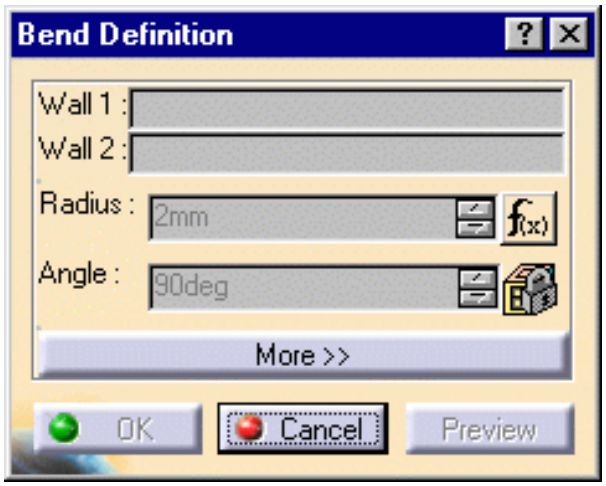
Manually Creating Bends from Walls

This task explains two ways to create bends between walls in the Sheet Metal part. These bends can be created on non-connex walls, and with a constant radius value.

Open the BendExtremities01.CATPart document from the samples directory.

1. Select the **Bend** icon .

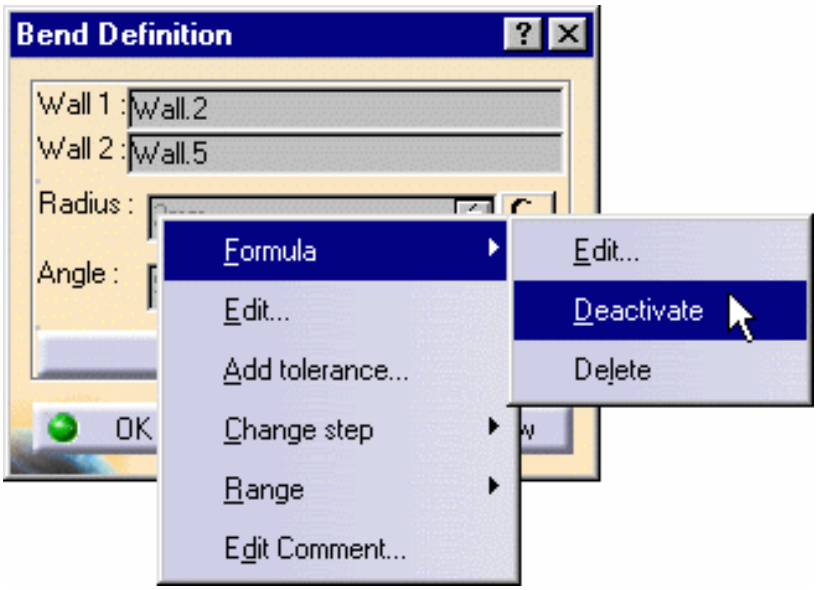
   The Bend Definition dialog box opens.



Note that the **Radius** field is in gray because it is driven by a formula when editing the sheetmetal parameters: at that time, you cannot modify the value.

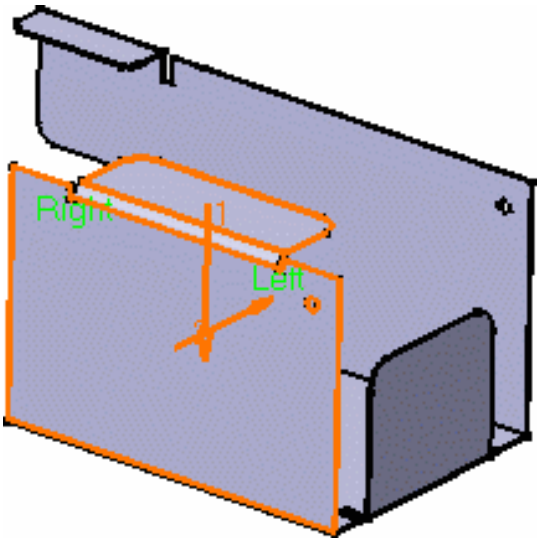
2. Select **Wall.2** and **Wall.5** in the specification tree.
   The Bend Definition dialog box is updated.

3. Right-click the **Radius** field and select **Formula -> Deactivate** from the contextual menu to change the value.

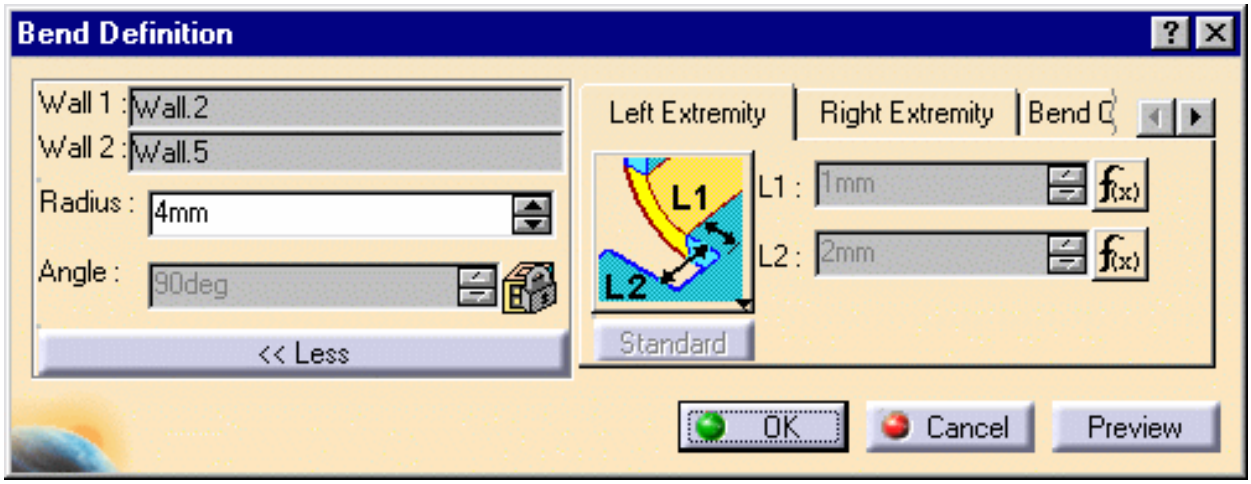


4. Enter 4mm for the Radius and click Preview.

   The bend is previewed, along with its orientation symbolized by arrows. The Left and Right texts further indicate this orientation and are useful to define different bend extremities.



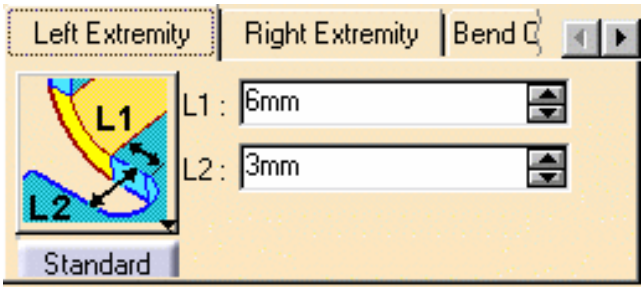
5. Click the **More>>** button to display further options:



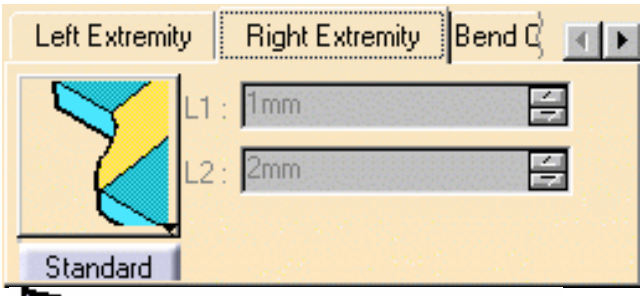
You can define:

- the left and right extremity settings (see also extremities definition settings)
- the corner relief definition settings
- and the bend allowance settings.

6. Within the Left Extremity tab, choose the Mini with round relief bend extremity type, deactivate the L1 and L2 length formulas, and set them to 6mm and 3mm respectively.

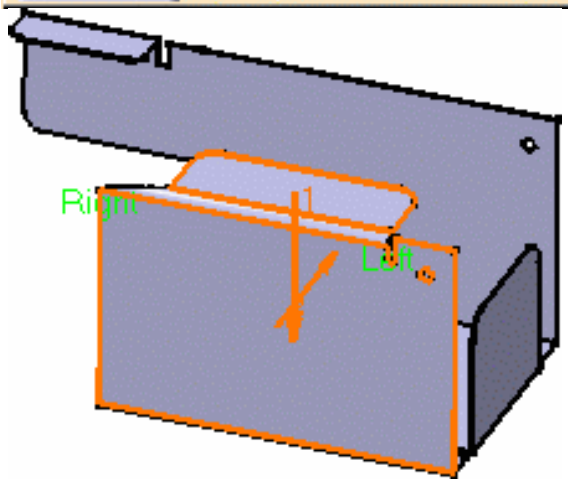


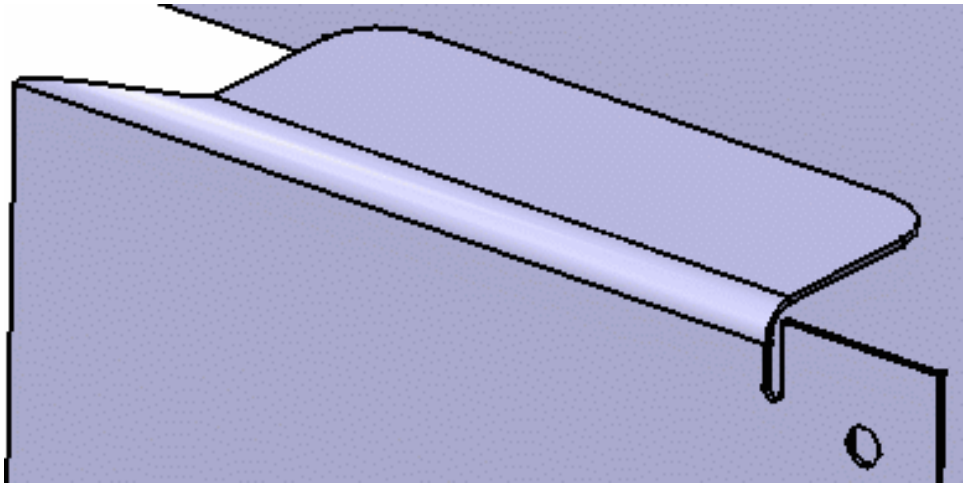
7. Click the Right Extremity tab, and choose the Curved shape type.



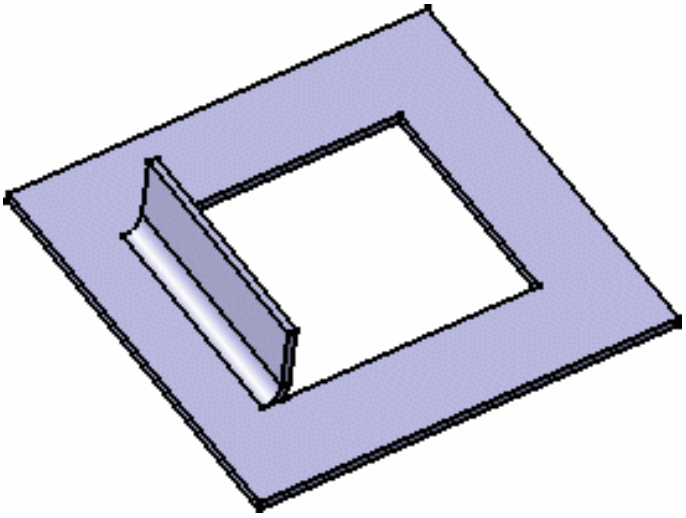
8. Click Preview to visualize the left and right extremities.
9. Click OK in the Bend Definition dialog box.

   The Bend (identified as Cylindrical Bend) is created with the specified extremity types and is added to the specification tree.

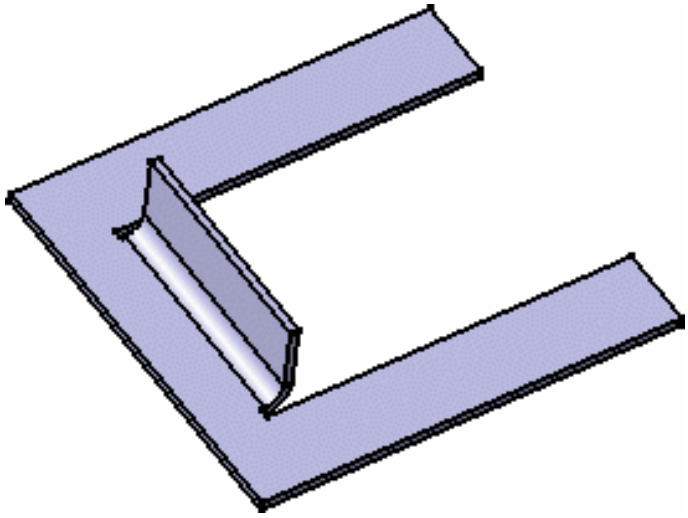




- The extremities and the corner relief defined with the current dialog box will apply locally and prevail over any other global definition.
- Be careful when creating bends with **square** or **round** relief. Depending on the geometry configuration, this can lead to removing more matter than you would expect. Indeed, a corner relief being computed on the whole intersection of the elements involved (bends or bend/wall), in the following configuration, the matter is removed till the end of the wall.

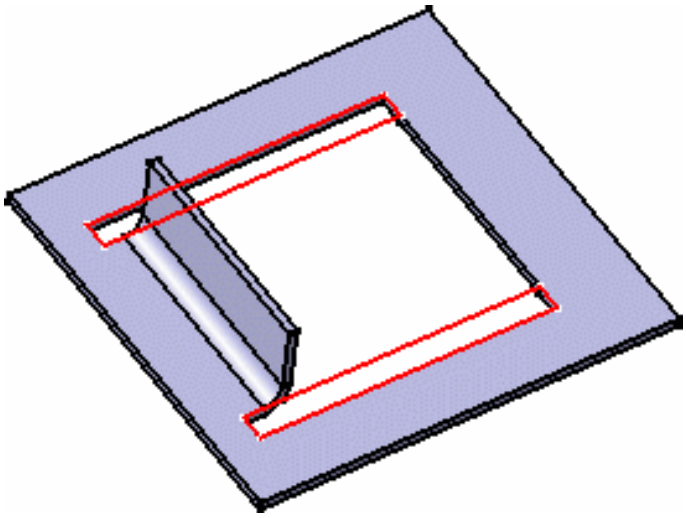


Bend with no relief



Bend with square relief

In this case, it is best to replace the bend corner relieves with cutouts (identified in red in the figure below) that will act as corner relieves removing matter to the part.



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Generating Bends Automatically

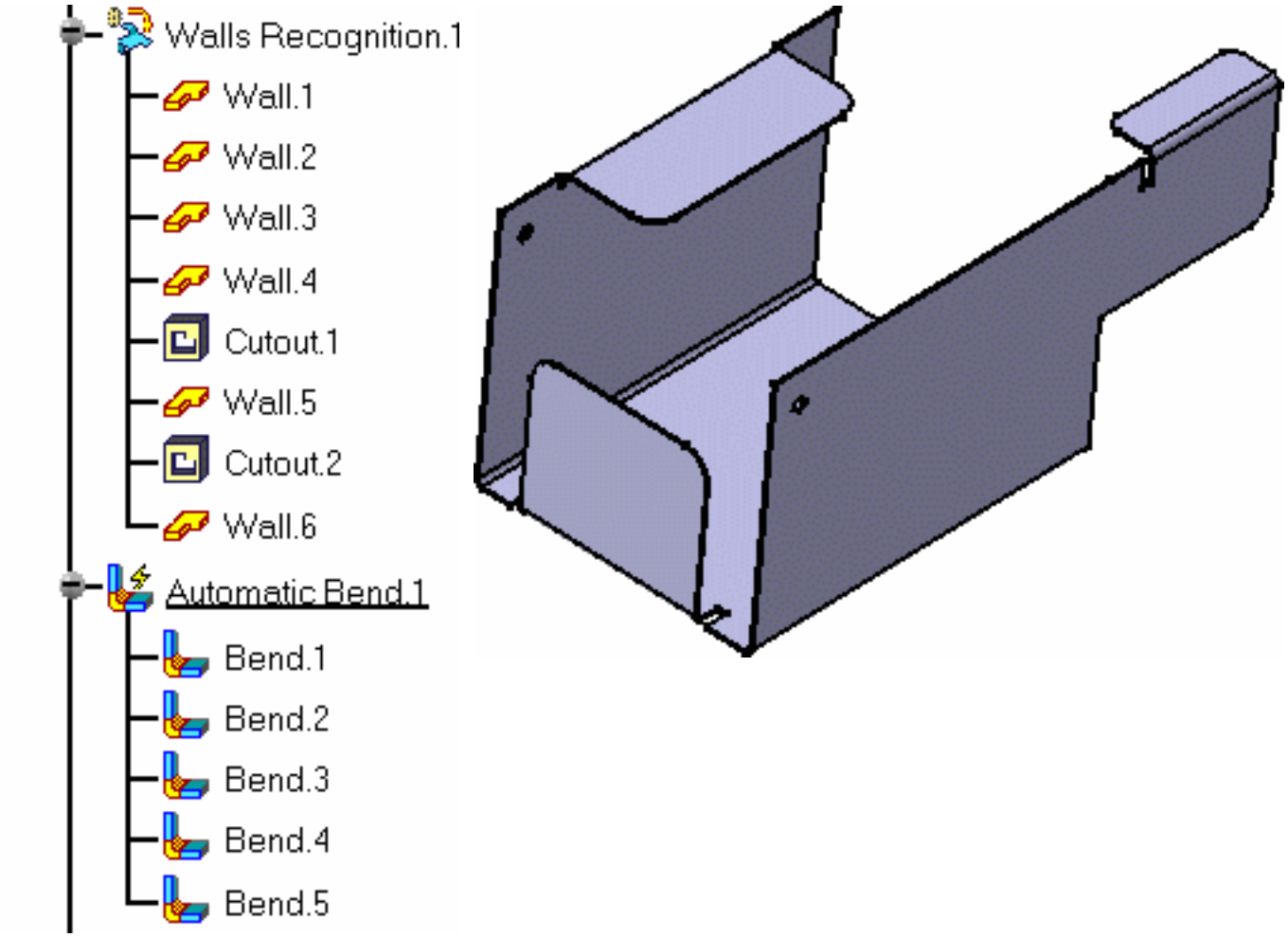
This functionality is only available with SheetMetal Design.

This task explains how to automatically generate bends between walls in the Sheet Metal part. You can first create all the bends, then modify the parameters for any of the generated bends.
However, when an ambiguity arises, that is when more than two bends end on the same vertex, the bends are not automatically generated. You then need to create them manually, so as to explicitly select the walls between which the bends are to be created.

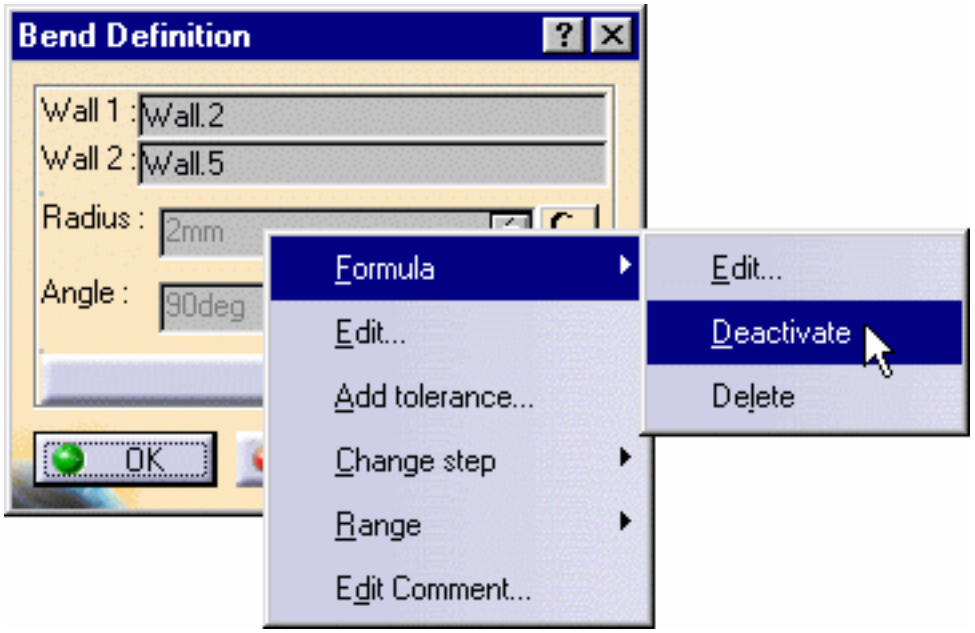
Open the BendExtremities01.CATPart document from the samples directory.

1. Select the **Automatic Bends** icon .

   The bends are created.

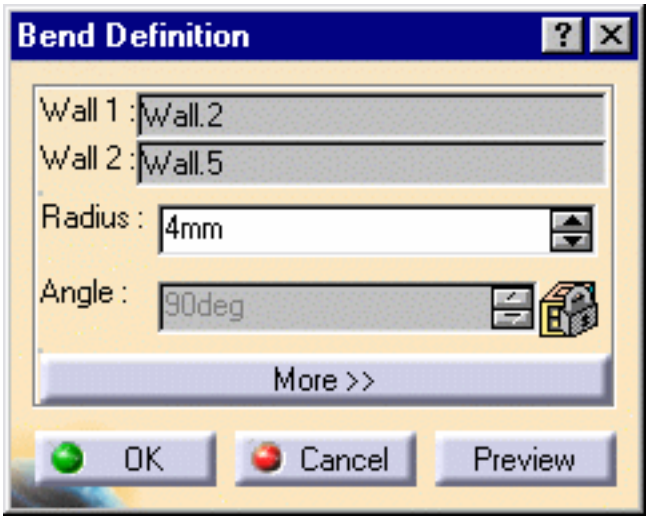


2. Double-click the bend of interest: **Bend.4**
   The Bend Definition dialog box opens.
3. Right-click the **Radius** field: the contextual menu appears.
4. Deactivate the formula: you can now change the value.

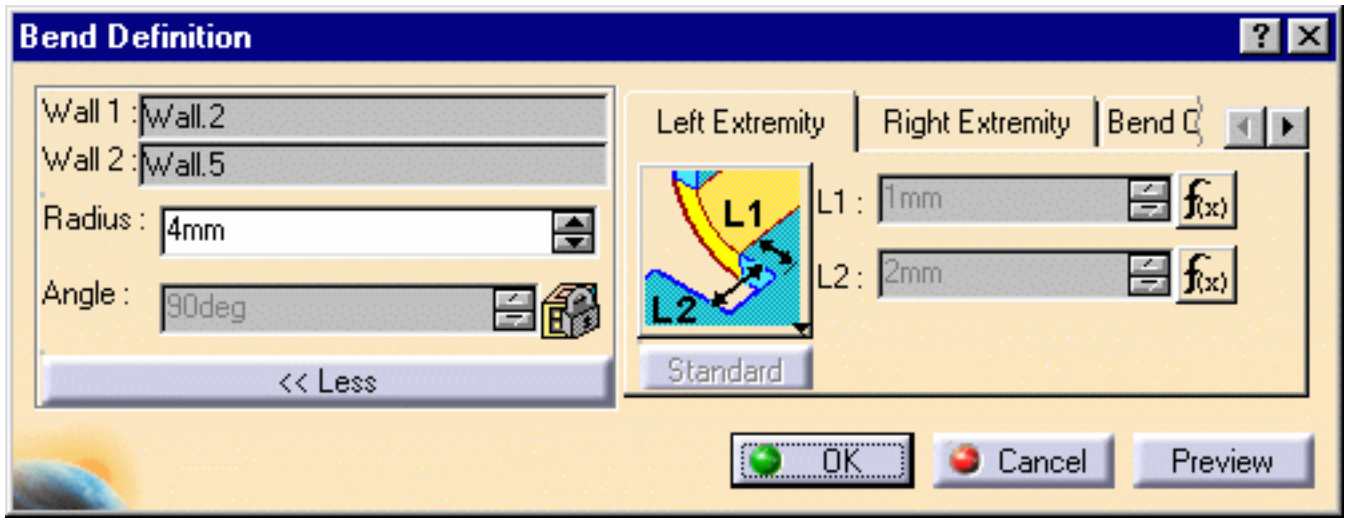


5. Enter 4mm for the Radius and click Preview.

   Bend.4 is modified.



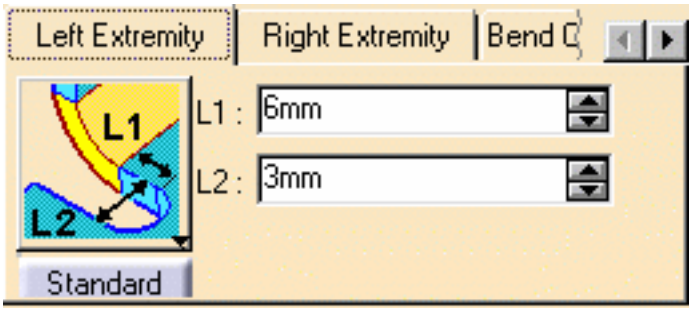
6. Click the More button to display further options:



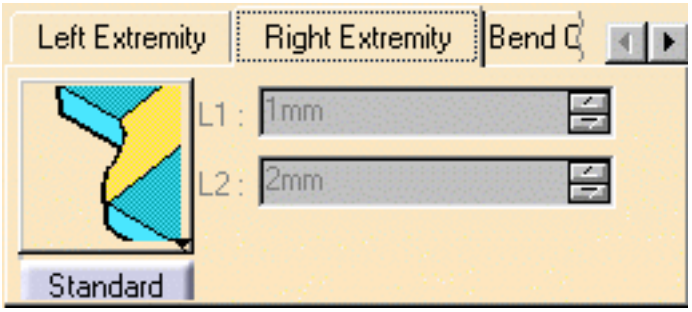
You can re-define:

- the left and right extremity settings (see also extremities definition settings)
- the corner relief definition settings
- and the bend allowance settings.

7. Within the Left Extremity tab, choose the Mini with round relief bend extremity type, deactivate the L1 and L2 length formulas, and set them to 6mm and 3mm respectively.

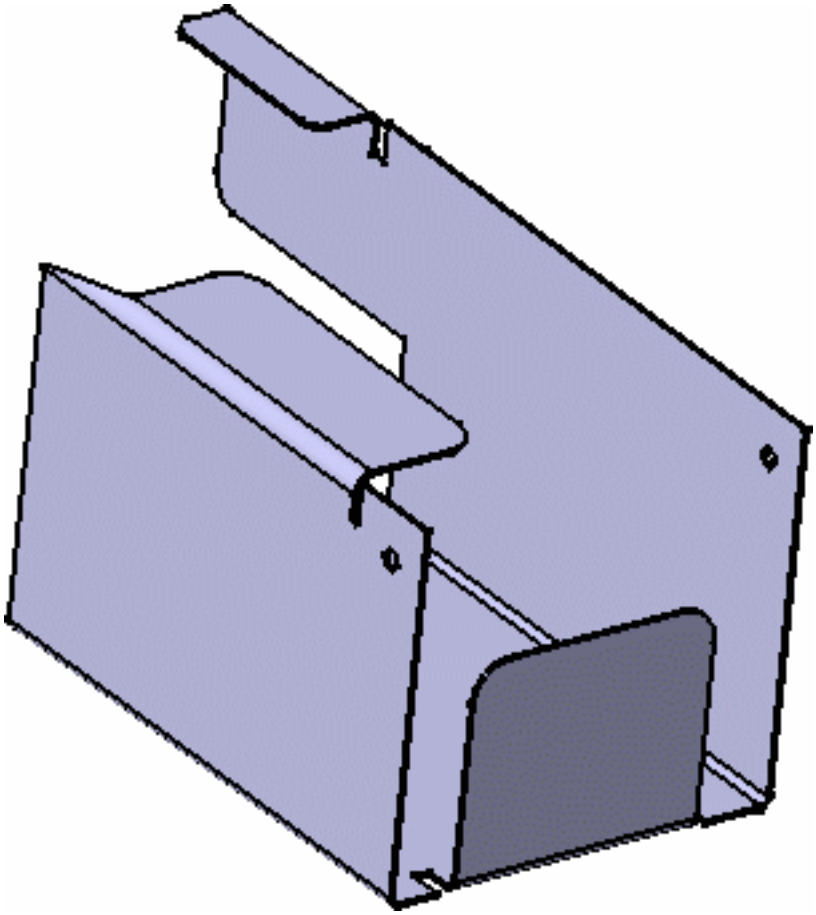


8. Click the Right Extremity tab, and choose the Curved shape type.



9. Click OK in the Bend Definition dialog box.

   The bend is modified with the specified options



The extremities and the corner relief defined with the current dialog box will apply locally and prevail over any other global definition.

Push the more button to display:

- the extremities definition settings,
- the corner relief definition settings,
- and the bend allowance settings.

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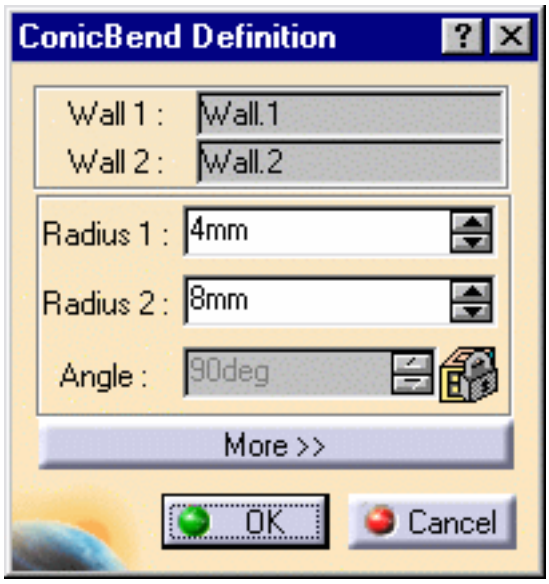
Creating Conical Bends

This functionality is only available with SheetMetal Design.

This task explains how to generate conical bends between two walls in the Sheet Metal part.
These bends are different from the standard bend in that they allow different radius values at each end of the bend.

Open the ConicalBend1.CATPart document from the samples directory.

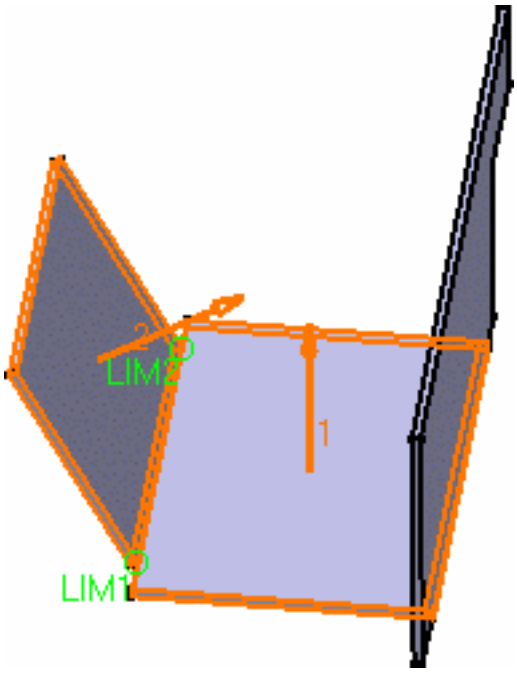
1. Select the **Conic Bend** icon .
   The Conic Bend Definition dialog box opens.



2. Select **Wall.1** and **Wall.2** in the specification tree or in the geometry.
   The Bend Definition dialog box is updated, and arrows are displayed indicating the walls orientation.

   You can click on the arrow to invert them if needed.

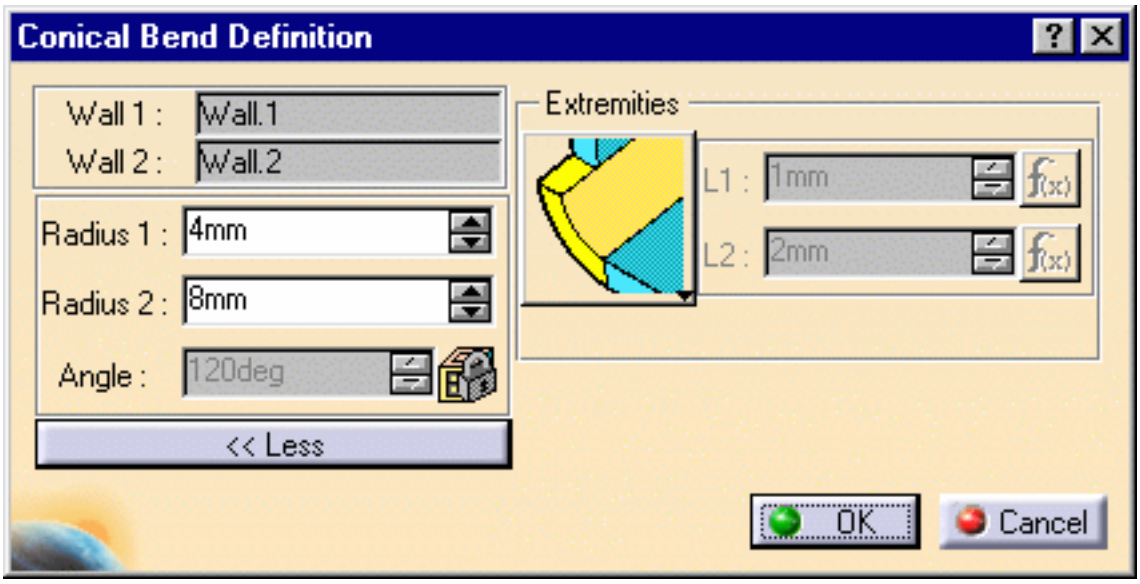
   The LIM1 and LIM2 texts indicate the endpoints for the bend.



3. Enter the radius values for each end of the conical bend.
   If the difference between the specified radius values does not allow the generation of a cone with an angle greater than 1 degree, a warning is issued prompting you to increase one of the radii.
   Click OK in the Warning dialog box, and increase/decrease the radius values.

By default, **Radius 2** is twice **Radius 1**.

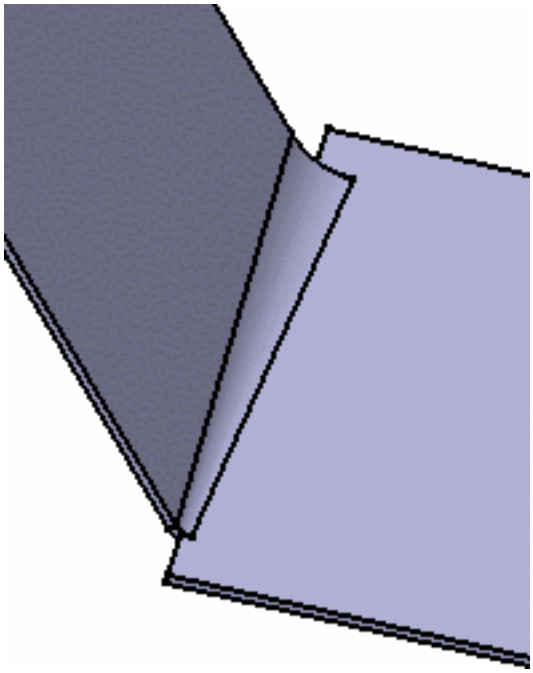
4. Click **More...** to display further options.



5. Choose the bend extremities:

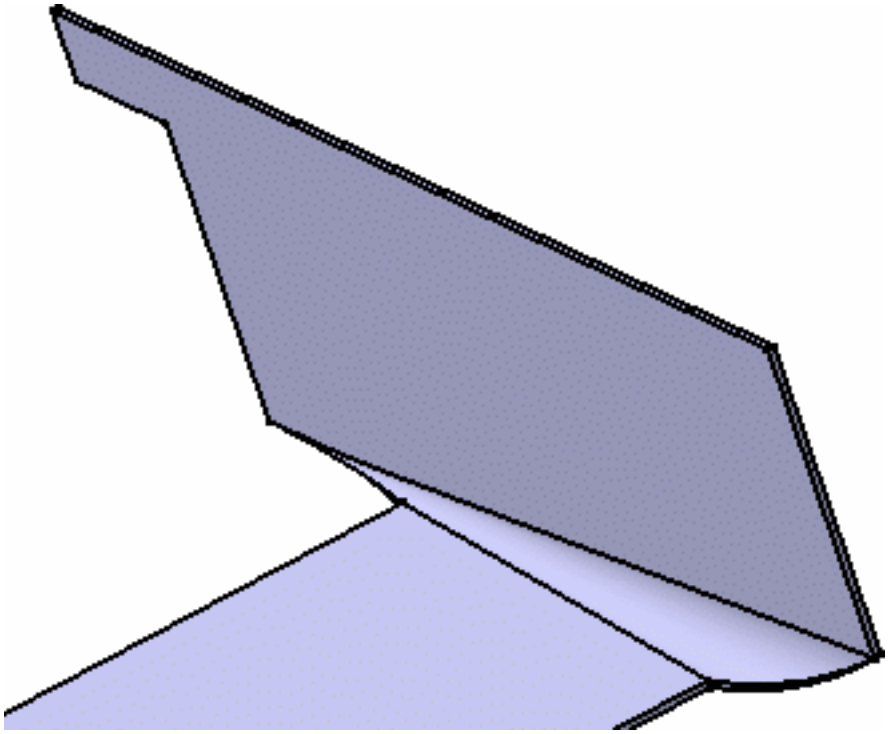
- **Mini with no relief**: the shortest possible bend is created, and presents no relief
- **Curve shaped**: the bend is created keeping the tangency continuity with the support walls.
- **Maximum**: the bend is calculated between the furthest opposite edges of the supporting walls.

6. Click OK.



The Bend is created.

- The two walls must be connected by the edge of their internal faces.
- The **Angle** field is locked. It indicates the angle value between the two walls between which the bend is computed.
- Should you choose the **Curve shaped** extremity option, the bend would look like this:



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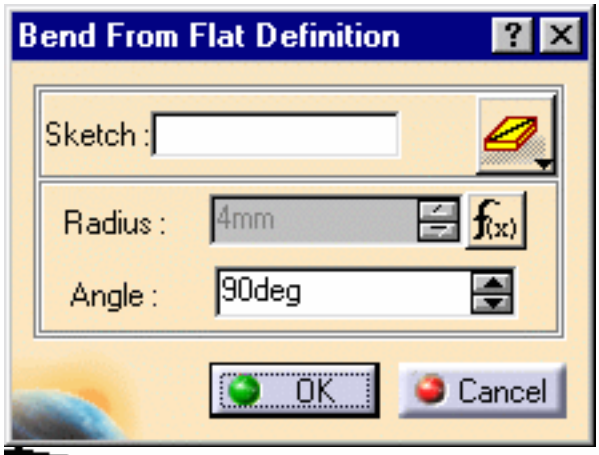
Creating Bends From a Line

This task explains how to generate bends based on a line (also called flat bends). A wall and a bend are created.

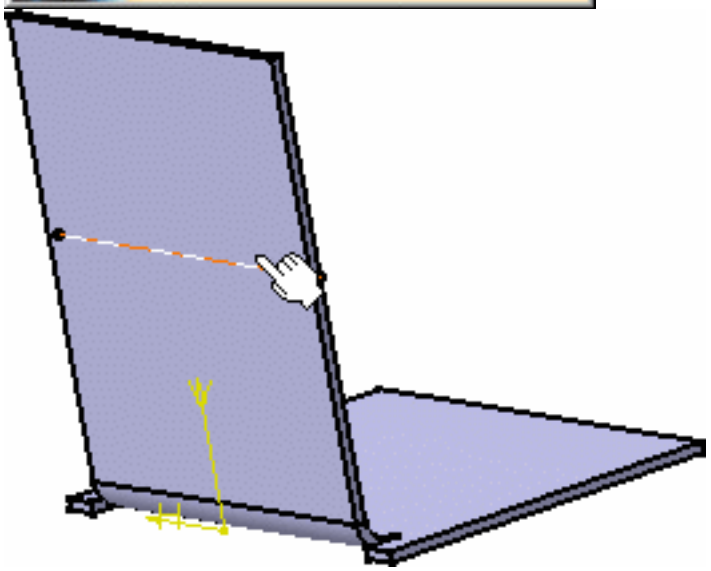
Open the FlatBend1.CATPart document from the samples directory.

1. Select the reference wall then the **Bend From Flat** icon .

   The Bend From Flat Definition dialog box opens.



2. Select a sketch (Sketch.3 here).
   This sketch must necessarily be a line.



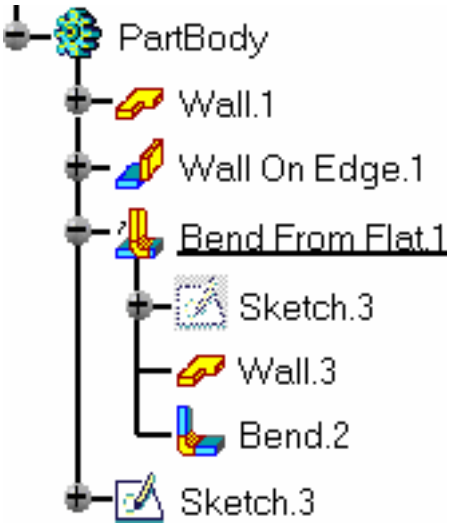
3. You can choose the line extrapolation option:

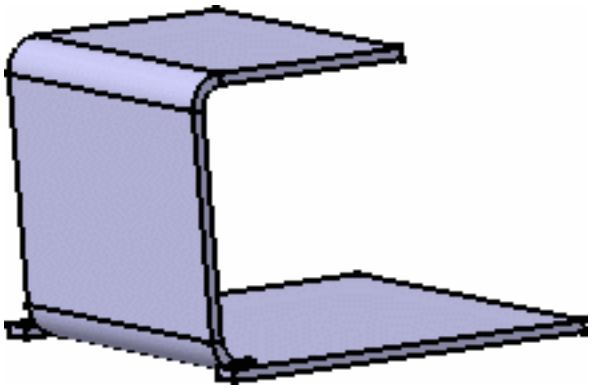
- the line is extrapolated up to the wall edge (**Bend From Flat Until**)
- the line is not extrapolated, and the bend is limited to the line's length (**Bend From Flat Length**)

The Radius value is the one defined when editing the sheetmetal parameters:
Right-click the **Radius** field and select **Formula -> Deactivate** from the contextual menu to change the value.

4. Set the angle value between the generated wall and the reference wall.

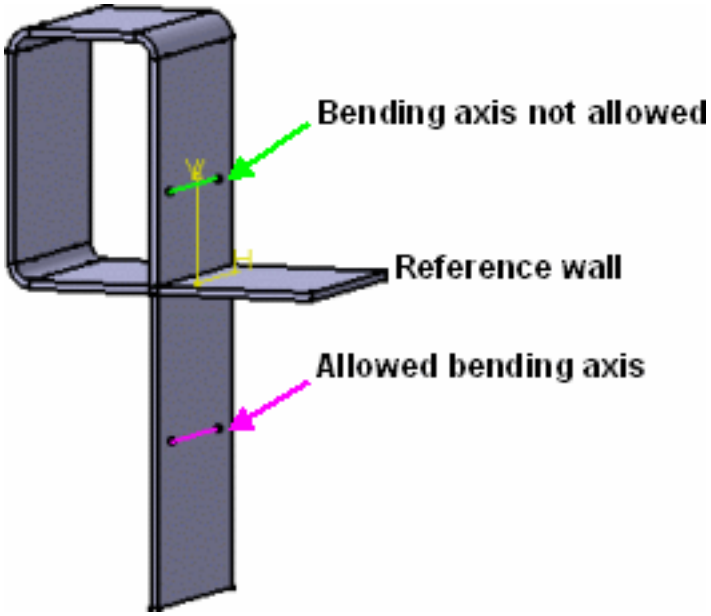
5. Click OK to create the bend.





- When creating such a bend on a reference wall (first wall), an arrow indicates which part is to be folded. Click this arrow to invert the side that will be bent.

- Such bends cannot be created, if the section to be folded already intersects the part.



- Bends from line should be performed on end walls, or prior to creating further walls on the bent one.
- Perform the bend before creating the stamping features, as stamps are not retained when the part is folded with the bend.

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Creating Swept Walls

This section explains and illustrates how to create and use various kinds of swept walls, i.e. walls based on a given contour that is swept along a spine.


Create a flange: select a spine, and set the radius, length, and angle values.


Create a hem: select a spine, and set the radius, and length values.


Create a tear drop: select a spine, and set the radius, and length values.


Create a swept flange: select a spine, and a user-defined profile

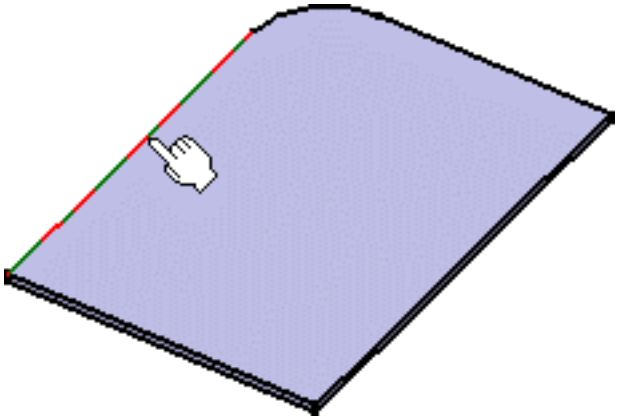
Redefine swept walls limits: choose the **Relimited** type, and select a point lying on the spine or a plane normal to the spine and intersecting it as limits


Display swept walls axes in drafting sheets: open a CATDrawing document, click the **Unfolded View** icon, and select a plane

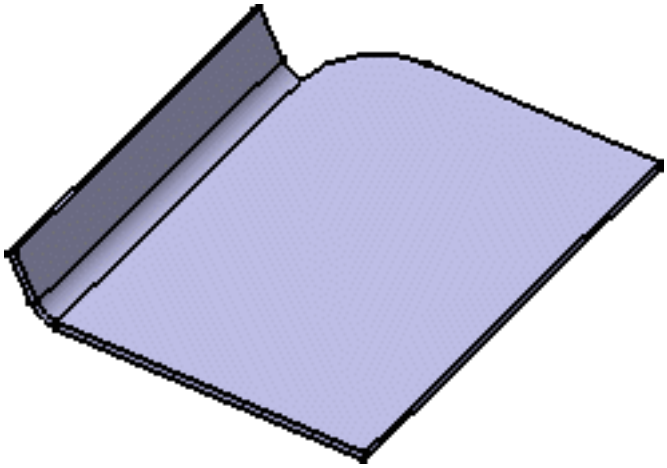
Selecting the Spine

Whatever the type of the swept wall you wish to create, you first need to select one or more contiguous edges to make up the spine along which the contour, either pre- or user-defined, is to be swept. You can:

- manually select one, or more, edge(s)

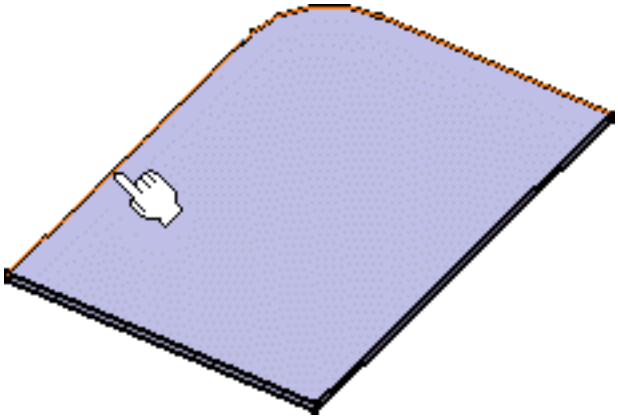


Selection without propagation

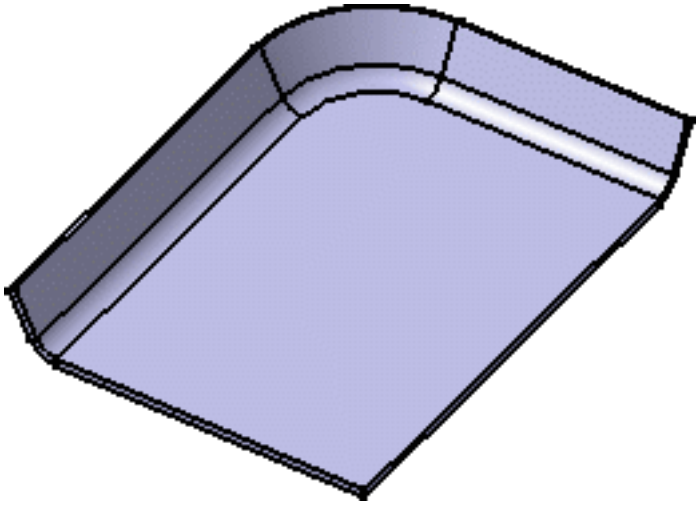


Resulting flange without propagation

- select one edge and click the **Tangency Propagation** button: all contiguous and tangent edges are selected. In this case, would you need to remove one edge, you need to manually select it. Remember that only extremity edges can be removed without breaking the continuity between edges.



Selection with propagation



Resulting flange with propagation

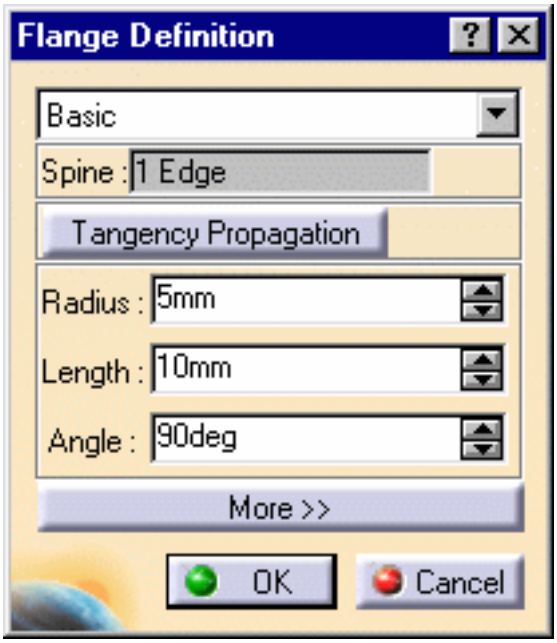
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Creating a Flange

This task explains how to generate a flange from a spine and a profile.

Open the SweptWall01.CATPart document from the samples directory.

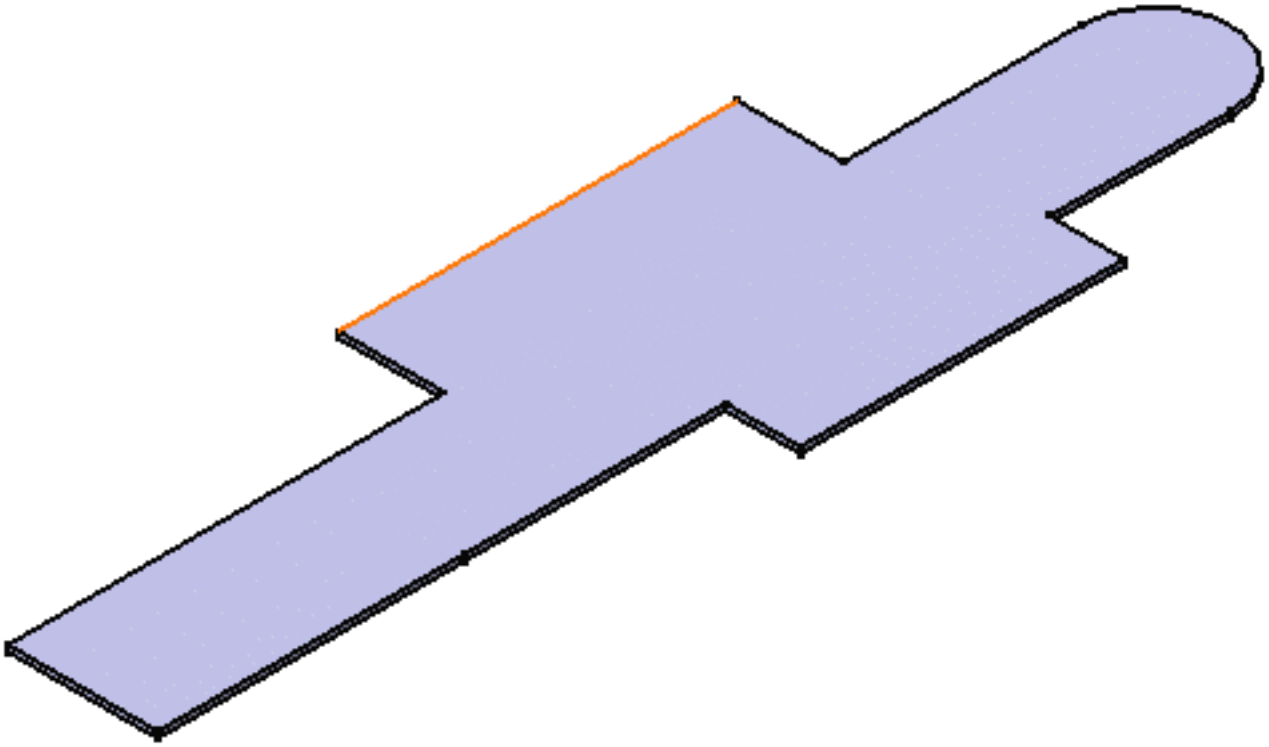
1. Select the **Flange** icon .



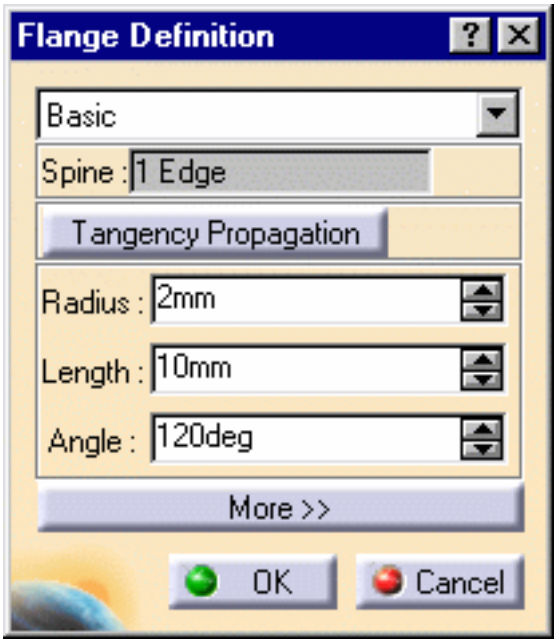
The Flange Definition dialog box opens.



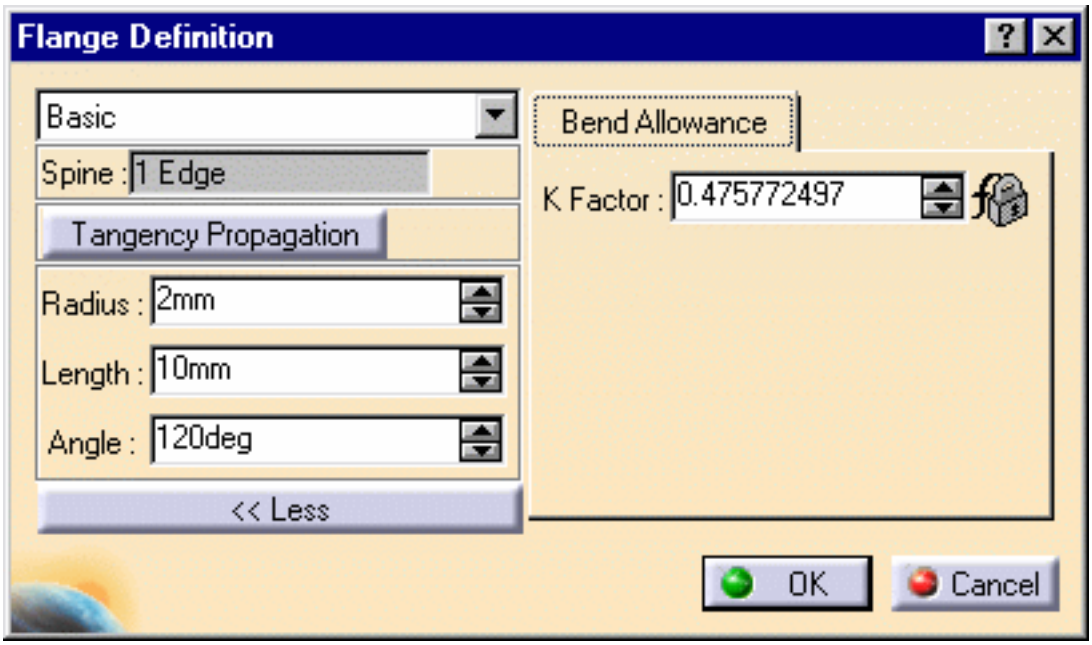
2. Select the edge as shown in red.



3. Enter 2mm in the Radius field, 10mm in the Length field and 120° for the Angle.

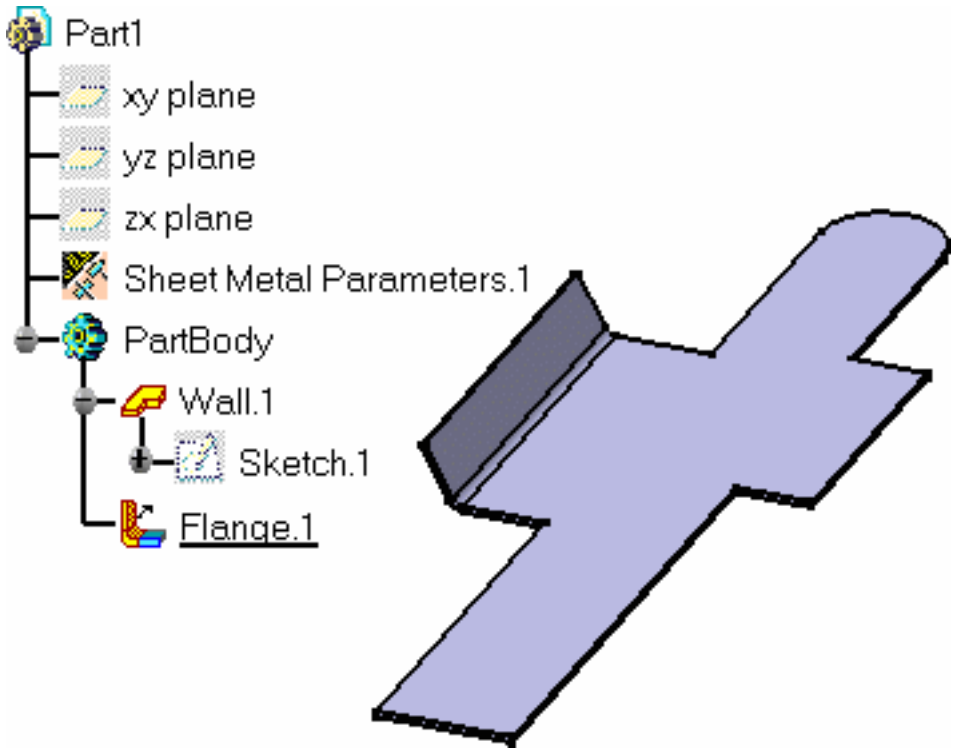


4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click OK to create the flange.



The feature is added to the specification tree.

- Use the **Tangency Propagation** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the flange limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

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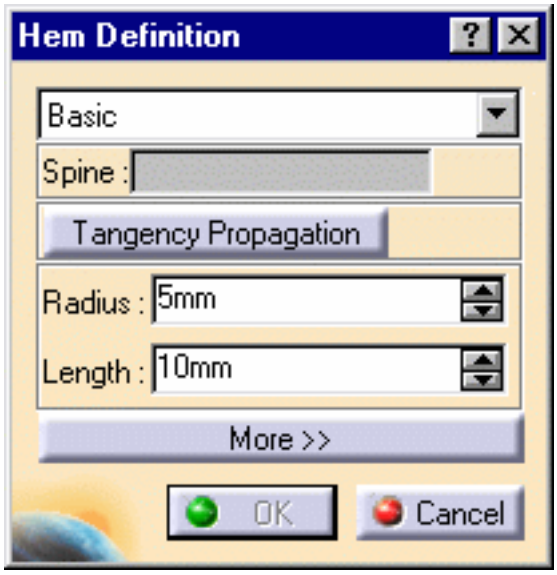
Creating a Hem

This task explains how to generate a hem from a spine and a profile.

The SweptWall01.CATPart document is still open from the previous task.
If not, open the SweptWall02.CATPart document from the samples directory.

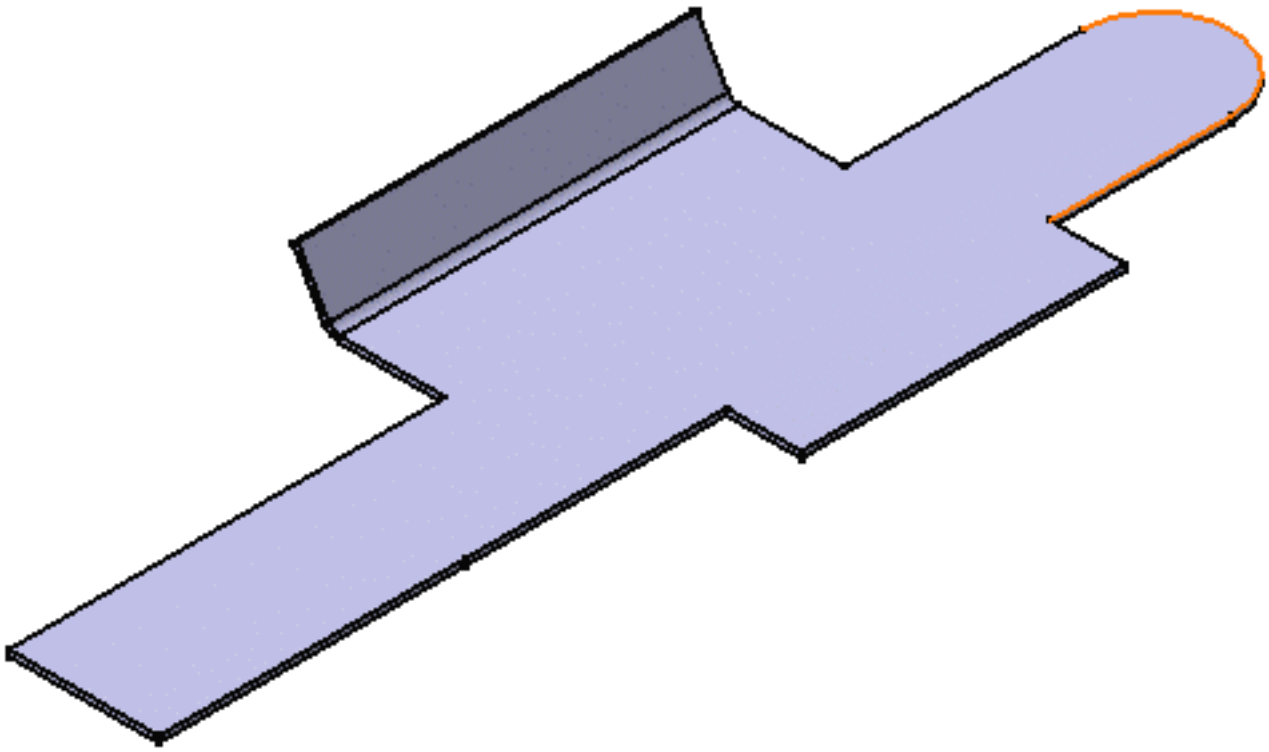
1. Select the **Hem** icon in the **Swept Walls** sub-toolbar.



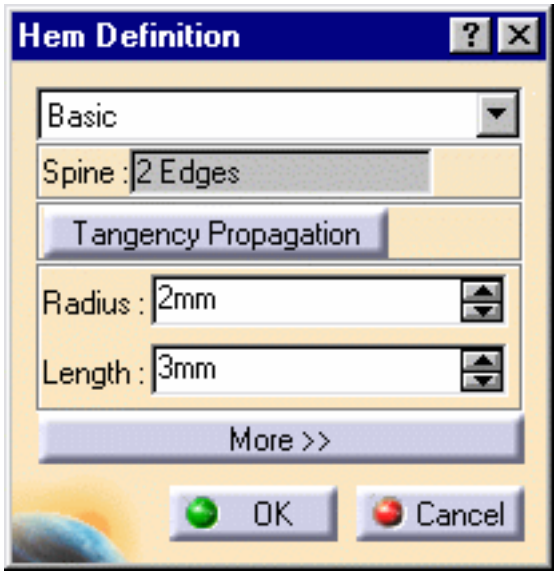


The Hem Definition dialog box opens.

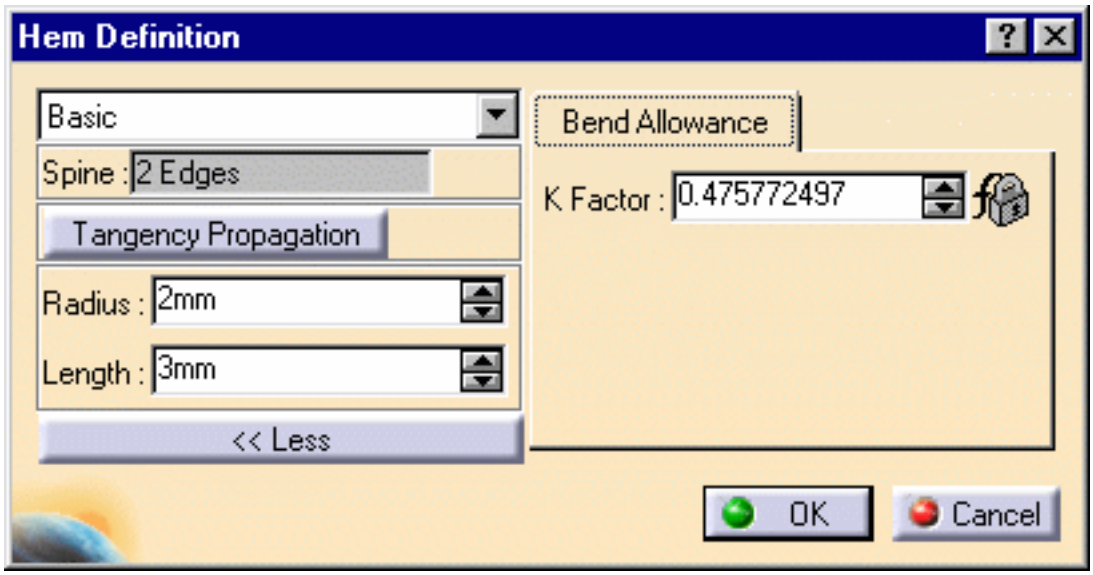
2. Select the edges as shown in red.



3. Enter 2mm in the **Radius** field, and 3mm in the **Length** field.

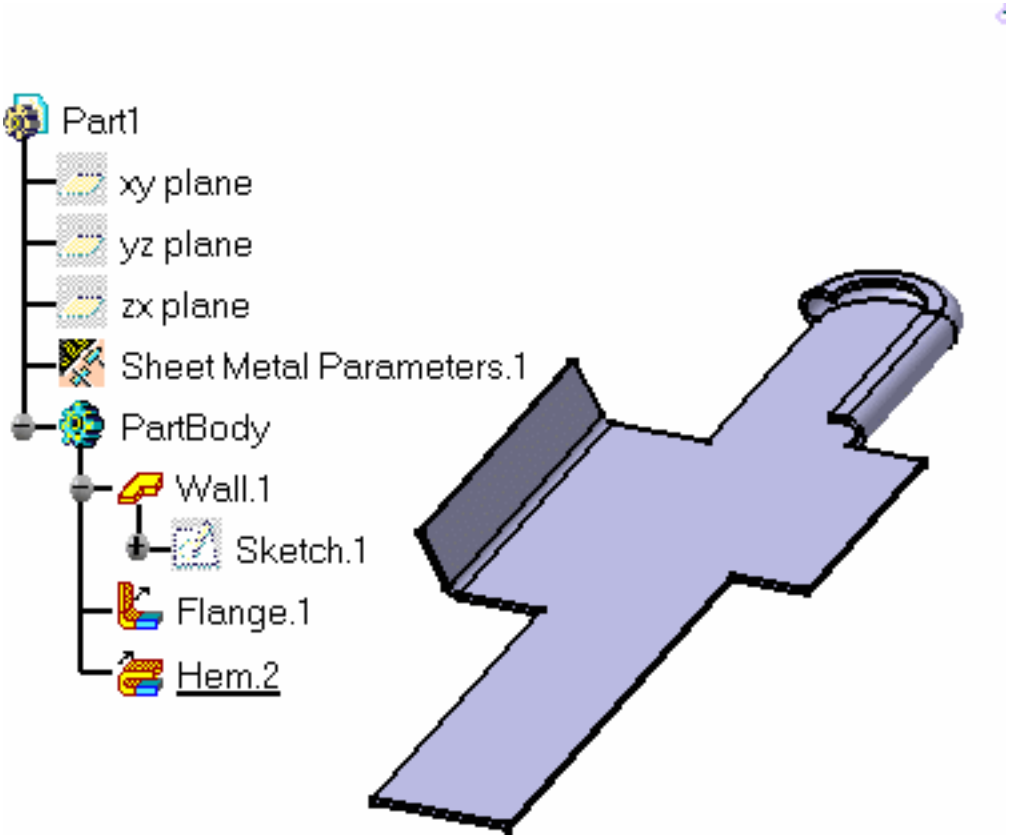


4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click **OK** to create the hem.



The feature is added to the specification tree.

- Use the **Tangency Propagation** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the hem limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

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Creating a Tear Drop

This task explains how to generate a tear drop from a spine and a profile.

The SweptWall01.CATPart document is still open from the previous task.
If not, open the SweptWall03.CATPart document from the samples directory.

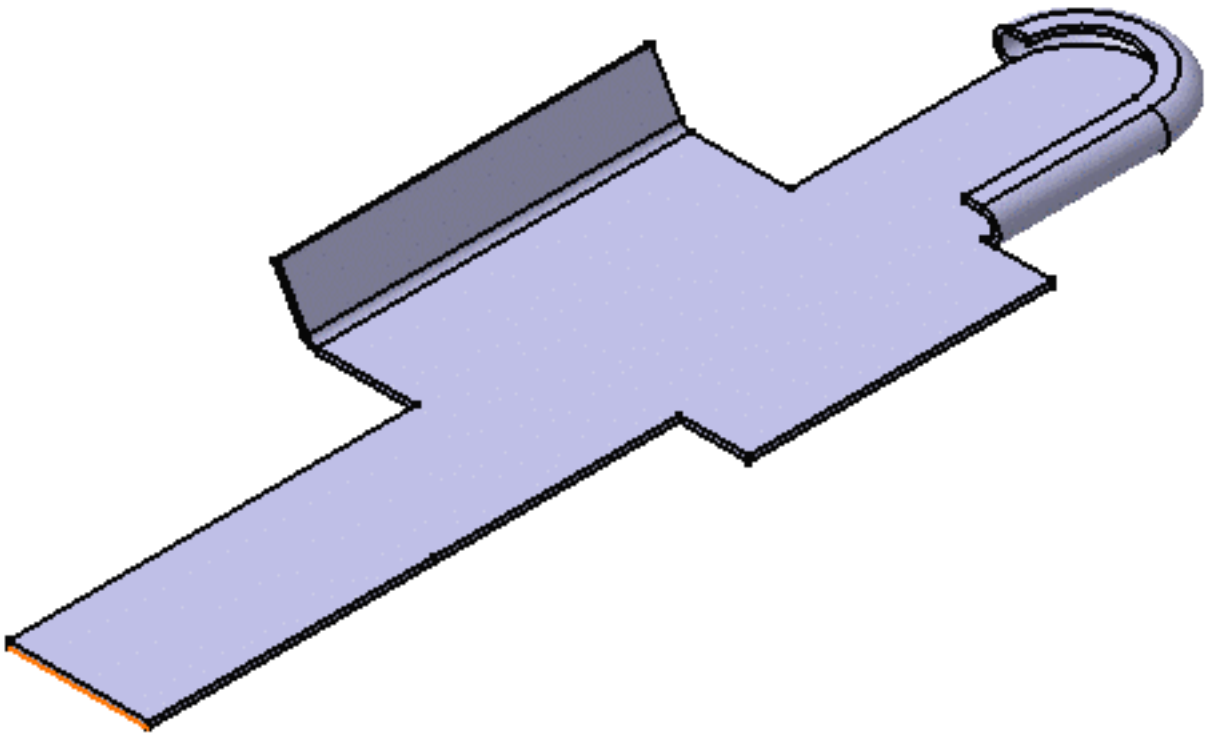
1. Select the **Tear Drop** icon in the **Swept Walls** sub-toolbar.



The Tear Drop Definition dialog box opens.



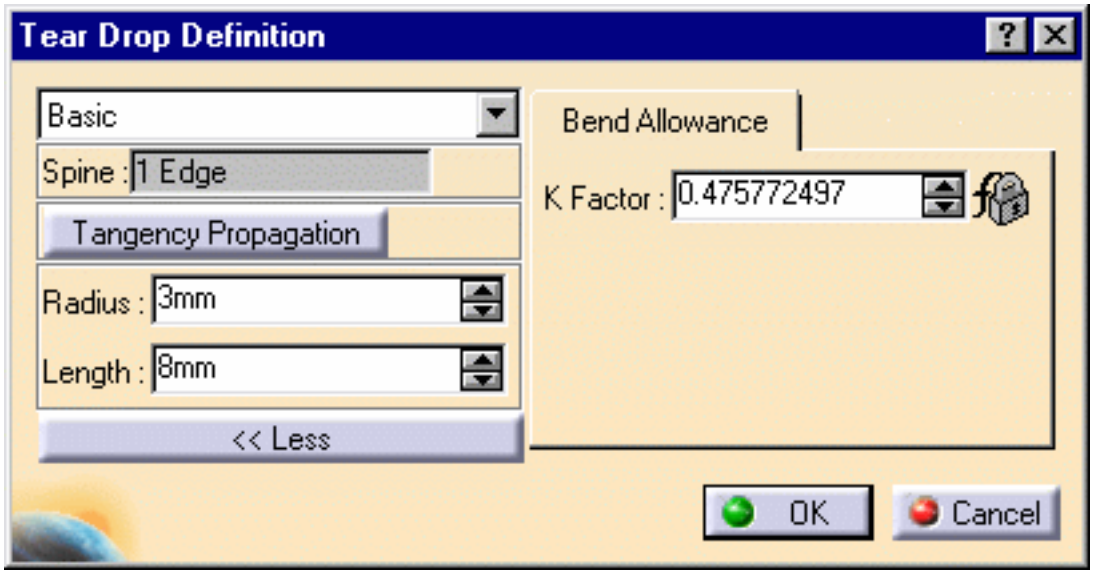
2. Select the edge as shown in red.





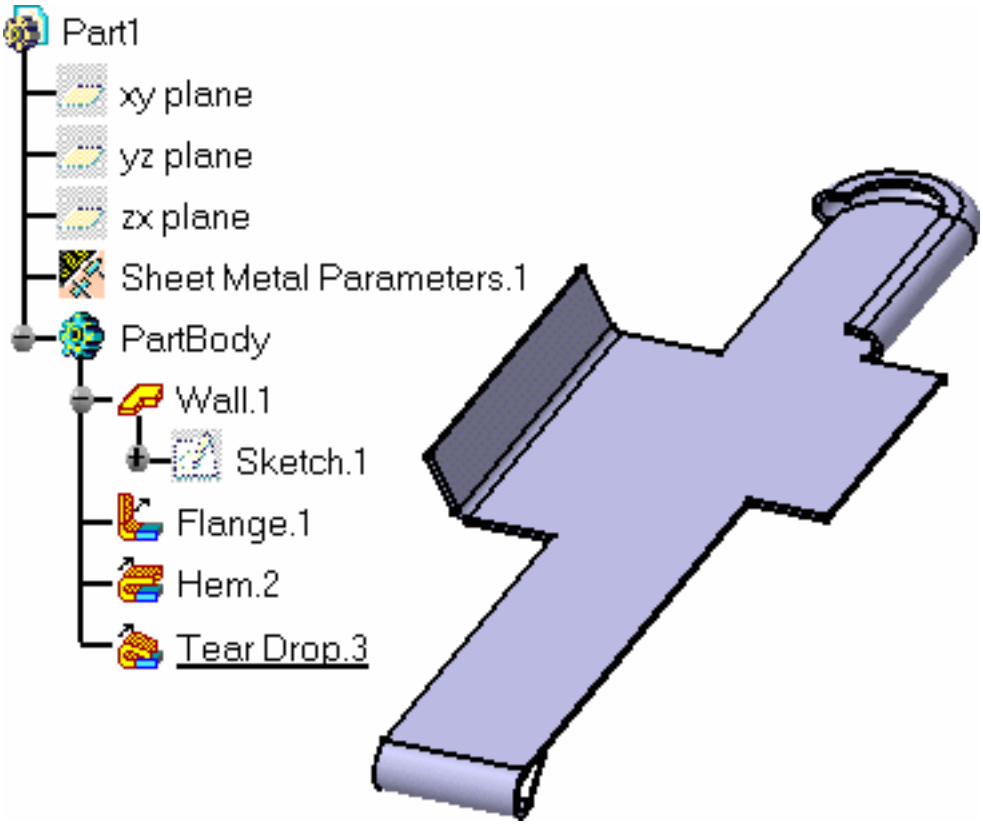
3. Enter 3mm in the **Radius** field, and 8mm in the **Length** field.

4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
   You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click **OK** to create the tear drop.



The feature is added to the specification tree.

- Use the **Tangency Propagation** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the hem limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

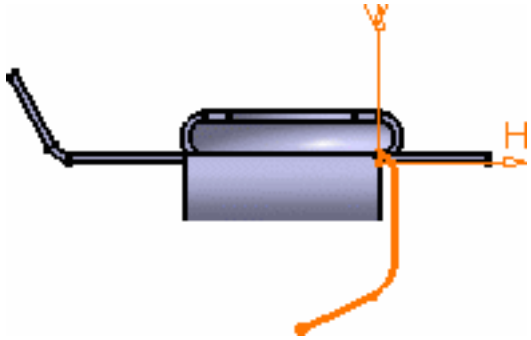
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Creating a Swept Flange

This task explains how to generate a swept flange from a spine and a user-defined profile.

The SweptWall01.CATPart document is still open from the previous task.
If not, open the SweptWall04.CATPart document from the samples directory.

1. Using the Sketcher , define a profile in the yz plane as shown below:

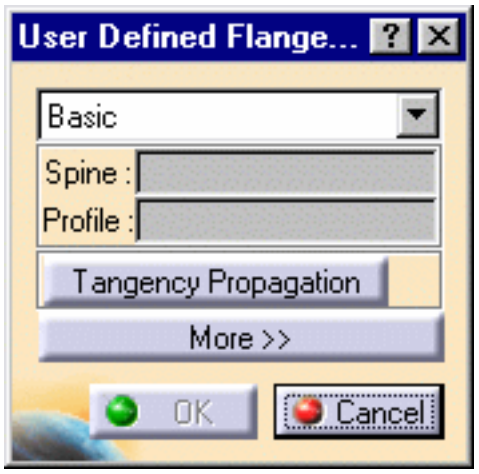


Then quit the Sketcher, using the **Exit** icon .

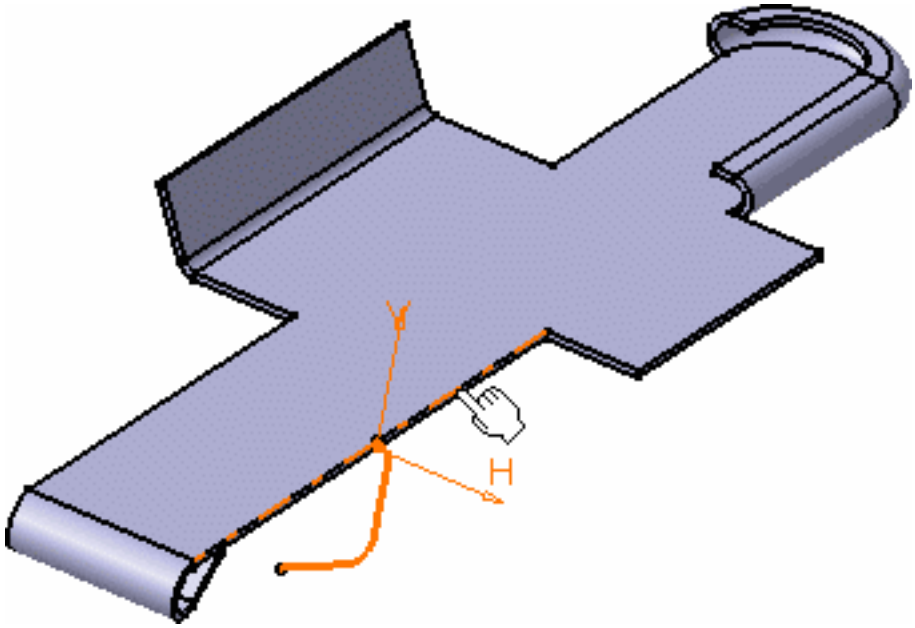
2. Select the **Swept Flange** icon in the **Swept Walls** sub-toolbar.



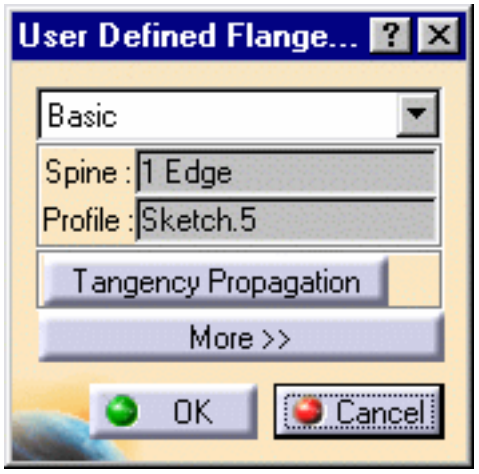
The **User Defined Flange** Definition dialog box opens.



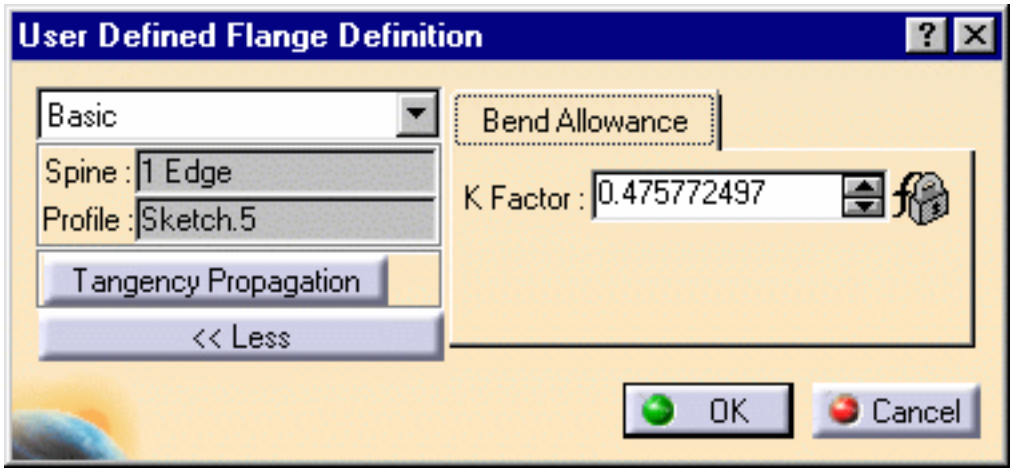
3. Select the edge and the profile, as shown in red.



The dialog box looks like this:

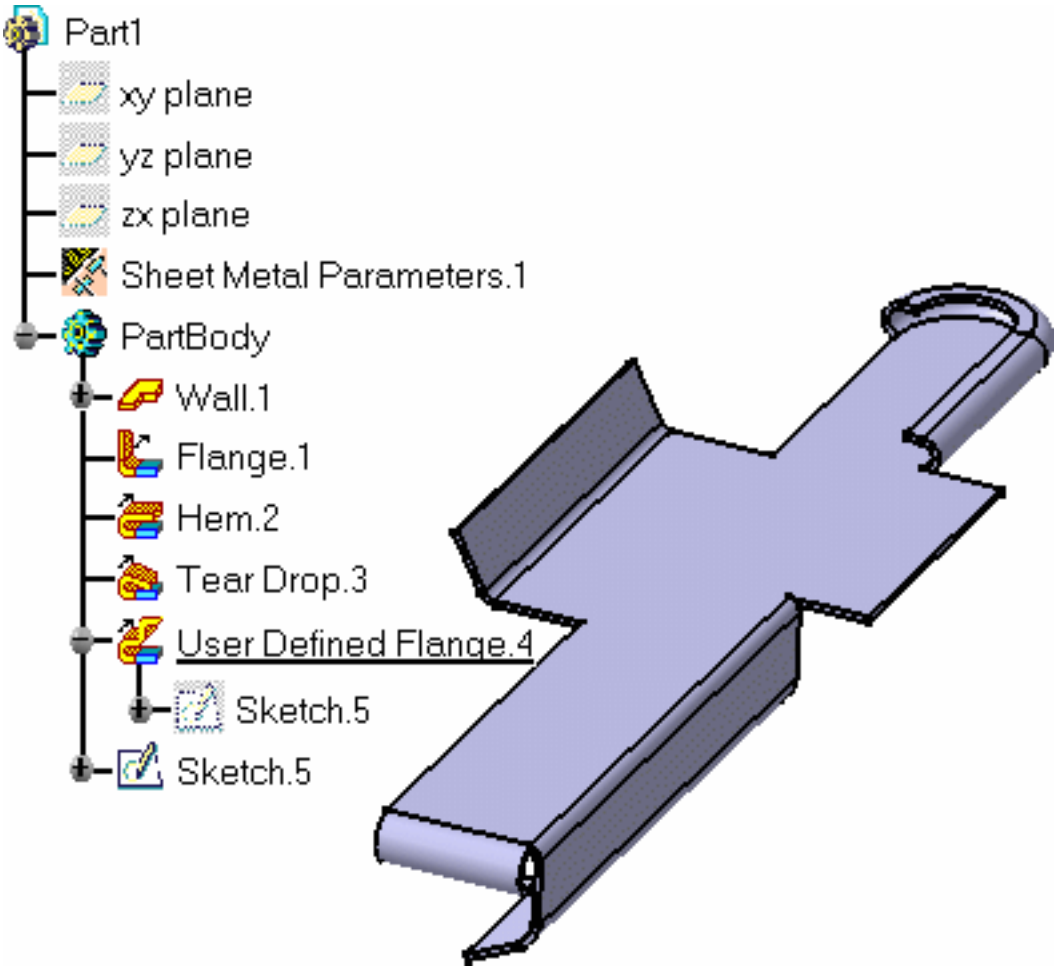


4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
   You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click **OK** to create the swept flange.



The feature is added in the specification tree.

- Use the **Tangency Propagation** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the hem limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

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Redefining Swept Wall Limits

This task explains how to redefine the spine's limits when creating any type of swept walls, using existing geometric elements: points lying on the spine or intersecting planes.

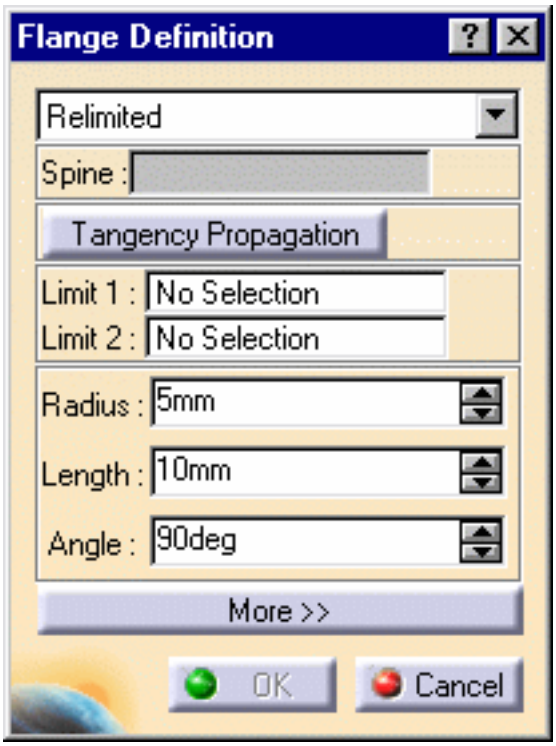
Open the SweptWall01.CATPart document.

1. Select the **Flange** icon .

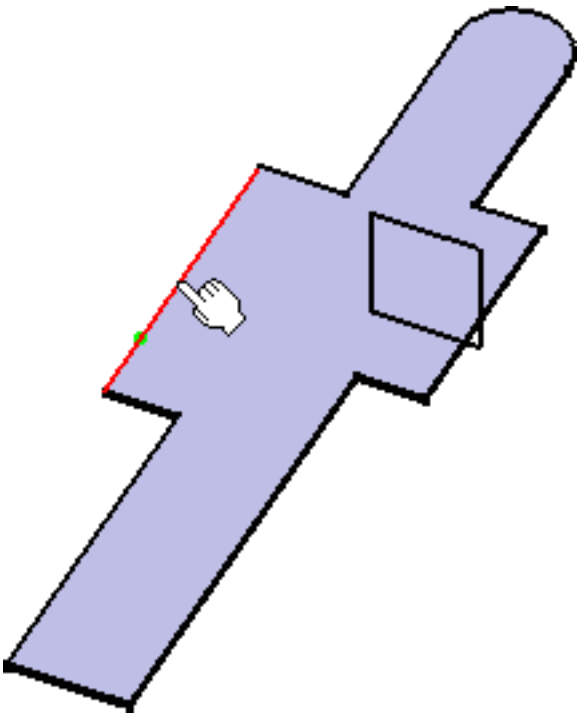
   The Flange Definition dialog box opens.

2. Using the combo list, choose the Relimited type.

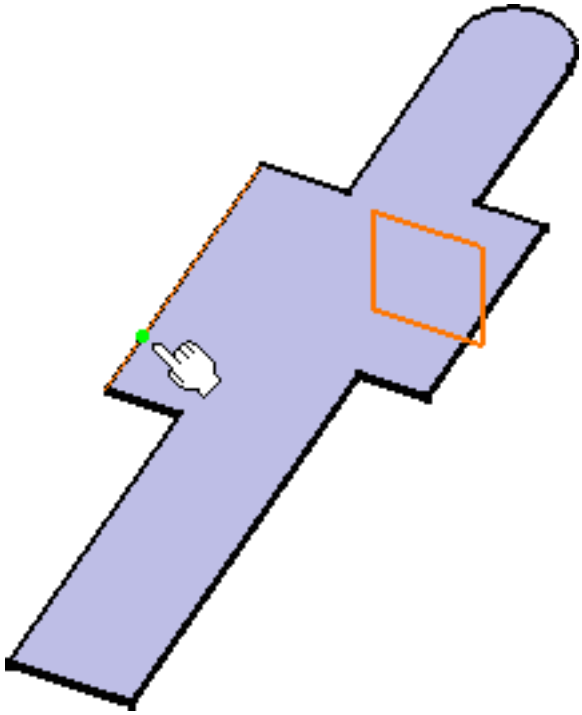
   The Flange Definition dialog box is updated and now displays two **Limit** fields.



3. Select the spine.
   Here we select a single edge. See also Selecting the Spine.



4. Successively select the two limiting elements.
   Here we select a point lying on the spine as the first selecting element, and a plane intersecting the spine as the second limiting element.

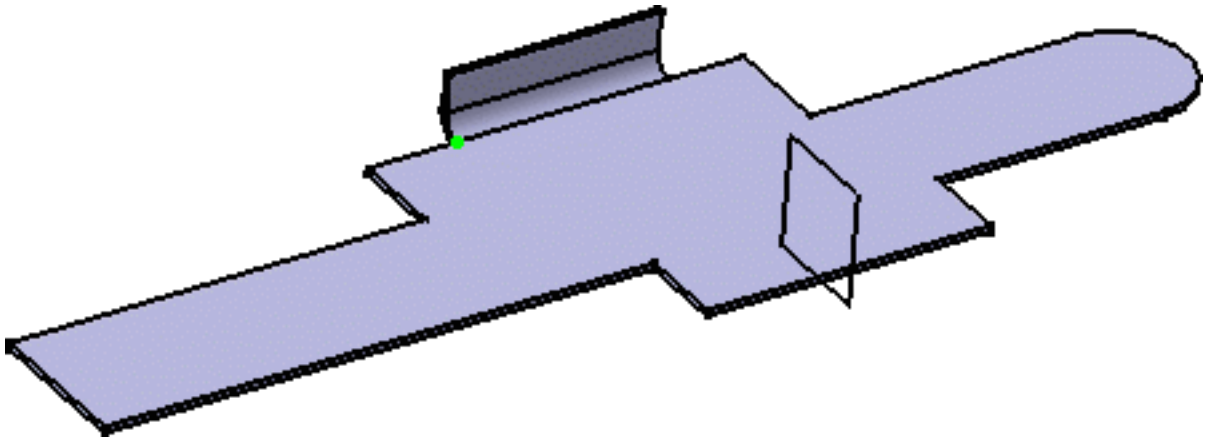


Make sure intersecting elements are normal to the spine, and they intersect it only once.

5. Specify the swept wall values.
   In the example of the flange you set the Radius, Length and Angle values. You may also click **More >>** to display further options. See Creating a Flange.

6. Click OK.

The swept wall is created within the limits on the spine.



Displaying Swept Wall Axes

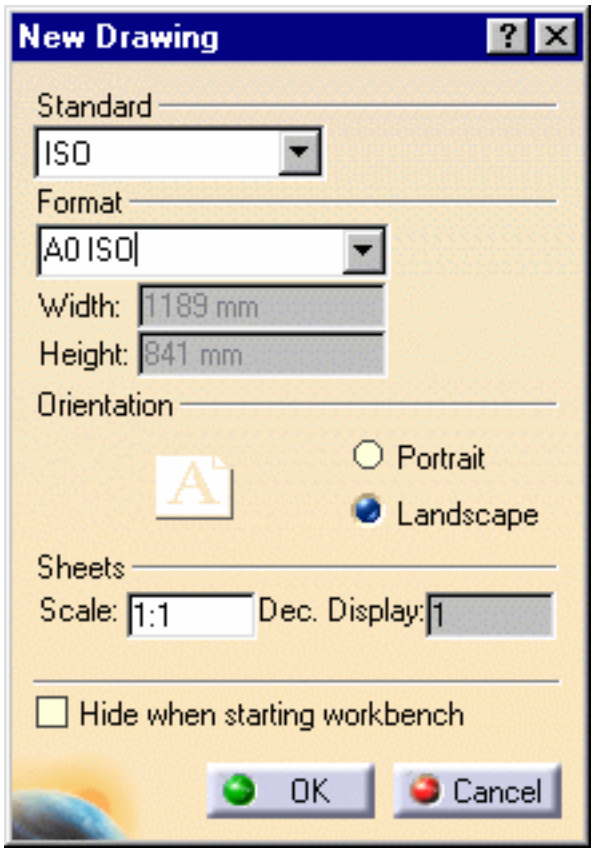
This task explains how to generate an unfolded view of a part with swept wall in a drawing sheet including the axes of planar hems, tear drops, and flanges are displayed within the drawing.

Open the SweptWall01.CATPart document from the samples directory.

If you use the NEW Sheet Metal Design, open the NEWSweptWall1.CATPart document.

Make sure that the **Generate axis** button is checked in the **Tools -> Options -> Mechanical Design -> Drafting -> View** tab, prior to generating a view in a . CATDrawing document.

1. Click  or select **File -> New...**
2. Select the Drawing type and click **OK.**



3. Click **OK**.

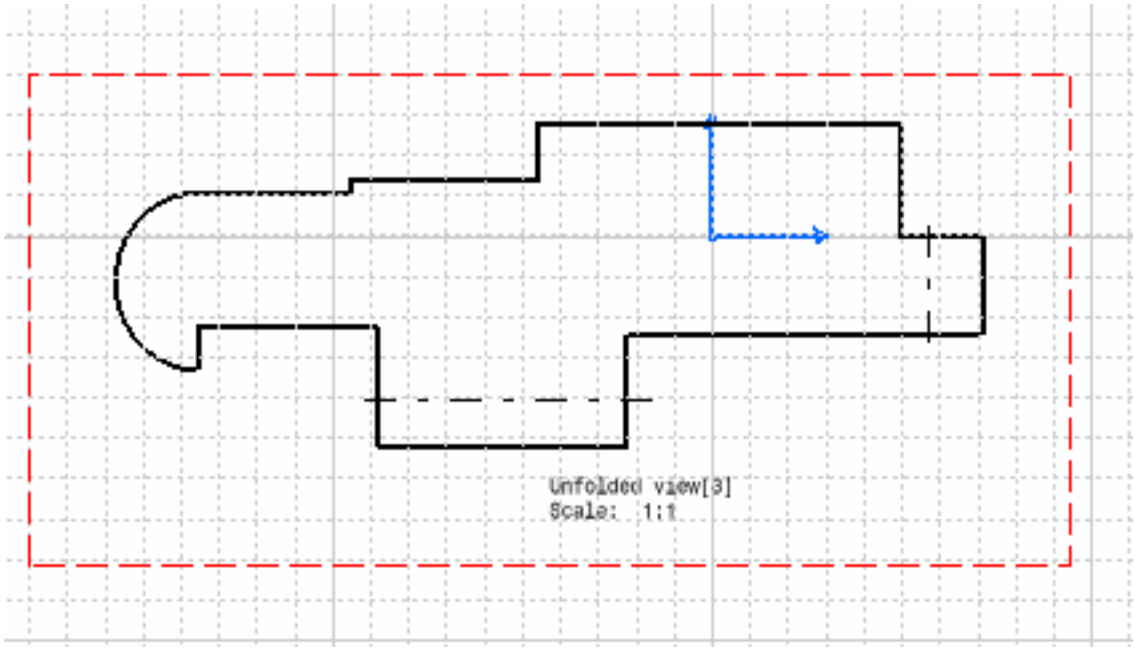
For more information about this workbench, refer to *Generative Drafting User's Guide*.

4. The drawing sheet appears.
5. Tile the windows horizontally using the **Window -> Tile Horizontally** menu item.
6. Select the **Unfolded View** icon  in the Projections toolbar from Generative Drafting Workbench.



This icon is added to the Projections toolbar provided the NEW Sheet Metal workbench is present.

7. Choose the **xy plane** in the Sheet Metal specification tree.
The unfolded view is displayed with the planar swept wall axes.





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Unfolding

Unfolded Sheet Metal parts can be displayed in two ways:

Folded/Unfolded View Access

Concurrent Access



Each Sheet Metal feature is created in a given view: folded, or unfolded. Editing a feature must be done in its definition view. If not, a message is automatically issued, prompting you to change views, before editing the feature.

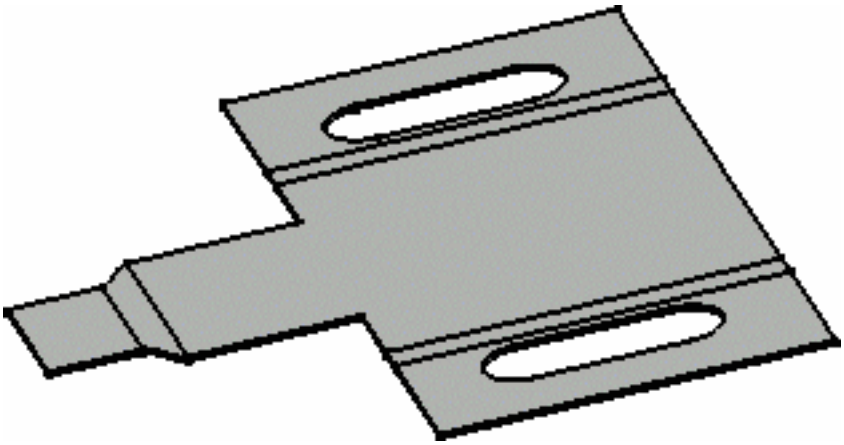
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Folded/Unfolded View Access

This task shows how to unfold the part.

1. Click the **Unfold** icon .

   The part is unfolded according to the reference wall plane or web, as shown below.



2. Click this icon again to refold the part for the next task.

- In Sheetmetal Design, bend limits and stamping are now displayed in the unfolded view. However, cutouts created on stamps are not.
- When designing in context, if a CATProduct document contains several Sheetmetal parts, only one part can be visualized in the unfolded view at a time.

Concurrent Access

This functionality is P2 for Sheetmetal Design.

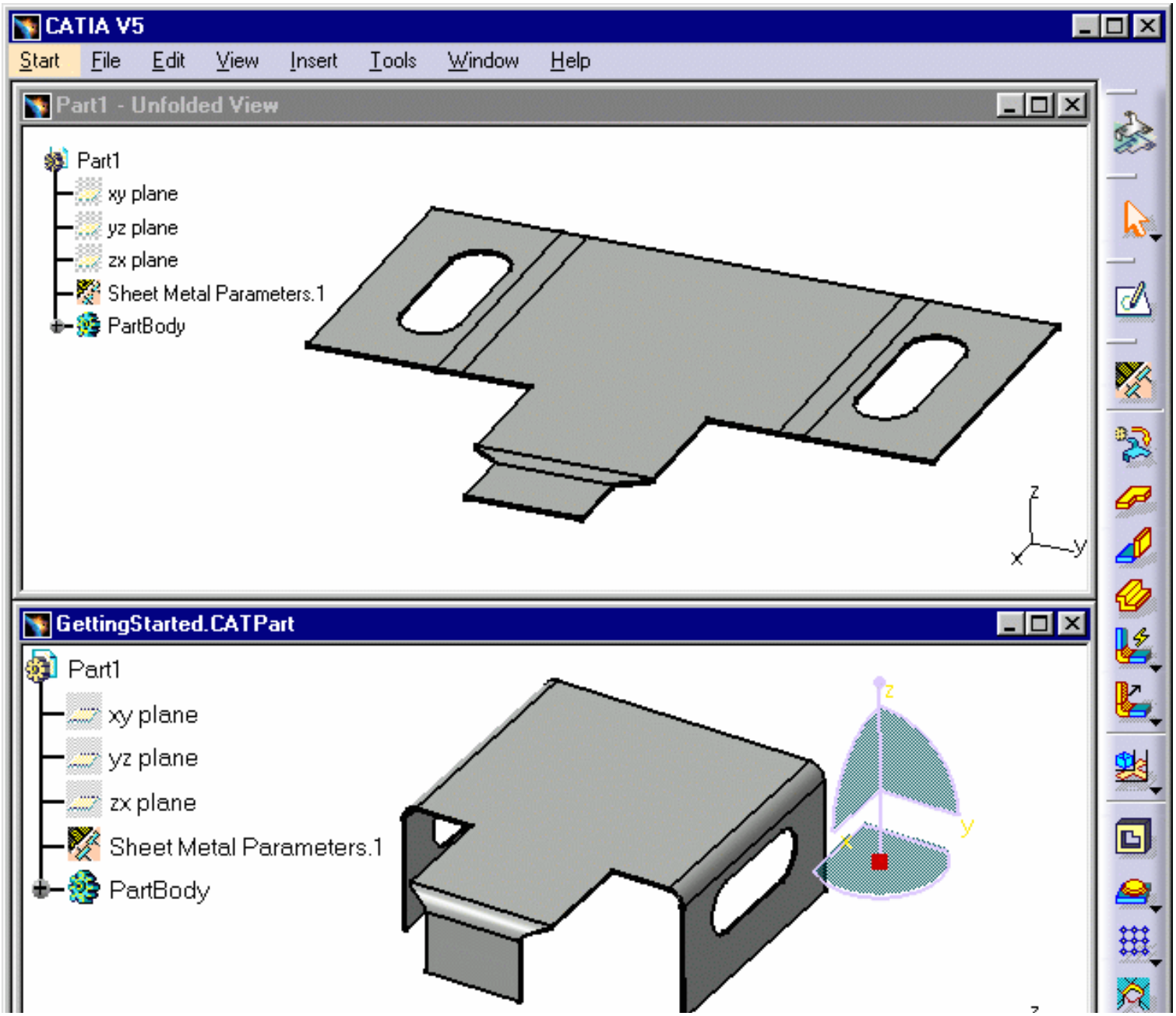
This task explains how to display the Sheet Metal part in two windows: one with the folded view, one with the unfolded view. Any modification in one window is displayed in the other window.

1. Click the **Multi-view** icon .

   The part is unfolded in a second window.

2. Choose the **Window -> Tile Horizontally** menu item.

   Both windows are tiled. Activate the window in which you want to work.





- Any modification in one view is taken into account in the other view enabling the user to make modifications in the best possible context.
- In the multi-view mode as in the standard unfolded view, all constraints are displayed in the geometrical views.

- Once in the Multi-view mode, the standard icon **Unfold** is not longer available.
- The Multi-view function is not available from a standard unfolded view.
- Only parts with bends can be unfolded.
- Cutting faces and open faces are not displayed in Multi-view mode (Sheetmetal Design)

Pockets

Creating a Cutout
Splitting Geometry

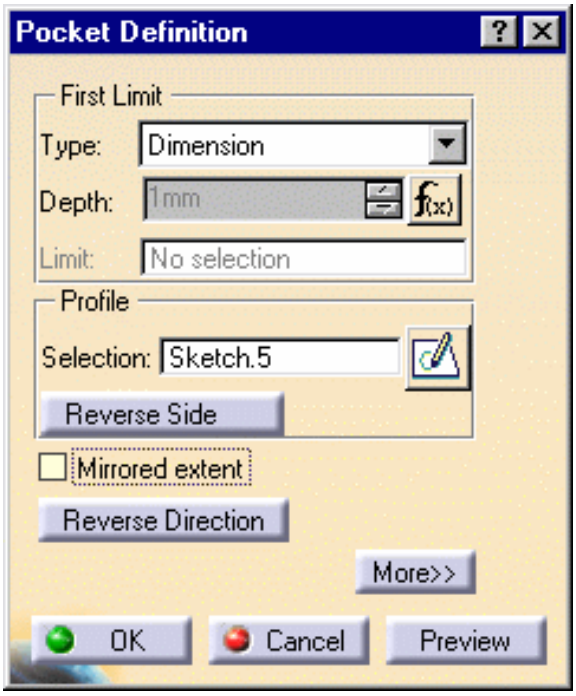
Back Up Next

Creating a Cutout

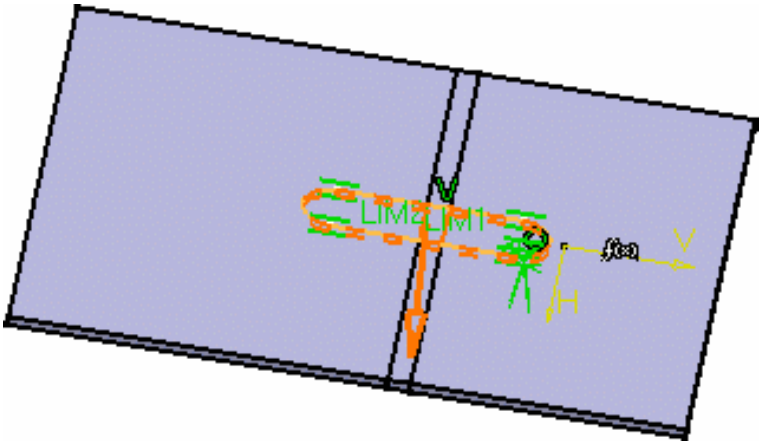
This task explains how to create a cutout in a wall.
Creating a cutout consists in extruding a profile and removing the material resulting from the extrusion.

Open the Cutout1.CATPart document.

1. Click the **Cutout** icon .
2. Select a profile.



The Pocket Definition dialog box is displayed and CATIA previews a cutout with default parameters.



Once the sketch is selected, you can modify it by clicking the Sketcher icon .

3. Select the type.

Several limit types are available:

- **Dimension**: the cutout depth is defined by the specified value
- **Up to next**: the limit is the first face the application detects while extruding the profile. This face must stops the whole extrusion, not only a portion of it, and the hole goes through material.
- **Up to last**: the last face encountered by the extrusion is going to limit the cutout
- **Up to plane**: the cutout is limited by the selected plane
- **Up to surface**: the cutout is limited by the selected surface

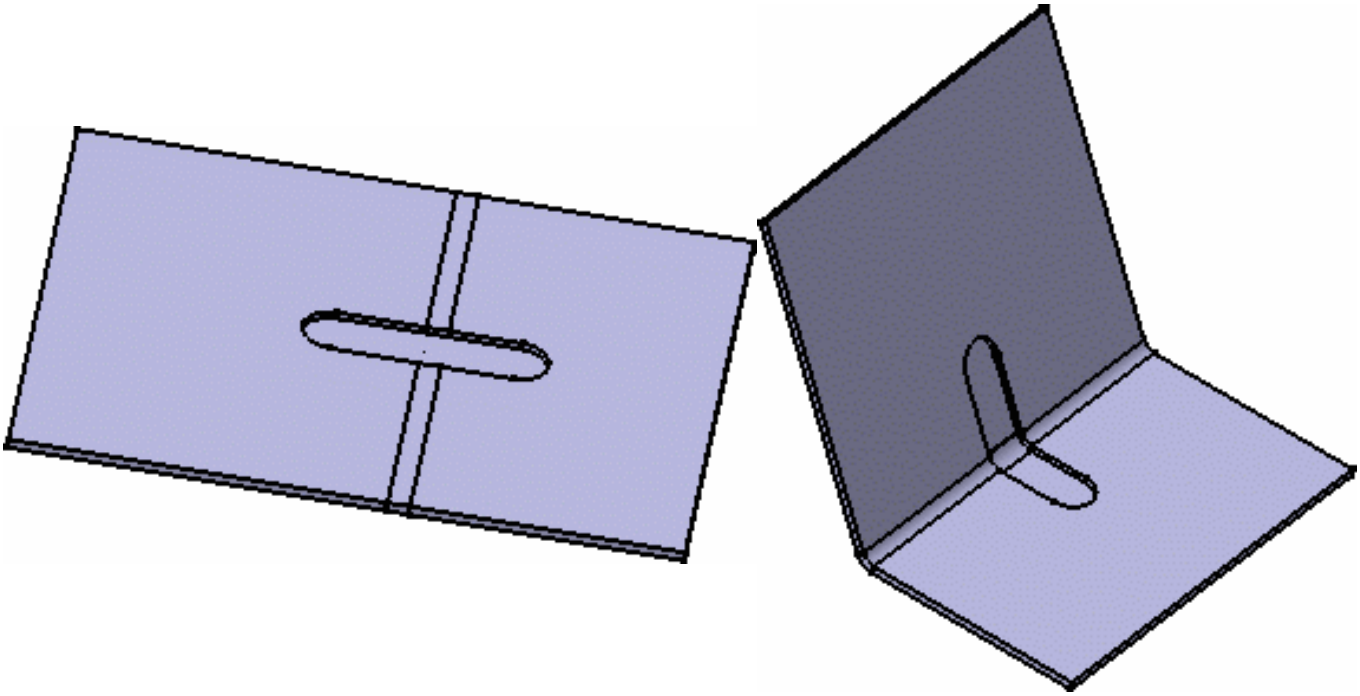
When an **Offset** value is required, it is the distance between the limiting element and the top face of the cutout, if the latter does not result in a complete hole through the material.

The LIM1 and LIM2 texts in the geometry area indicate the top and bottom limits of the cutout.

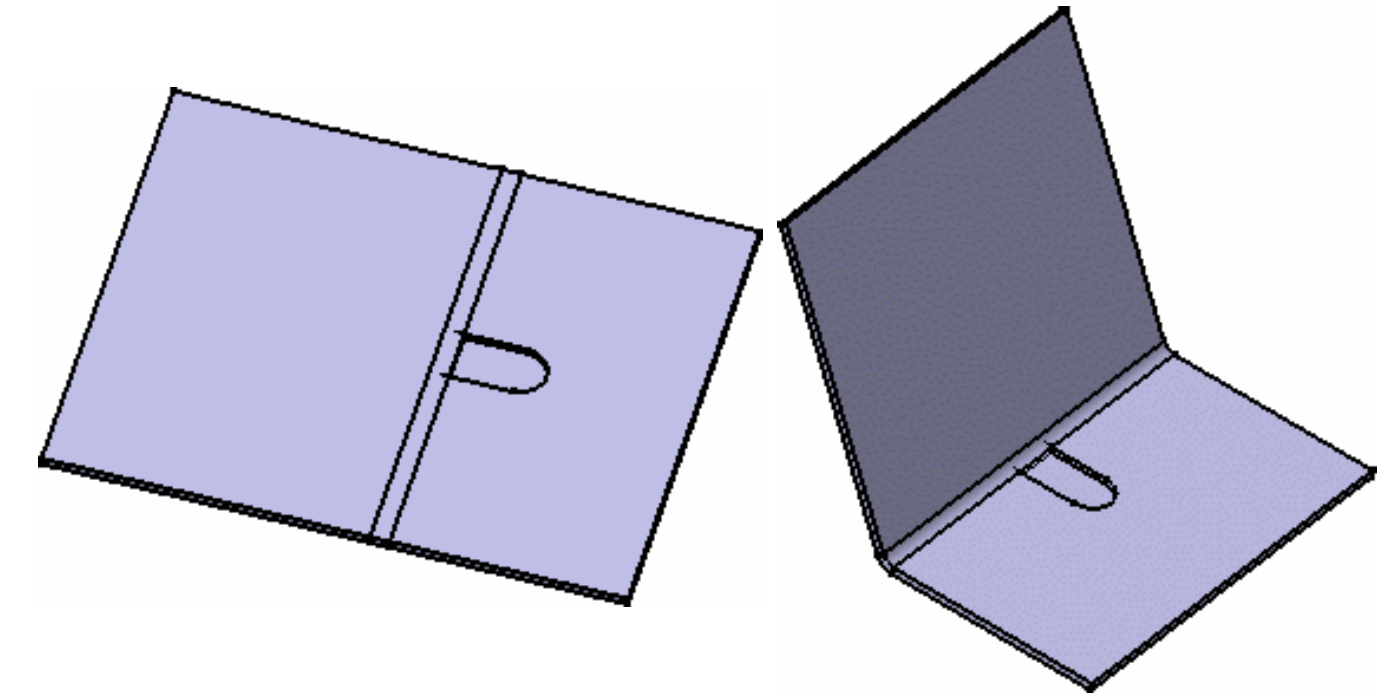
- The **Reverse Side** option lets you choose between removing the material defined within the profile, which is the application's default behavior, or the material surrounding the profile.
- The **Reverse Direction** option allows you to invert the direction of the extrusion pointed by the arrow.

4. Click OK in the Pocket Definition dialog box.

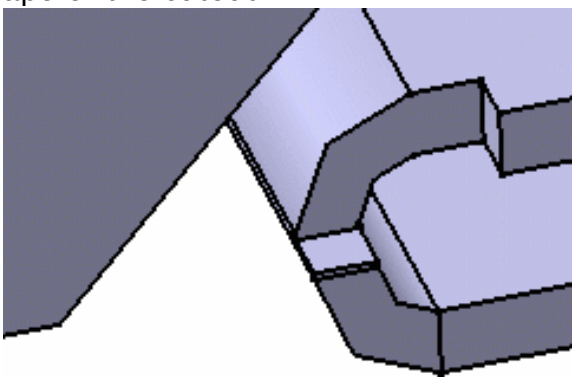
   The cutout is created.



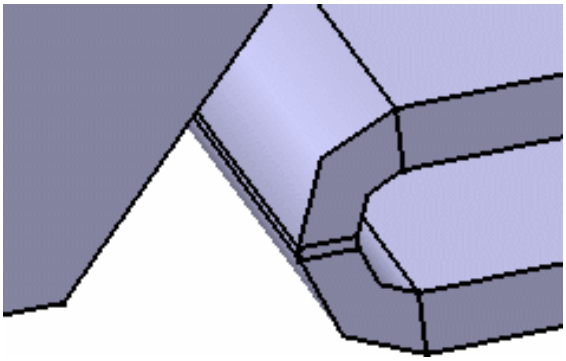
You may want to fold or unfold the part prior to creating the cutout, depending on the selected profile and the expected results. Selecting the same profile, the resulting cutout when created in the unfolded view is seen above, while below, the cutout was created in folded view.



However, you also have to be careful when designing the Part, not to create the cutout in the unfolded view as this may lead, when folding the part, to completely or partially removing another section of the Part. It is best, whenever possible, to create the wall based on a sketch integrating the shape of the cutout.



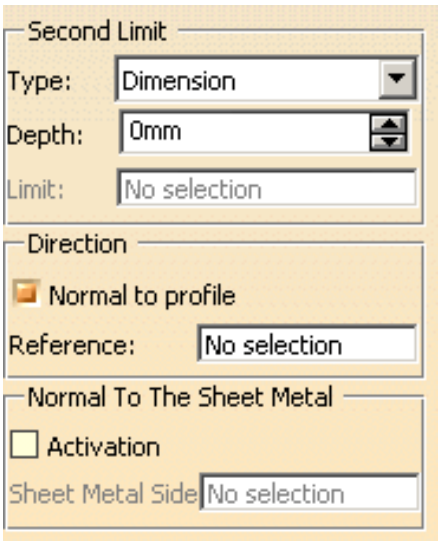
Cutout removing material



Cutout not removing material based on wall's sketch modification

4. Click **More>>** to display the maximum information.

- You can define 'Limit2' as the second limit by using the same options as for Limit 1 (Dimension, Up to last, up to plane, up to surface).
- You can choose between a direction normal to the profile or define a new direction by selecting geometry in the Reference field.
- You can choose the cutout to be normal to the sheet metal part: check the **Activation** button and define the sheet metal side.



5. Select the **Support** (here we chose the wall)

May you want to create a cutout on an overlapping element or a bend with radius=0, either choose the top skin of the element, or unfold the part to create the cutout.

- Refer to the *Component Catalog Editor* documentation to have further information on how to use catalogs.
- Refer to the Create a Pocket task in the *Part Design User's Guide* for further details on how to create cutouts.



Up Next

Splitting Geometry

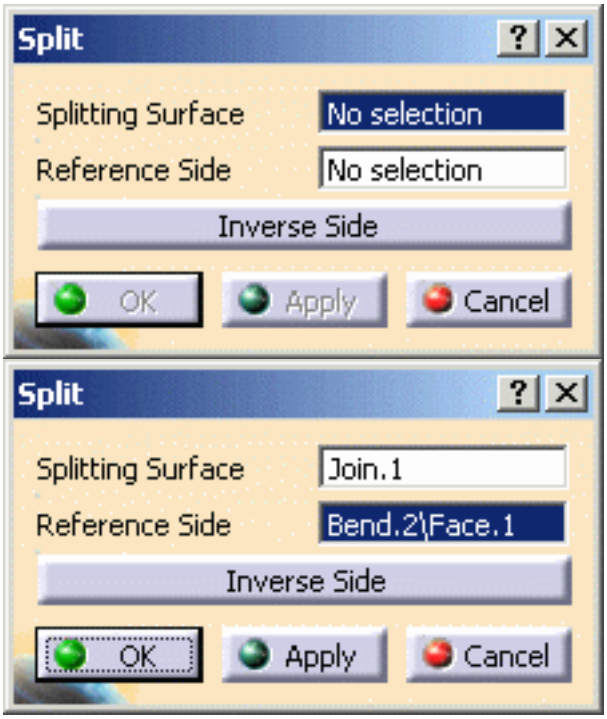
This functionality is only available with SheetMetal Design.

This task shows how to create a split normal to a sheet metal part by means of a cutting surface.

Open the Split1.CATPart document.

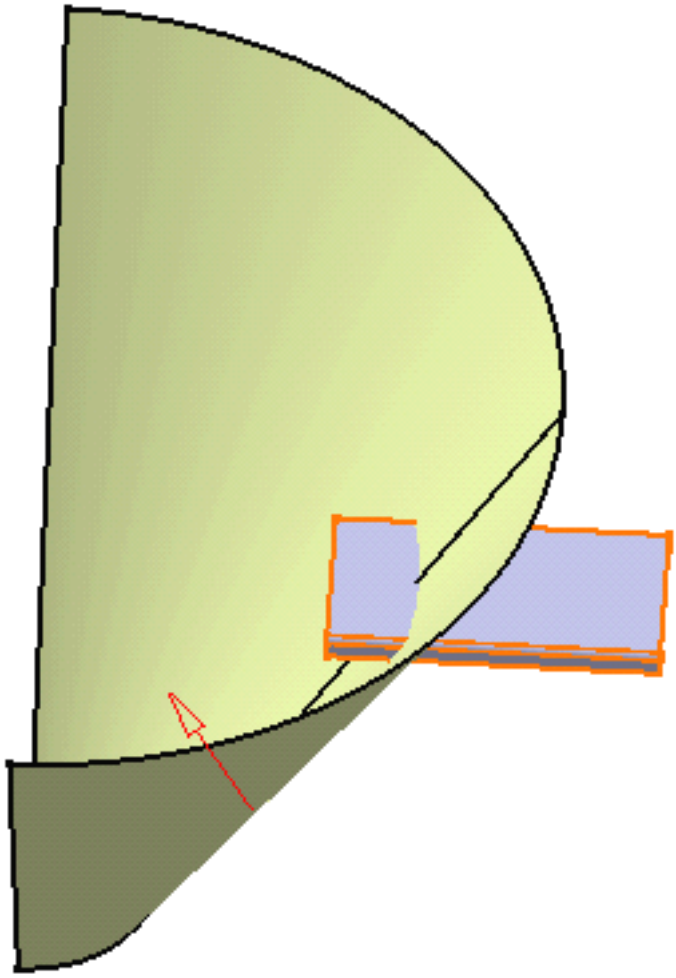
1. Click the **Split** icon .

   The Split dialog box appears.



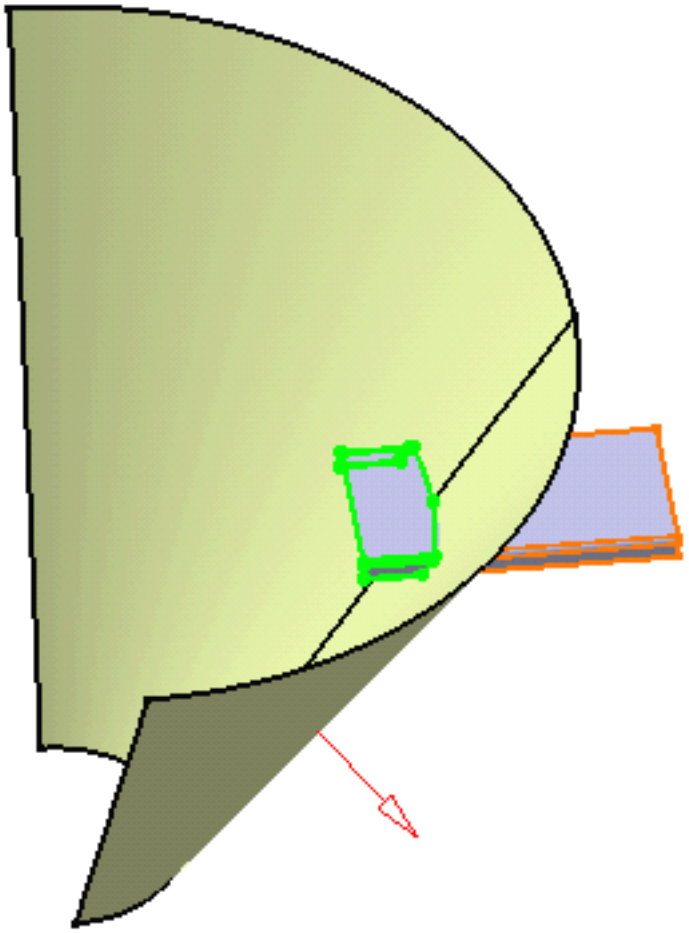
2. Select the **Splitting Surface**.
3. Select the **Reference Side** of the element to split.

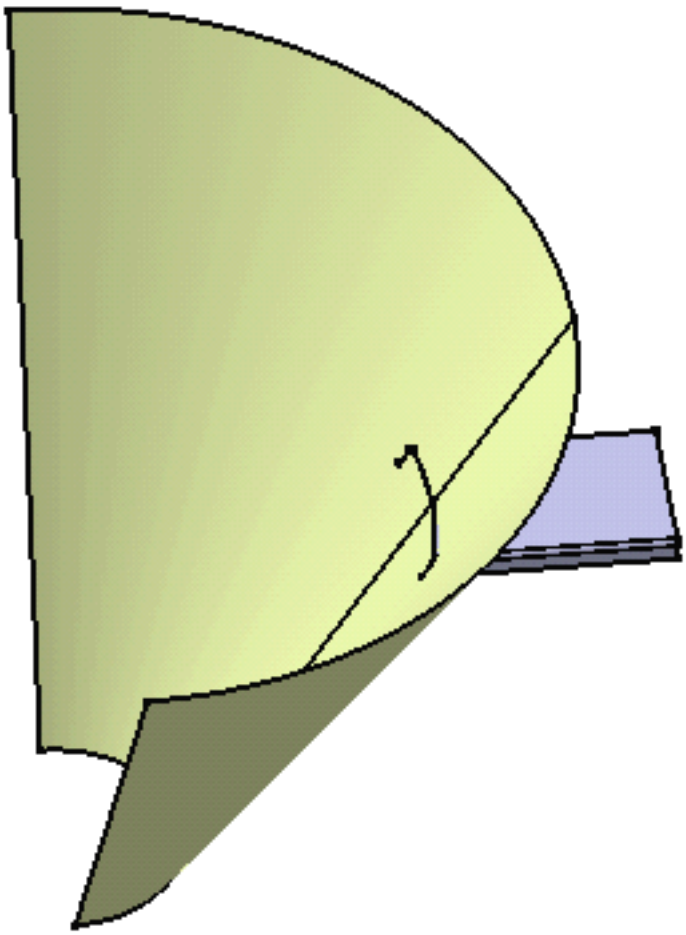
The red arrow shows the side to keep



4. Click the **Inverse Side** button to keep the other side.

5. Click **Apply** to preview the split: the side to be removed is highlighted in green.





6. Click **OK** to split the element.

   The created element (identified as Split.xxx) is added to the specification tree.

You can create a split using the unfolded view of the part.



Back Up

Stamping

This section explains and illustrates how to create and use various kinds of stamps.

- Stamps must be created on walls, or walls on edge, except for the stiffness rib that is to be created on a bend.
- If a stamp is created over the limit between several supports, such as walls, bends, and so forth, this stamp is not visible on the unfolded view.
- When unfolding a part, only the largest imprint of the stamp is retained on the stamped wall.
- Stamps cannot be created on an unfolded part.

Standard Features
Creating User-defined Stamping Features

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Creating Standard Stamping Features

This section explains and illustrates how to create and use various kinds of stamps.
The table below lists the information you will find.


Create a point stamp: select a point on a face, and set the stamping parameters.


Create an extruded hole: select a point on a face, and set the stamping parameters.


Create a curve stamp: select a sketch, and set the stamping parameters.


Create a surface stamp: select a sketch, and set the stamping parameters.


Create a bridge: select a point on a face, set the stamping parameters, and select an edge to give the bridge orientation.


Create a louver: select a sketch, and set the stamping parameters.


Create a stiffness rib: select the external surface of a bend, and set the stamping parameters.

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Creating a Point Stamp

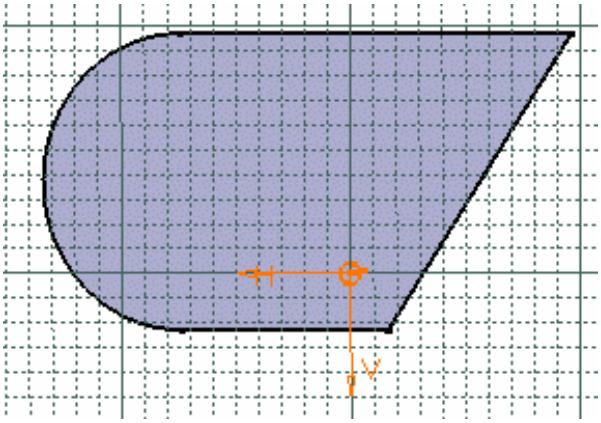
This task shows you how to create a point stamp by specifying the punch geometrical parameters.

Open the Stamping.CATPart document from the samples directory.

1. Click the **Point Stamp** icon .

2. Select a point on the top face.

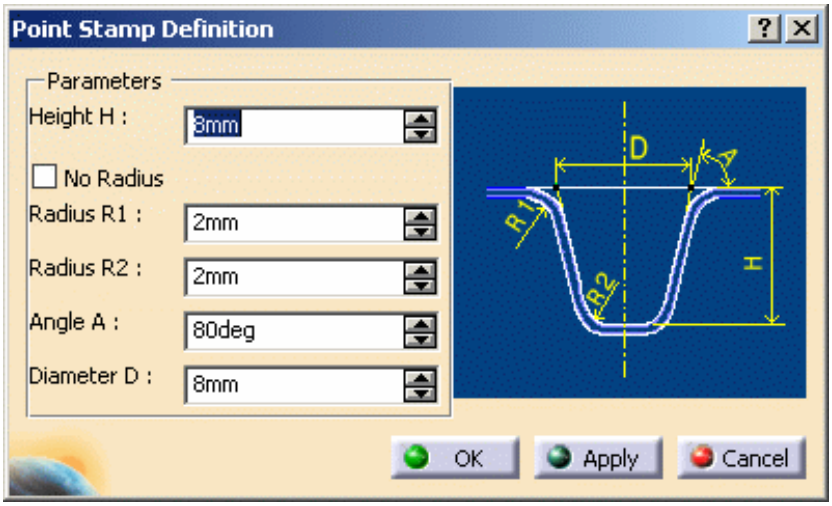
   A grid is displayed to help you position the point stamp.



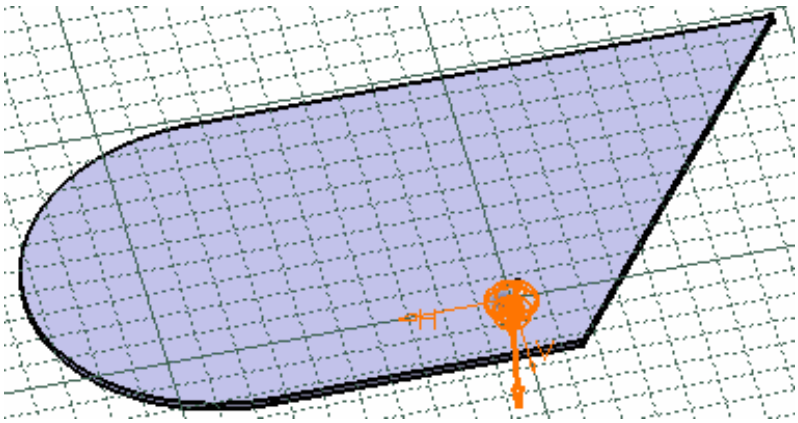
The Point Stamp Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

   - Height H
   - Radius R1
   - Radius R2
   - Angle A
   - Diameter D

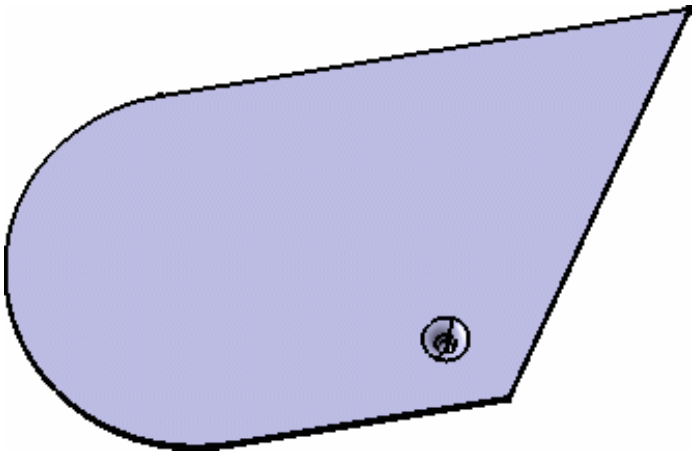


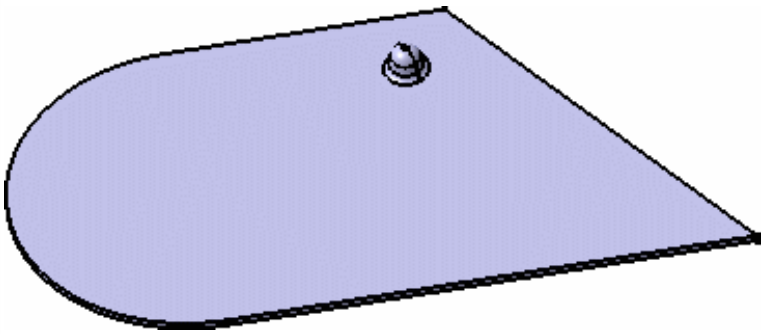
4. Click **Apply** to preview the point stamp.



5. Click **OK** to validate.

   The point stamp (identified as Point Stamp.xxx) is created and the specification tree is updated accordingly.





Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the bridge stamp without a fillet.



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Creating an Extruded Hole

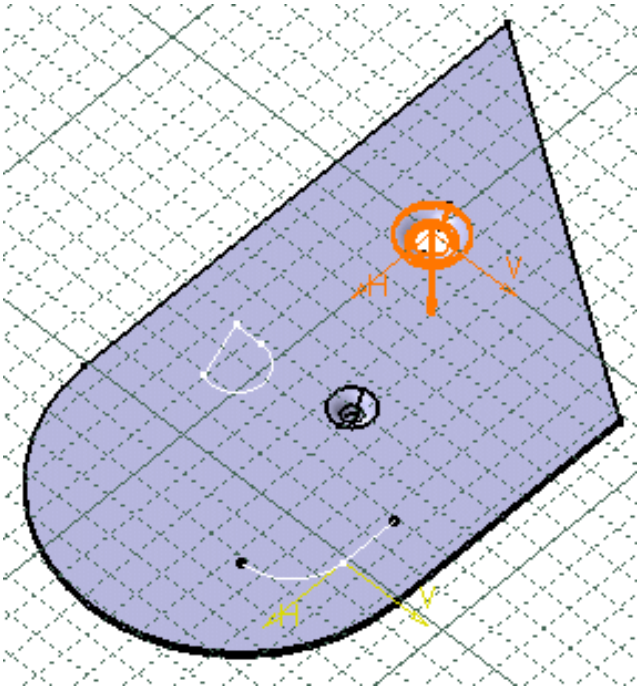
This task shows you how to create an extruded hole by specifying the punch geometrical parameters.

The Stamping.CATPart document is still open from the previous task.
If not, open the Stamping8.CATPart document.

1. Click the **Extruded Hole** icon .

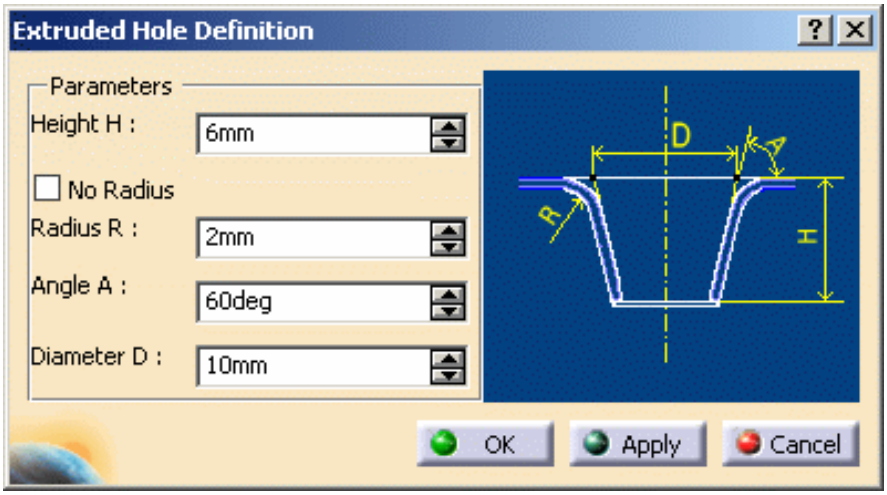
2. Select the surface where you want to place the hole.

   A grid is displayed to help you position the flanged hole and the Flanged Hole Definition dialog box opens, providing default values.



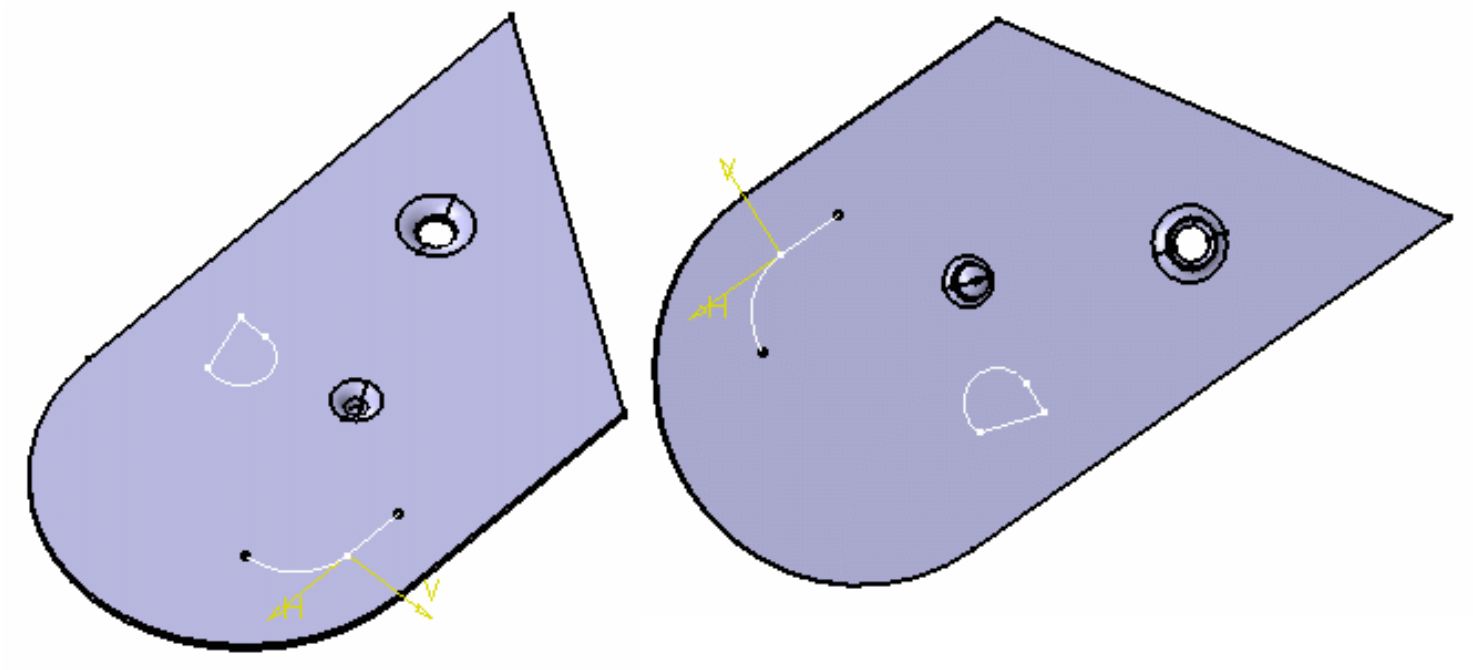
3. Change the value in the different fields, if needed:

   - Height H
   - Radius R
   - Angle A
   - Diameter D

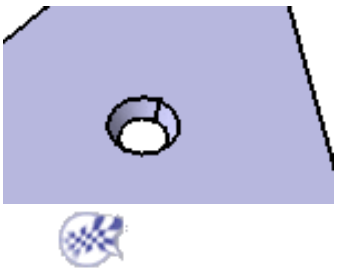


4. Click **Apply** to preview the flanged hole.
5. Click **OK** to validate.

The extruded hole (identified as Extruded Hole.xxx) is created and the specification tree is updated accordingly.



Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the point stamp without a fillet.



Back Up Next

Creating a Curve Stamp

This task shows you how to create a curve stamp by specifying the punch geometrical parameters.

The Stamping.CATPart document is still open from the previous task.
If not, open the Stamping3.CATPart document from the samples directory.

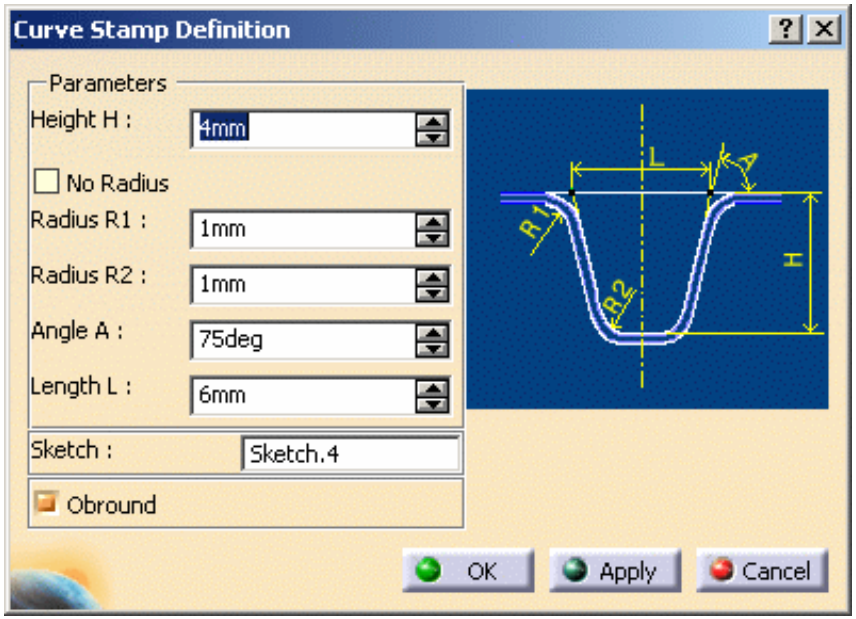
1. Click the **Curve Stamp** icon .

2. Select **Sketch-for-Curve-Stamp**, the curve previously defined.

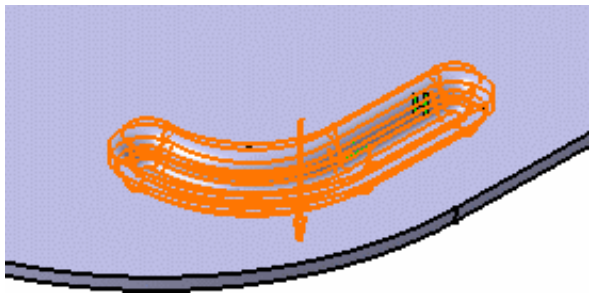
   The Curve Stamp Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

   - Height H: the total height
   - Radius R1: the outer bend radius
   - Radius R2: the outer bend radius
   - Angle A: the stamping draft angle
   - Length L: the stamps' maximum width

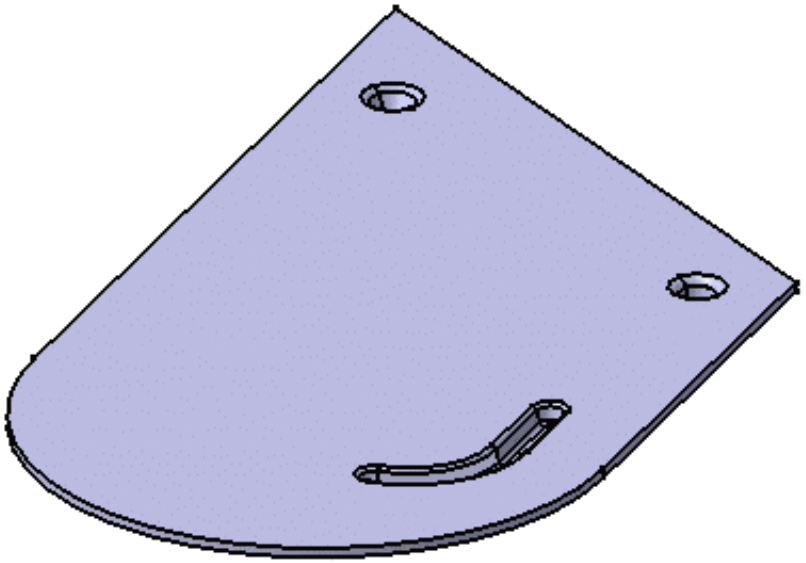


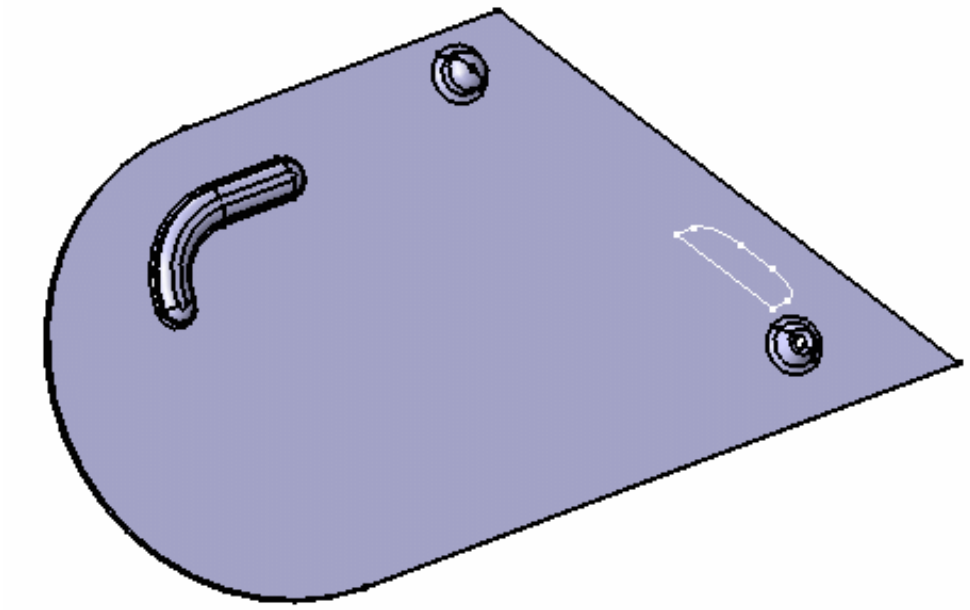
4. Click Apply to preview the curve stamp.



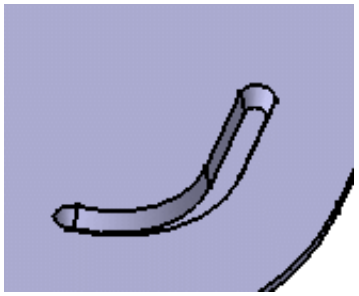
5. Click **OK** to validate.

The curve stamp (identified as Curve Stamp.xxx) is created and the specification tree is updated accordingly.

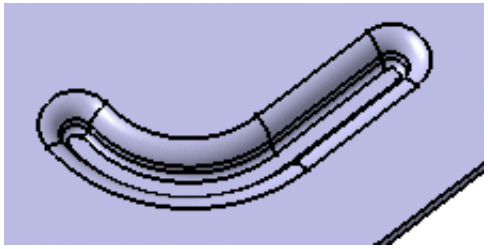




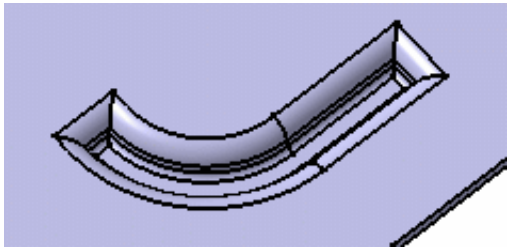
- Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the curve stamp without a fillet.



- Check the **Obround** option to round off the edges of the curve stamp.



Obround option checked



Obround option unchecked

Back Up Next

Creating a Surface Stamp

This task shows you how to create a surface stamp by specifying the punch geometrical parameters.

The Stamping.CATPart document is still open from the previous task.
If not, open the Stamping4.CATPart document from the samples directory.

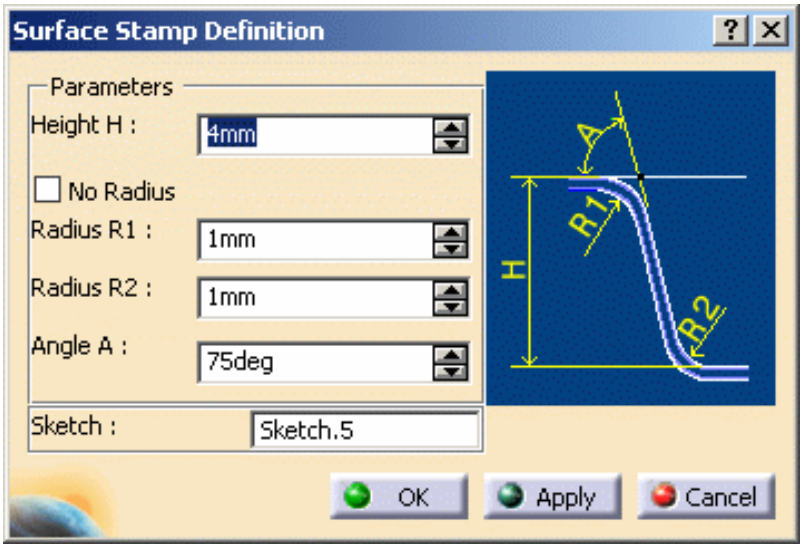
1. Click the **Surface Stamp** icon .

2. Select **Sketch-for-Surface-Stamp**, the profile previously defined.

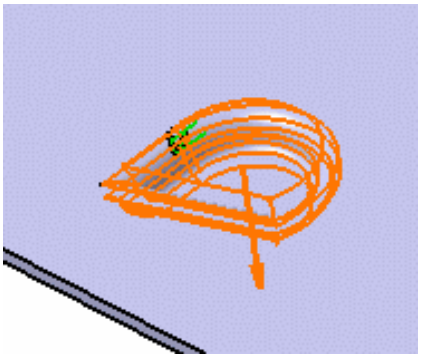
   The Surface Stamp Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

   - Height H
   - Radius R1
   - Radius R2
   - Angle A

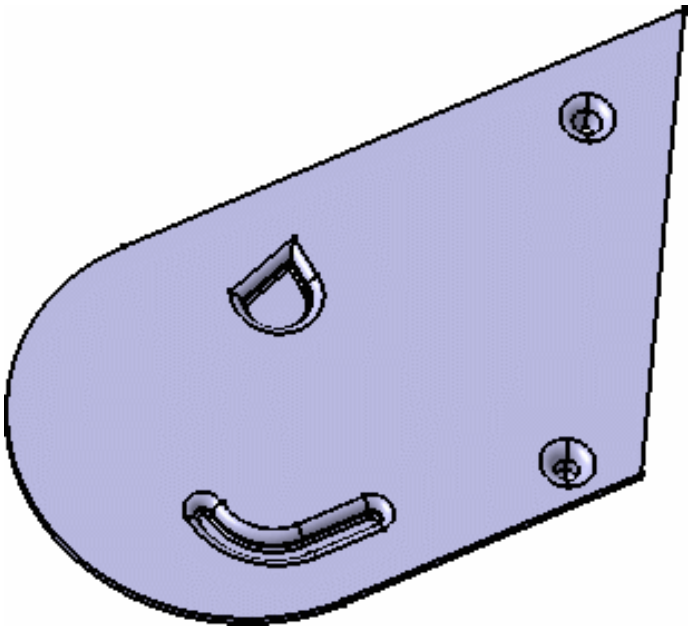


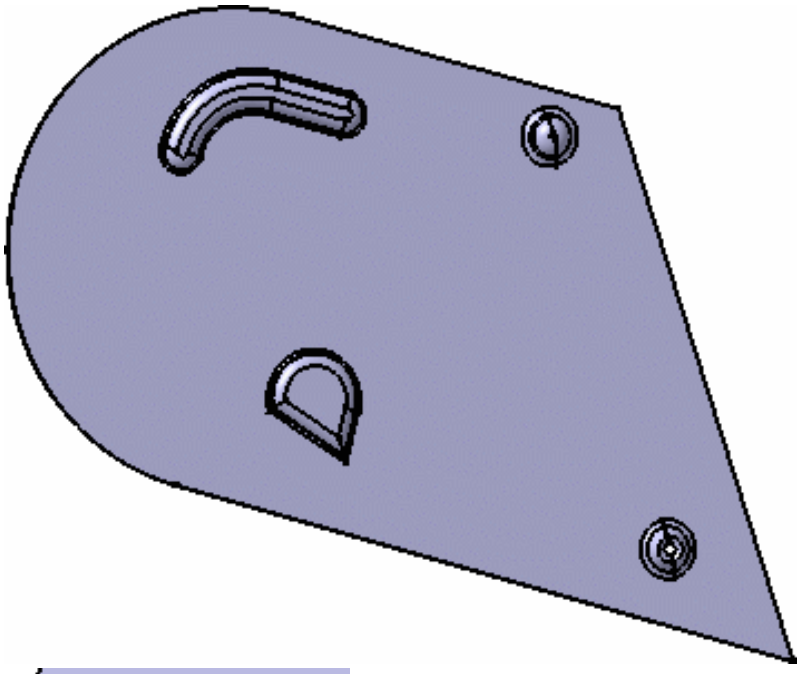
4. Click **Apply** to preview the surface stamp.



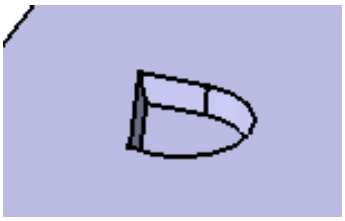
5. Click **OK** to validate.

   The surface stamp (identified as Surface Stamp.xxx) is created and the specification tree is updated accordingly.





Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the surface stamp without a fillet.



Back Up Next

Creating a Bridge

This task shows you how to create a bridge by specifying the punch geometrical parameters.

The Stamping.CATPart document is still open from the previous task.
If not, open the Stamping5.CATPart document from the samples directory.

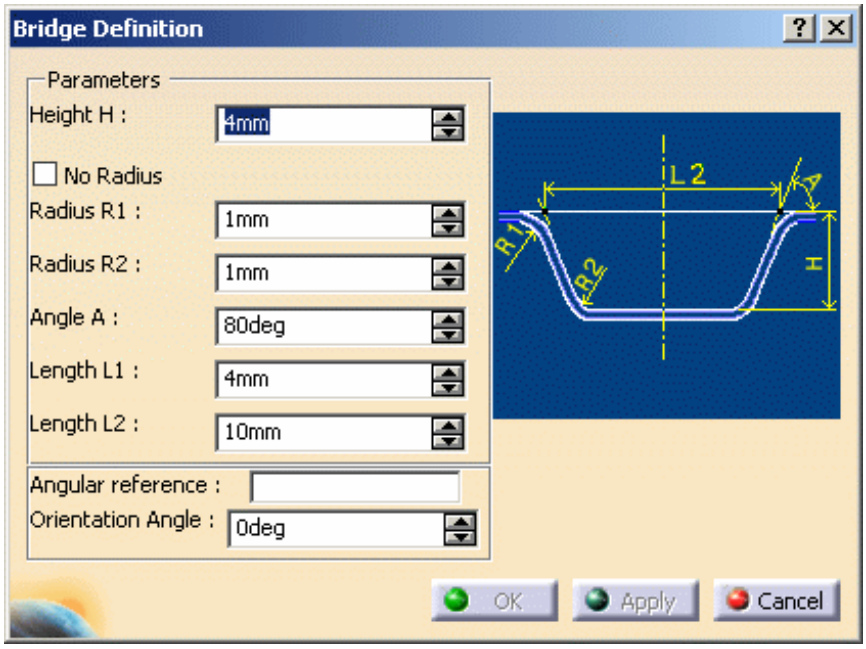
1. Click the **Bridge** icon .

2. Select a point on the top face where you want to place the bridge.

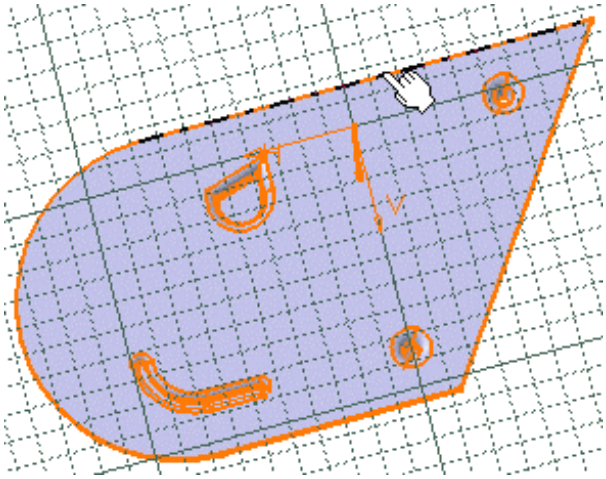
   The Bridge Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

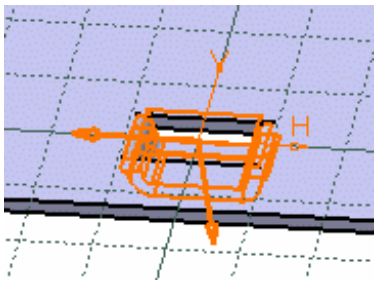
   - Height H
   - Radius R1
   - Radius R2
   - Angle A
   - Length L1
   - Length L2



4. Select an edge to give the direction of the bridge.

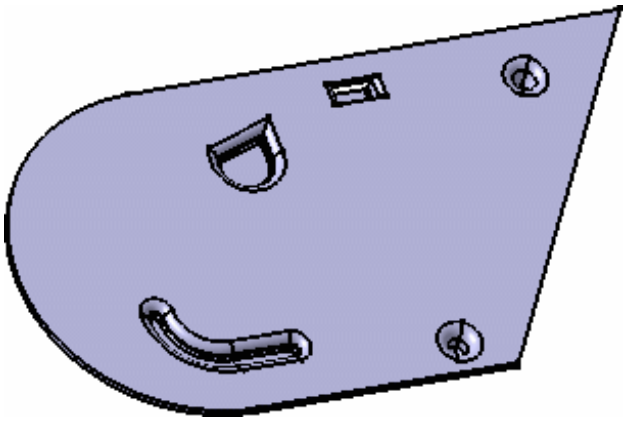


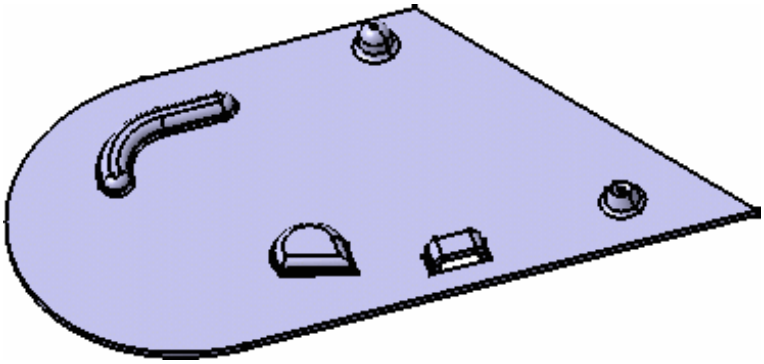
5. Click Apply to preview the bridge.



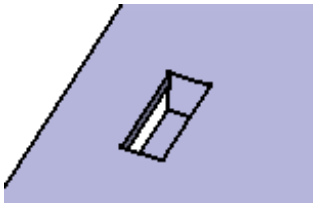
6. Click **OK** to validate.

The bridge (identified as Bridge.xxx) is created and the specification tree is updated accordingly.





Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the bridge stamp without a fillet.



Back Up Next

Creating a Louver

This task shows you how to create a louver by specifying the punch geometrical parameters.

Open the Stamping6.CATPart document from the samples directory.

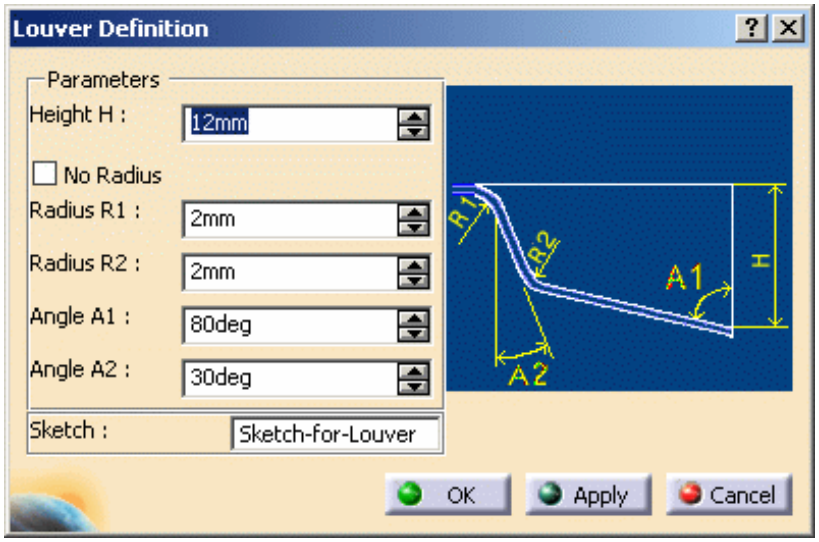
1. Click the **Louver** icon .

2. Select **Sketch-for-Louver**, a profile previously defined on **Wall.2**. The **Louver Definition** dialog box opens, providing default values.

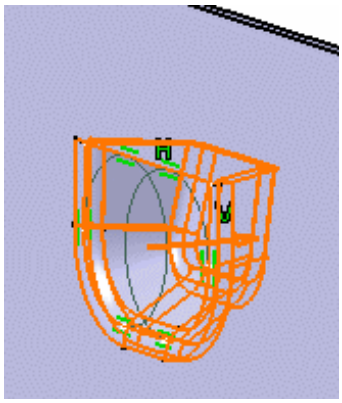
The louver opening face is represented in the sketch by the element that does not present any tangency continuity with the other lines/curve segments of the sketch. In case there are several non-continuous elements, the first one is used as the opening face.

3. Change the value in the different fields, if needed:

   - Height H
   - Radius R1
   - Radius R2
   - Angle A1
   - Angle A2

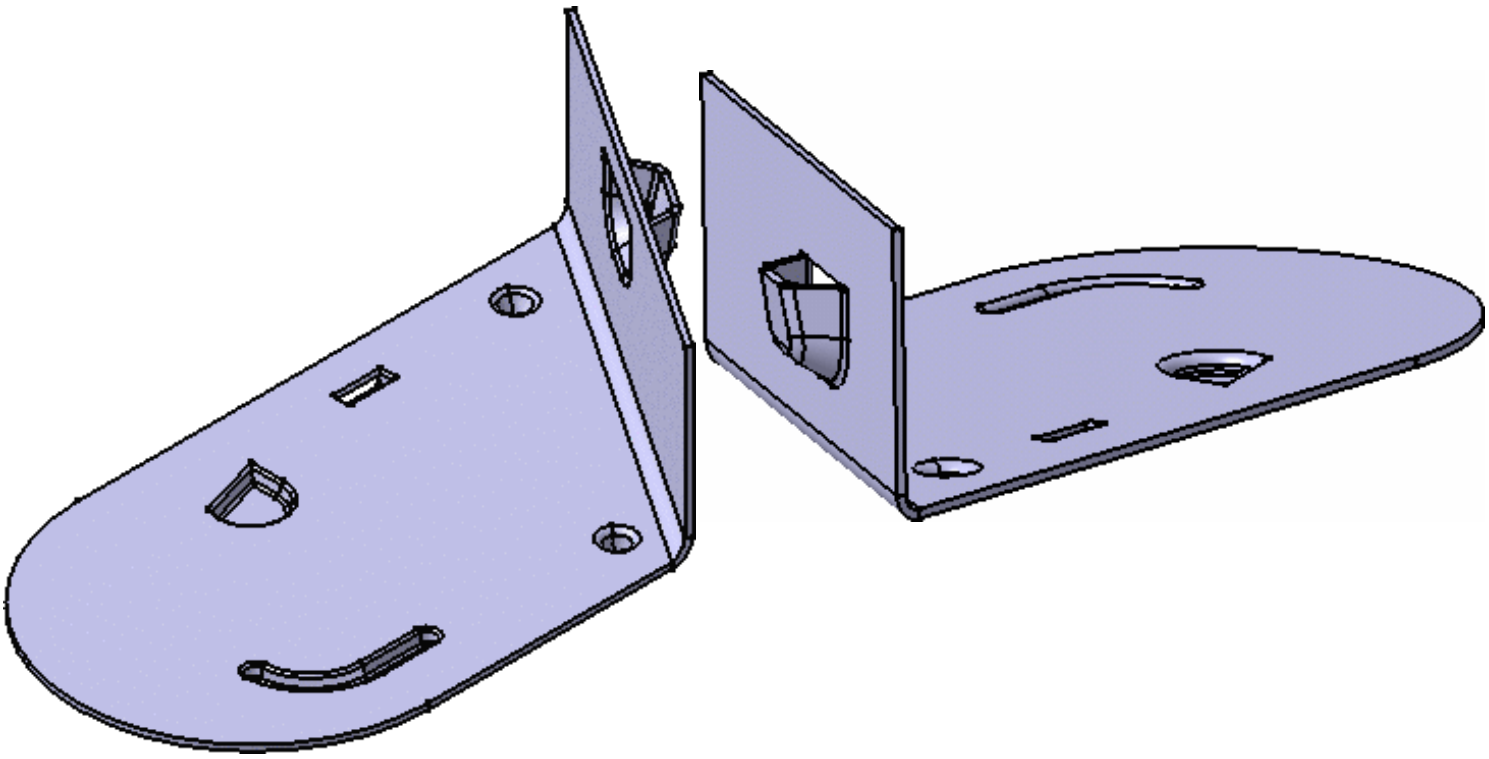


4. Click **Apply** to preview the louver.

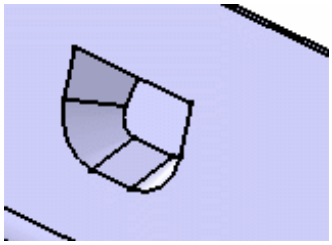


5. Click **OK** to validate.

   The louver (identified as Louver.xxx) is created and the specification tree is updated accordingly.



Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the louver stamp without a fillet.



Back Up Next

Creating a Stiffening Rib

This task shows you how to create a stiffness rib by specifying the punch geometrical parameters.

Open the Stamping7.CATPart document from the samples directory.

1. Click the **Stiffness Rib** icon .

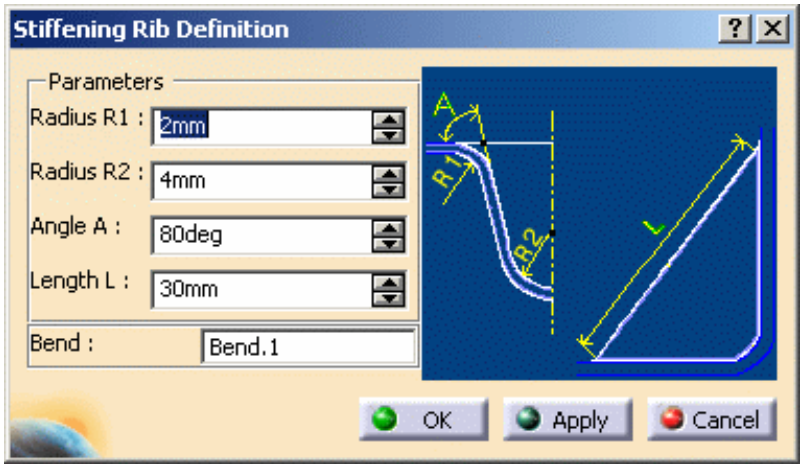
2. Select the external surface of Bend.1, where you want to place a stiffener.
   Note that the stiffener will always be centered on the bend radius, wherever the point may be along the curve.

   A grid is displayed.

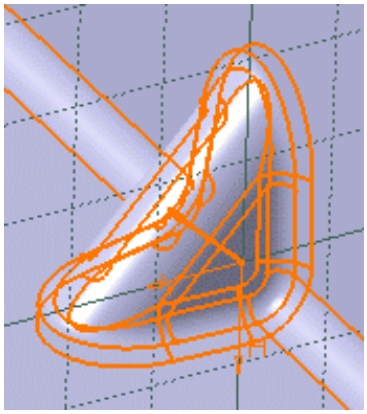
   The Stiffening Rib Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

   - Radius R1
   - Radius R2
   - Angle A
   - Length L

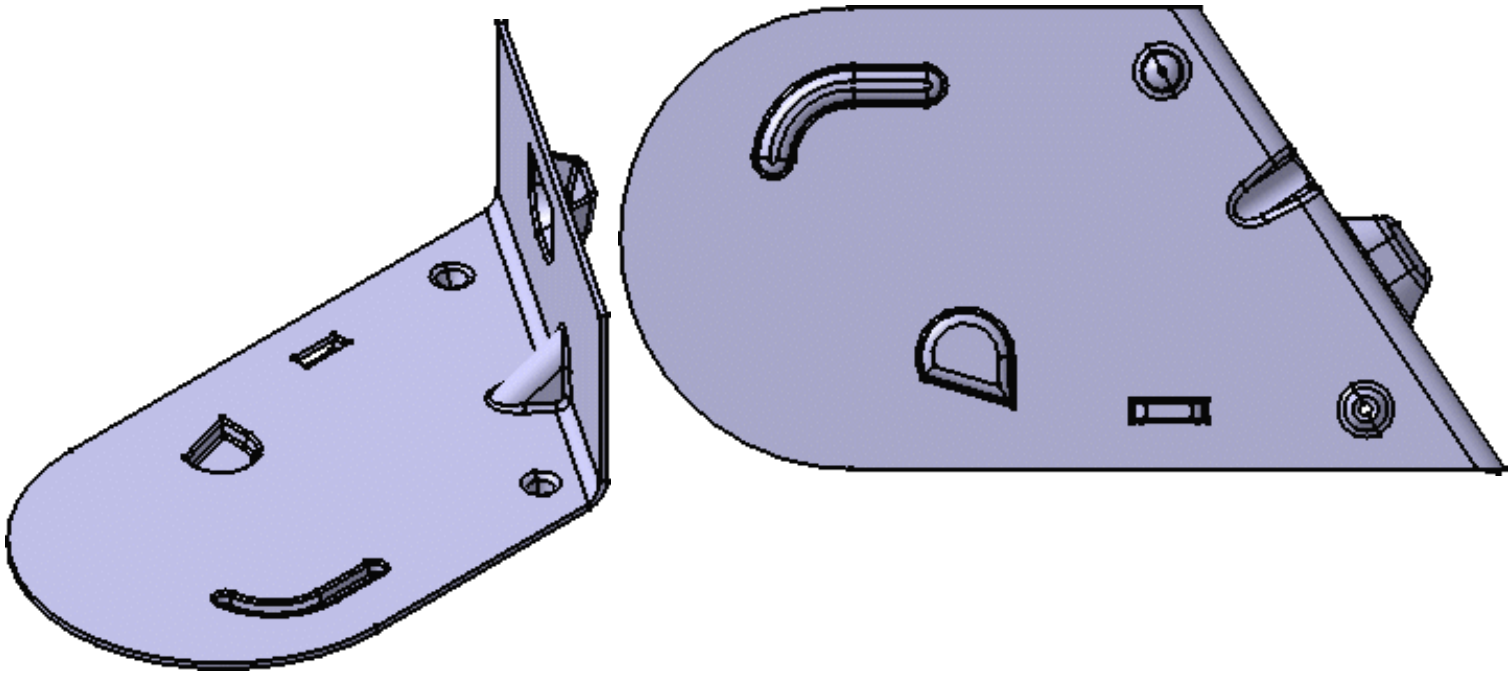


4. Click **Apply** to preview the stiffness rib.



5. Click **OK** to validate.

   The stiffening rib (identified as Stiffnening Rib.xxx) is created and the specification tree is updated accordingly.



Check the **No radius** option to deactivate the Radius R1 and R2 values, and to create the stiffening rib without a fillet.

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Creating User-Defined Stamping Features

Two user-defined stamping features are available:

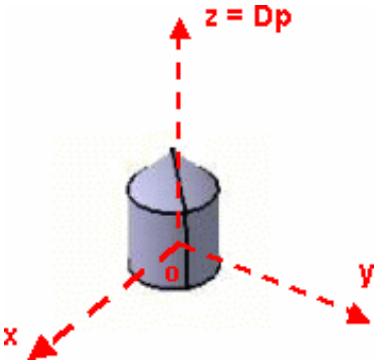
- Create a punch and a die: define the punch and die features, select a wall, choose the punch and die as stamping elements, select an edge on the wall and give an angle for orientation purposes.
- Open and cut faces : define the punch, select a wall, define the cutting faces and opening faces of the punch, select an edge on the wall and give an angle for orientation purposes.
- Edit a user-defined stamp: double-click the existing stamp and change its type, or select, or remove cutting and opening faces

What You Should Know

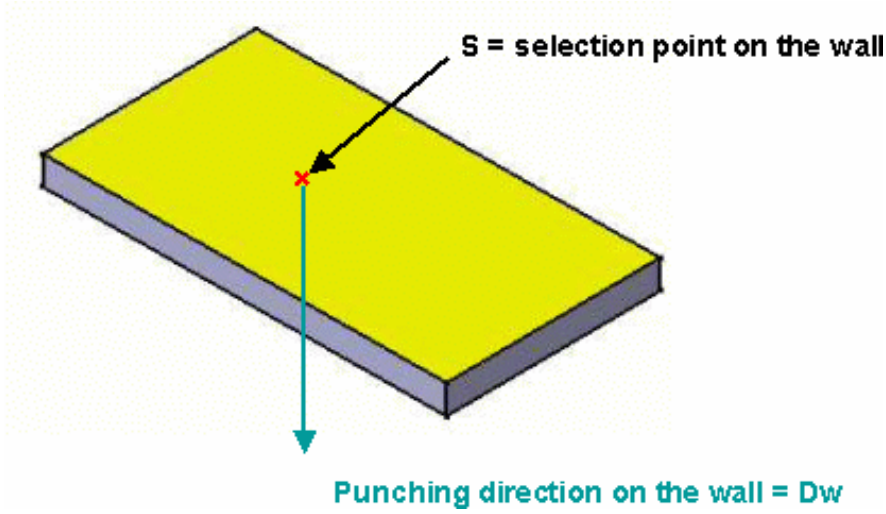
In both tasks illustrating either a stamp based on a punch and a die, or a punch with cutting and opening faces, the punch positioning is defined as below:

Defining the Punch in Relation to the Wall to be Stamped

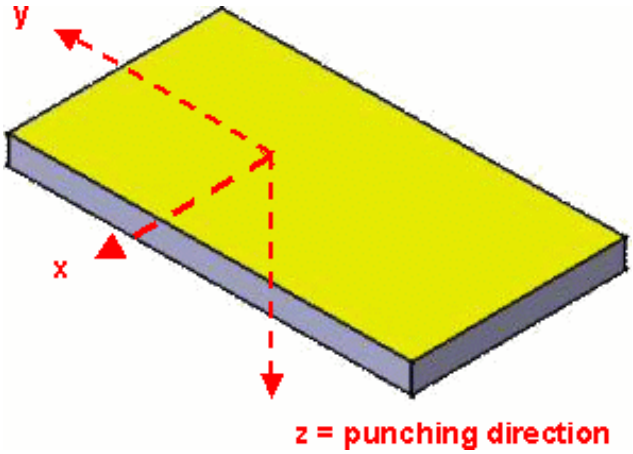
The punch is defined within the absolute (default) axis-system of the .CATPart document. (o, x, y, z) is the axis associated with the punch. The punching direction on the punch (*Dp*) must be equal to z.

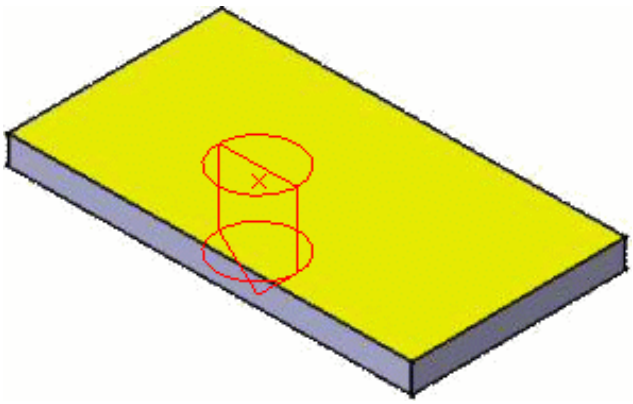


The punching direction on the wall (*Dw*) is normal to the selected wall face, and is oriented from the selected wall face towards the opposite face.



The punch is applied matching *Dp* on *Dw* and matching the punch's (x, y) plane onto the selected wall face:



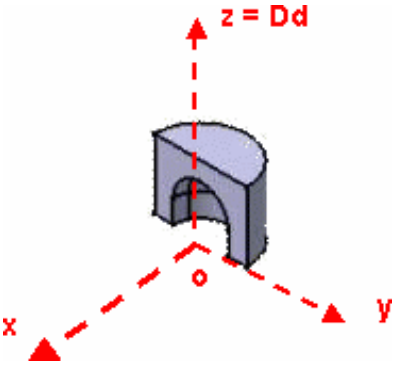


Defining the Die in Relation to the Wall to be Stamped

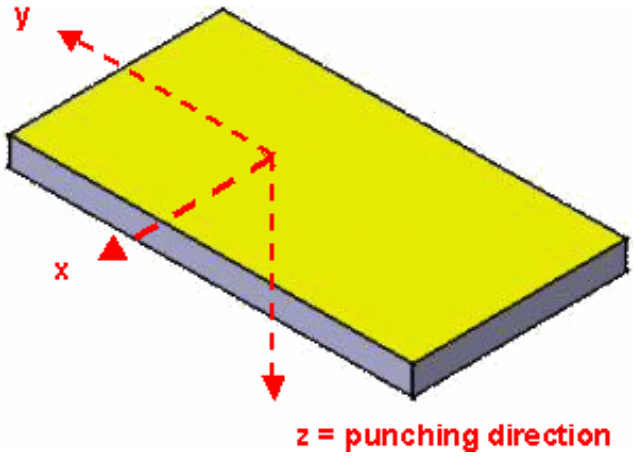
This is useful only when defining a punch a die, and does not apply to punches with cutting and opening faces.

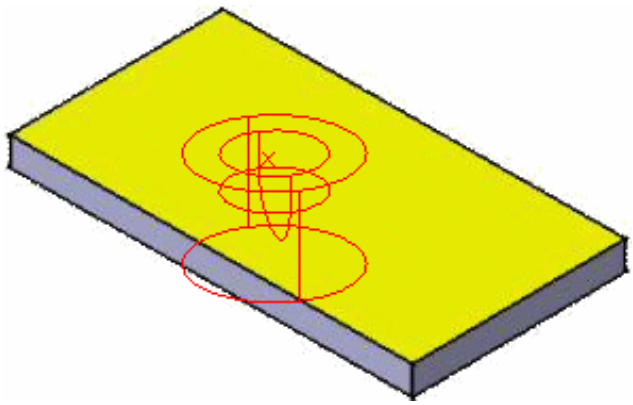
The die is also defined within the absolute (default) axis-system of the .CATPart document. (o, x, y, z) is the axis associated with the punch. The punching direction on the die (*Dd*) must be equal to z.

The illustration is a section view of the die.



The die is applied matching *Dd* on *Dw* and matching the die's (x, y) plane onto the selected wall face:





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Punch and Die

This task explains how to create a stamp from punch and die features.

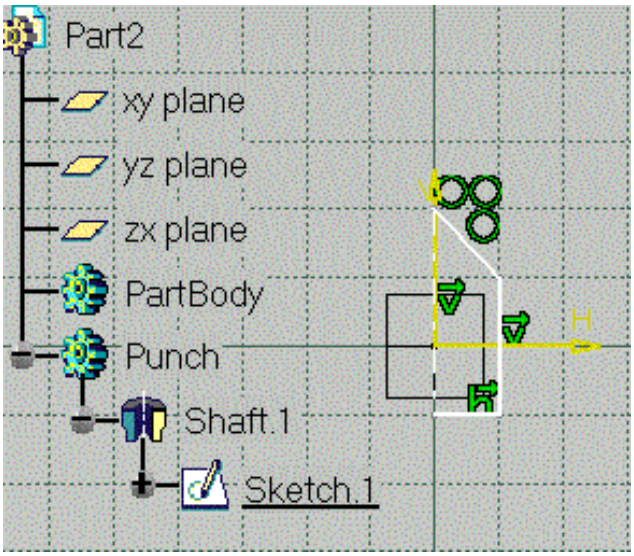
First, you will define a punch and a die in Part Design, in the absolute axis-system.
Then, in a Sheet Metal part, you will bring the punch and the die features (and their axis system) to a point you have selected. If necessary, you will define a rotation of the axis system from a reference line.

This user-defined stamping can not be combined with the Opening and Cutting Faces approach.

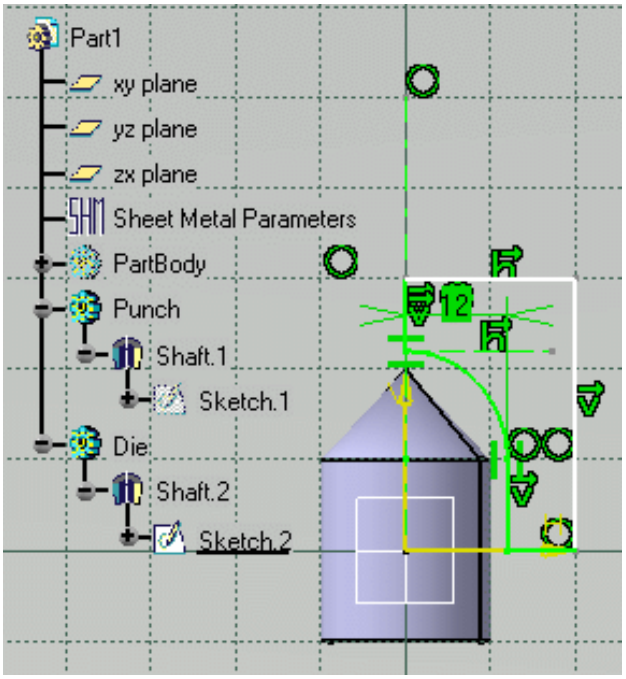
All .CATParts are available from the samples directory (PunchDie1.CATPart, Punch1.CATPart and Die1.CATPart or NEWPunchDie1.CATPart, NEWPunch1.CATPart and NEWDie1.CATPart for New SheetMetal Design)

1. Start the Part Design application.
2. Insert a PartBody (menu **Insert -> Body**) to define the punch.
3. Enter the sketcher select the yz plane, and draw the profile of the punch, and a rotation shaft.

   The punch must be oriented as described in Defining the Punch in Relation to the Wall to be Stamped.



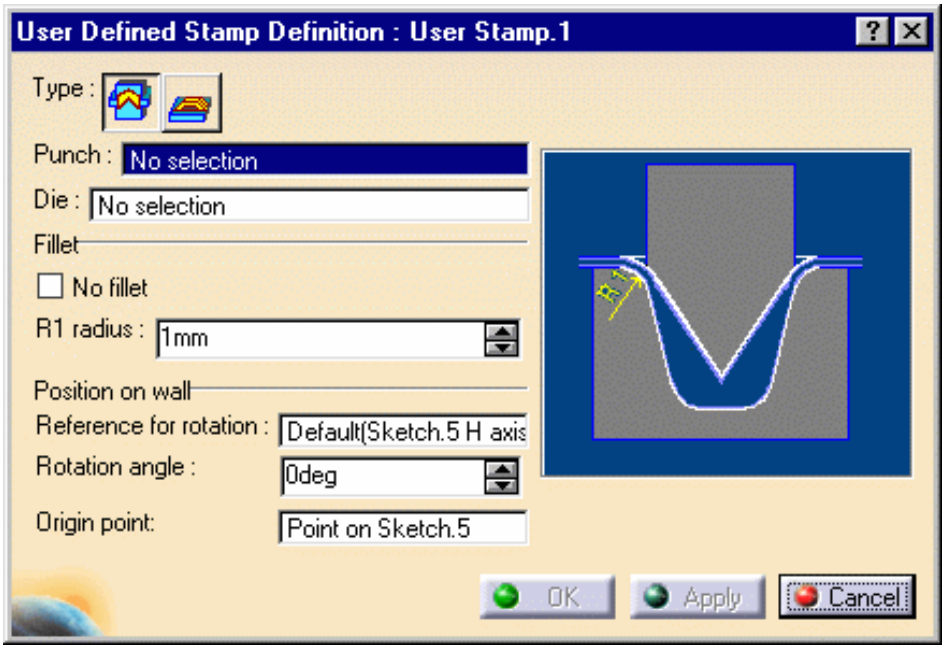
4. Return to the 3D space and create the punch using the Shaft icon .



5. Repeat from step 2 to step 4 to define the die, making sure that it is oriented as described in Defining the Die in Relation to the Wall to be Stamped.

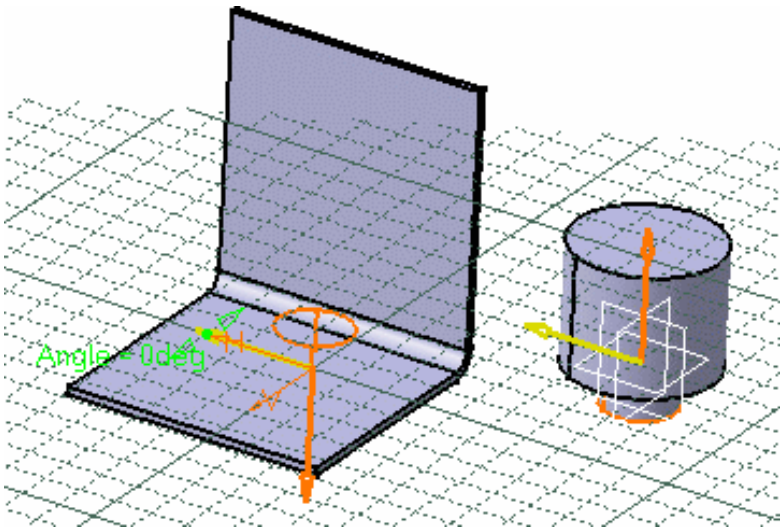
6. Return to the Sheet Metal application, and if needed, use the **Define In Work Object** on the PartBody containing the wall to be stamped.
7. Click the **User Stamping** icon from the Stamping tool bar and select a wall or a face where the stamping is to be created.
   This wall or face is used to define the stamping location and direction, by matching the punch's origin to the selected point on the wall.

   The User Defined Stamp Definition dialog box is displayed:



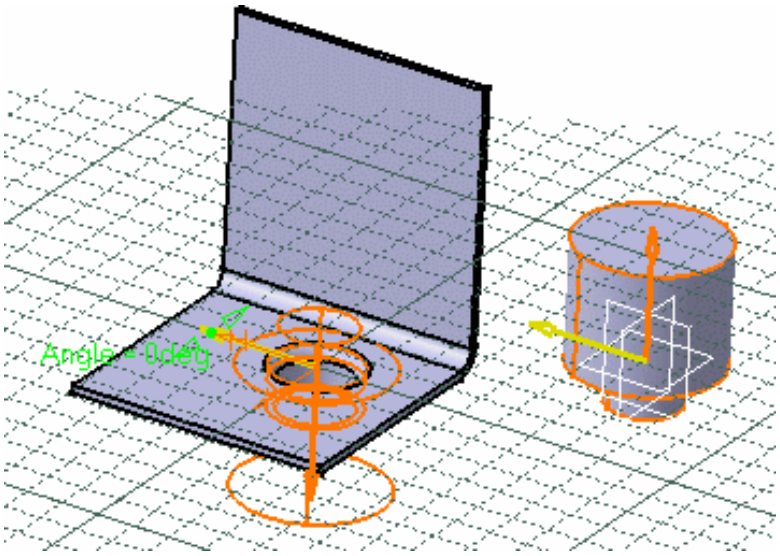
8. Make sure the **With die** icon is pressed down and select the Punch feature from the specification tree.

   The punch's positioning is previewed in the geometry.

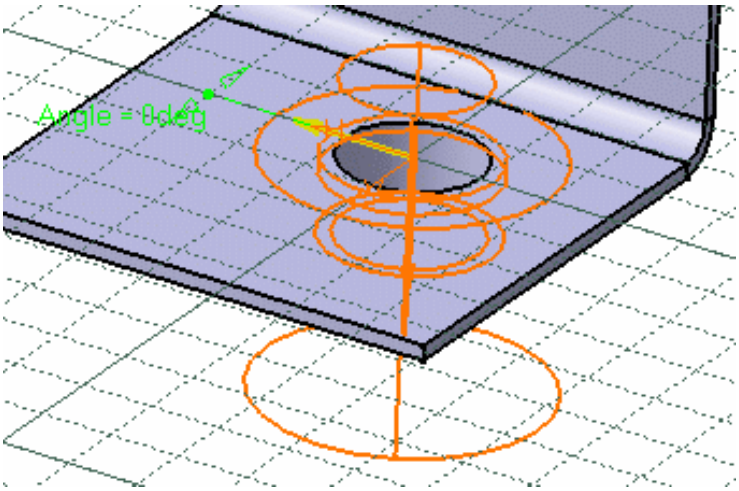


9. Select the Die feature, and click Apply.

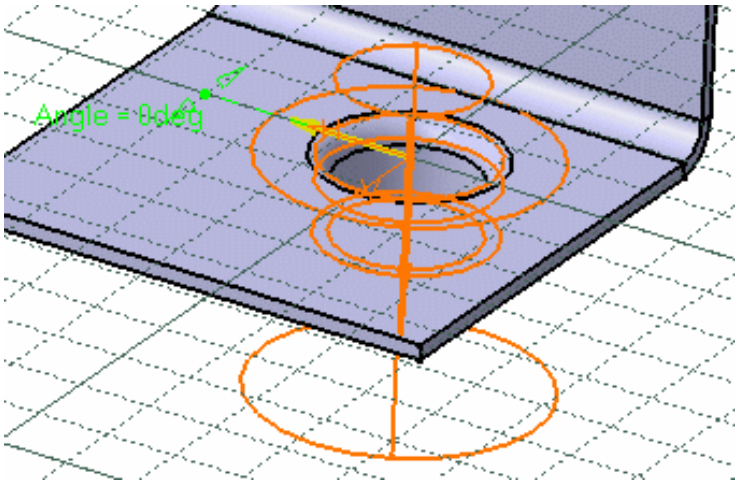
   The die's positioning is previewed in the geometry as well.



10. Check the No Fillet button is you do not wish the stamp to be filleted, or set the radius value if you wish the stamp to be filleted.



Stamp without fillet



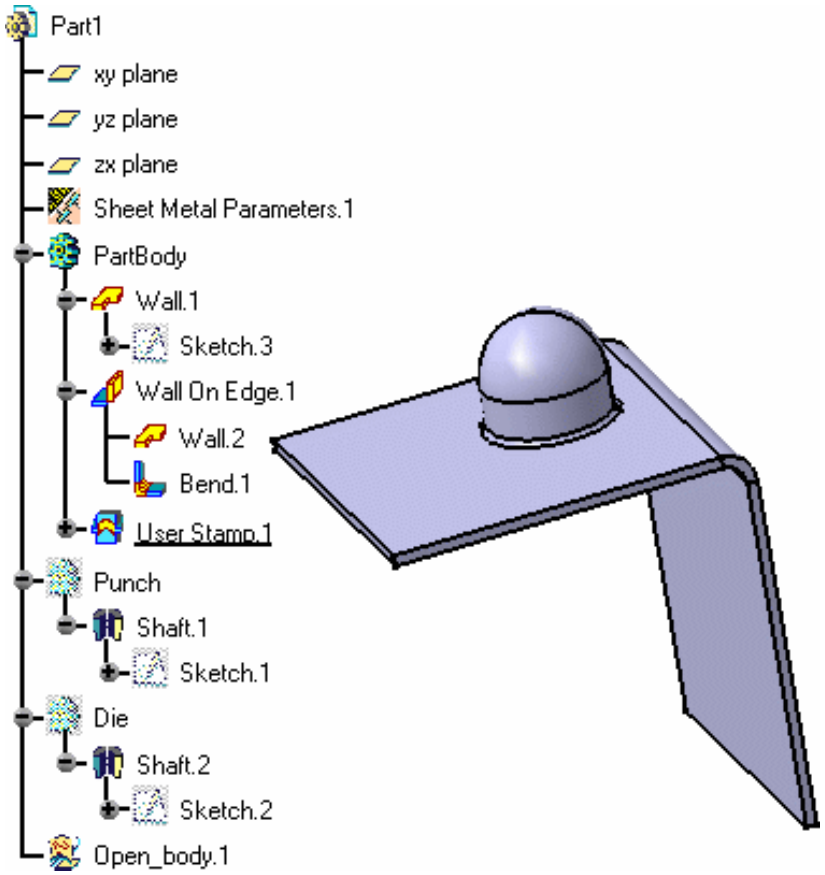
Stamp with fillet

11. If needed, define the stamp's positioning on the selected wall by choosing:

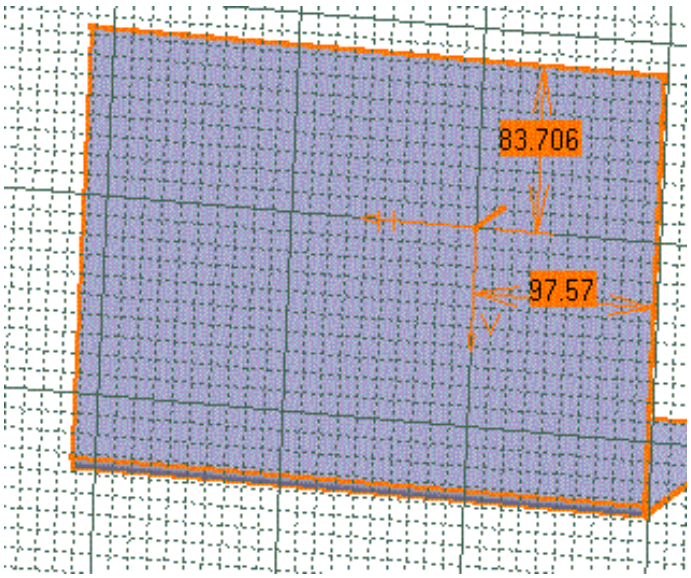
    - a **Reference for rotation**: by default, it is the sketch axis, but you can also select any line or edge on the wall.
    - a **Rotation angle** value: you can either enter a value in the dialog box, or use the manipulator in the geometry to define this value.
    - a new **Origin point** on the wall to coincide with the punch's point of origin.

    This is especially useful for non-circular stamps, but you can very well create the stamp as is, without further positioning.

12. Click OK to validate and create the stamping.
    By default the Punch and Die parts are set in No Show mode when clicking OK to create the stamp on the wall.



- **Radius** is the radius of the bend between the stamping and the wall.
- **Punch** and **Die** are the bodies you have defined previously. If the punch and the die are in another CATPart document, activate this document before clicking the punch or the die.
- If you select two reference lines in addition to the plane, this will create two editable constraints to position the stamping. These constraints are editable.

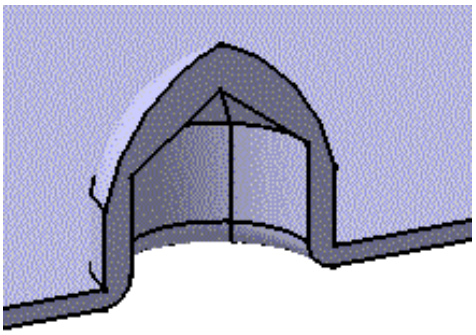


- A user-defined stamping can be edited (punch, die, position, constraints)
- If you enter a punch and a die, the stamping is the difference of the shape of both feature.



- You may create a user-defined stamping from a punch only. The stamping will be the offset of the punch.



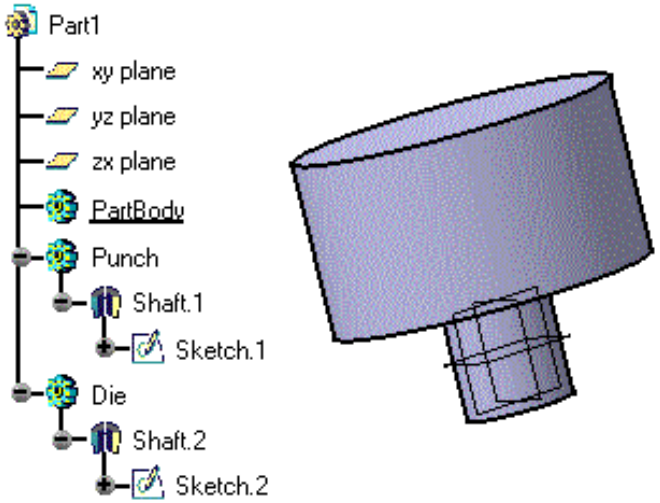


- If you create a stamping on an edge, you will have a section view of the stamping.

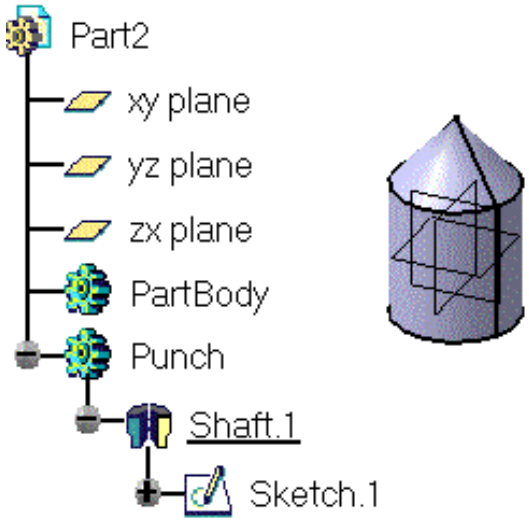
- Only the stamping sketch is displayed in unfolded views.

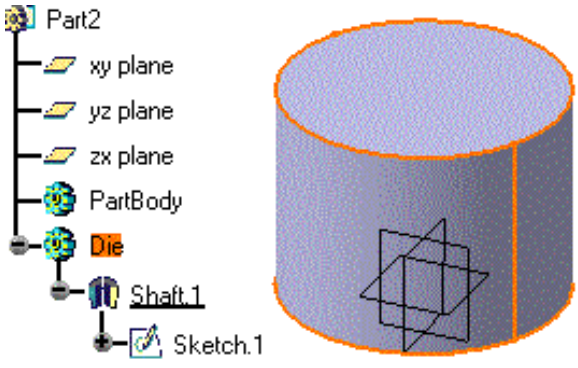
- The punch and die bodies can be defined in the Sheet Metal part where the stamping is to be created (see PunchDie1.CATPart in the samples directory).

  In this case, make sure you select the **Define In Work Object** on the PartBody containing the wall to be stamped, prior to actually creating the stamp.



or as two separate Part Design parts (Punch1.CATPart and Die1.CATPart from the samples directory)





In this case, when selecting the punch or die feature, the system automatically copies this feature into the .CATPart document into which the wall to be stamped is located.
A link is retained between the initial punch or die feature and its copy.



Up Next

Opening and Cutting Faces

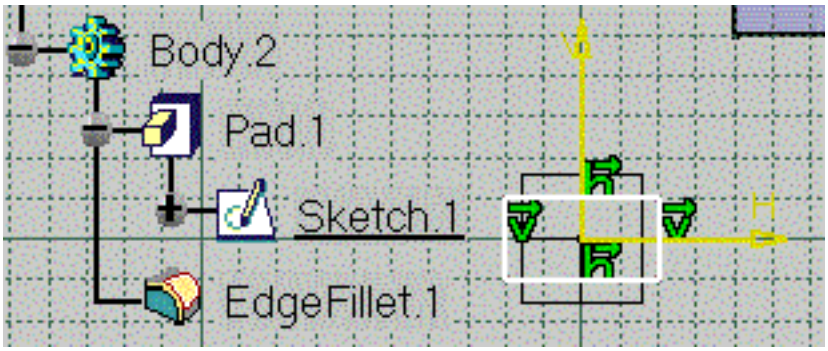
This task explains how to create a stamp from a punch feature with cutting and opening faces.

First, you will define a punch in Part Design, in the absolute axis system.
Then, in a Sheet Metal part, you will bring the punch feature (and its axis system) to a point you have selected. If necessary, you will define a rotation of the axis system from a reference line.

This user-defined stamping can not be combined with the Punch and Die approach.

All CATParts are available from the samples directory (OpenFaces1.CATPart and CuttingFaces1.CATPart or NEWOpenFaces1.CATPart and NEWCuttingFaces1.CATPart for New SheetMetal Design)

1. Start the Part Design application.
2. Insert a PartBody (menu **Insert -> Body**) to define the punch.
3. Enter the sketcher  select the yz plane, and draw the profile of the punch.
4. Return to the 3D space and create the punch using the pad icon  and the fillet icon .



The punch must be oriented as described in Defining the Punch in Relation to the Wall to be Stamped.

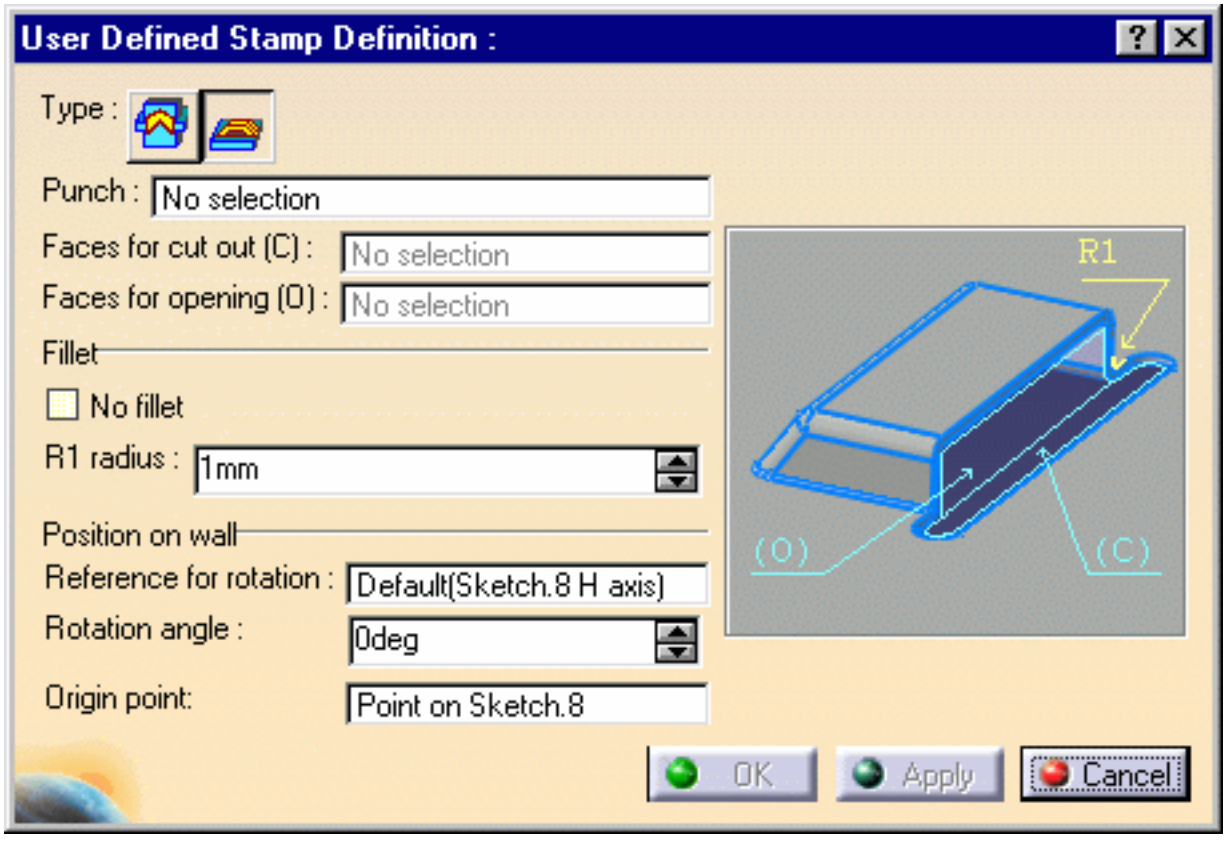
- The punch can be defined in the Sheet Metal part where the stamping is to be created or in another part.
  In this case, when selecting the punch feature, the system automatically copies it into the .CATPart document into which the wall to be stamped is located.
  A link is retained between the initial punch feature and its copy.
- If you define a punch with cutting faces, they should come below the sheet.



5. Return to the Sheet Metal application, and if needed, use the **Define In Work Object** on the PartBody containing the wall to be stamped.
6. Click the **User Stamping** icon  from the Stamping tool bar and select a wall or a face where the stamping is to be created.
   This wall or face is used to define the stamping location and direction, by matching the punch's origin to the selected point on the wall.

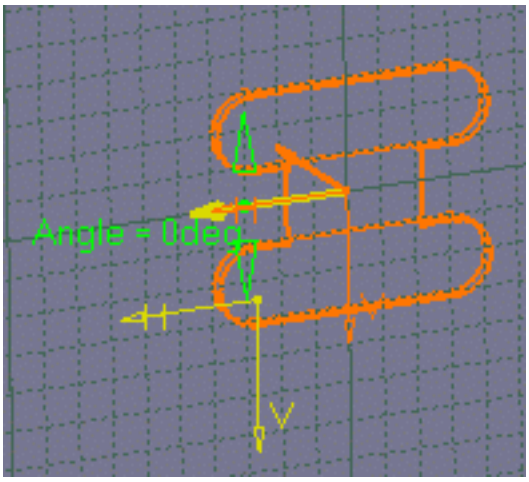
   The User Defined Stamp Definition dialog box is displayed, along with a grid that will help you position the punch.

7. Click the **With cut-out and opening**  icon.



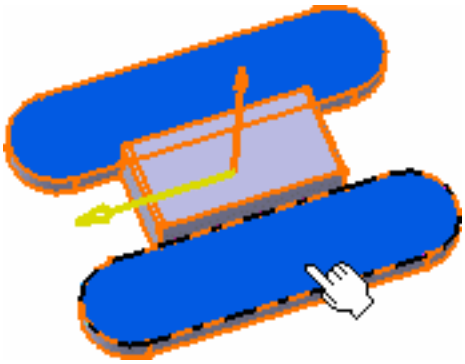
8. Select the punch (Body.2).

   The punch is previewed on the wall.

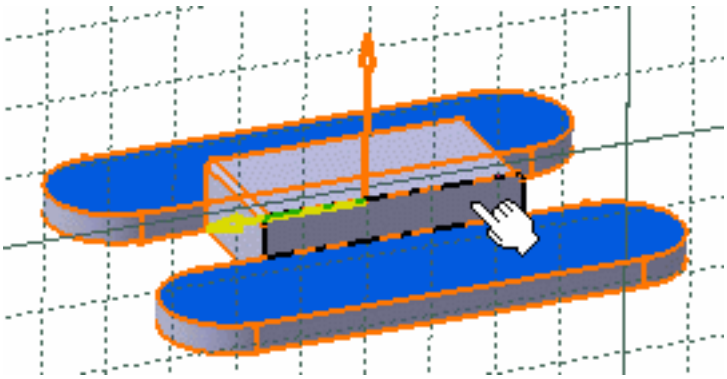


9. Select both top faces of the oblong features of the part (Pad.2 and Pad.3).

   The **Faces for cut-out** field is updated in the dialog box, and now reads: **2 Faces**.

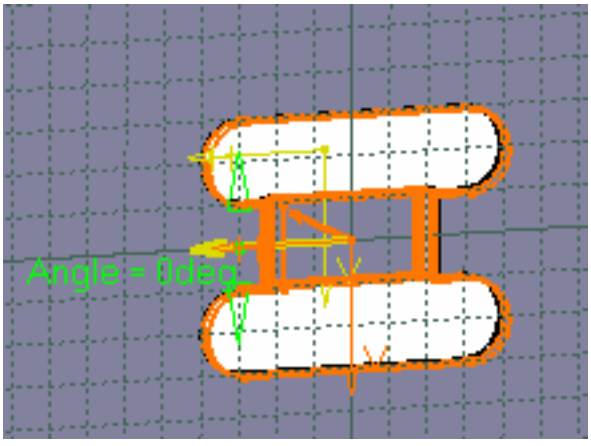


10. Click the **Faces for opening** field and select the lateral faces of the punch (Pad.1).



11. Click Apply.

The stamp is previewed with the opening faces:



12. Check the No Fillet button is you do not wish the stamp to be filleted, or set the radius value if you wish the stamp to be filleted.

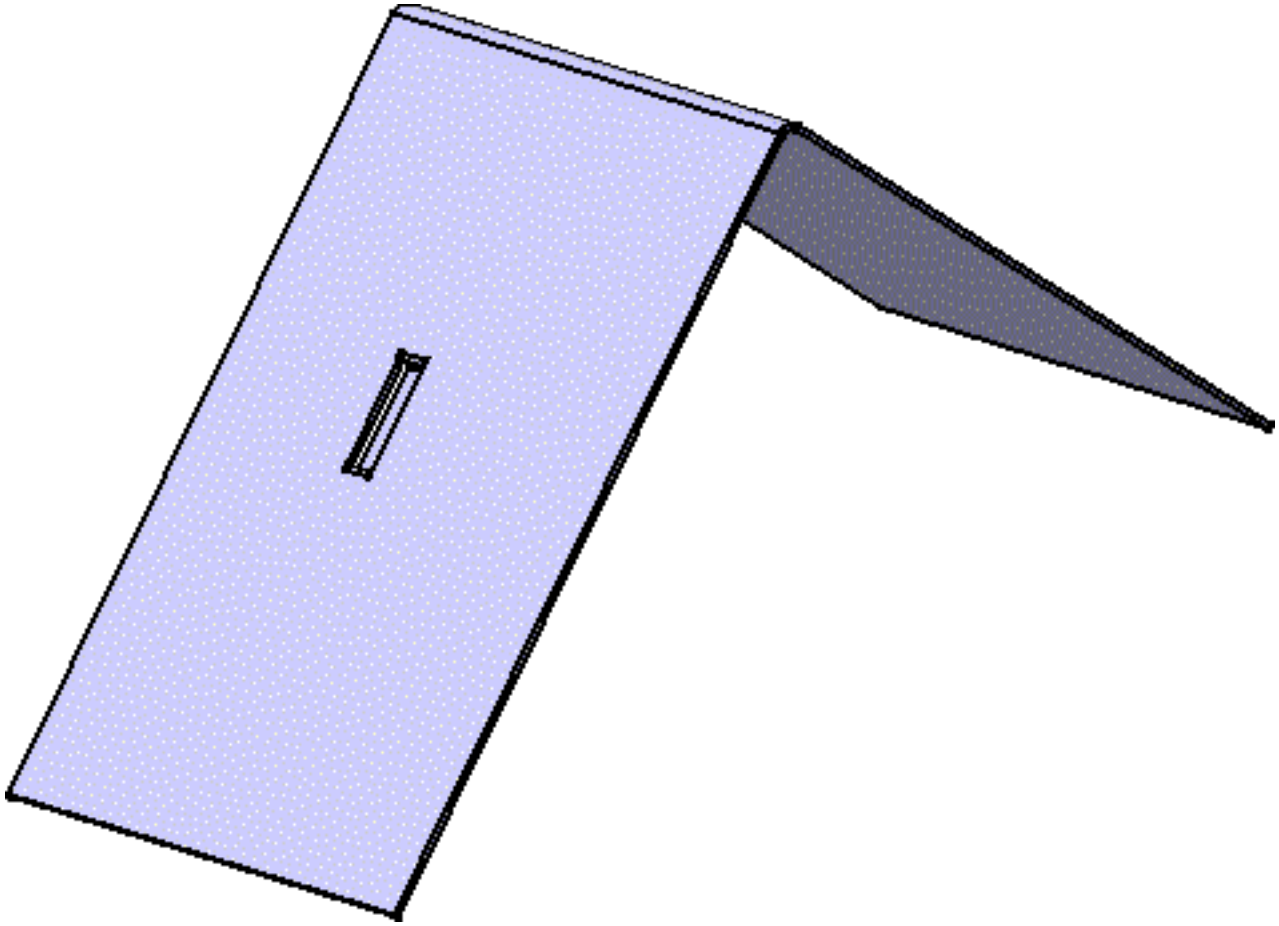
13. If needed, define the stamp's positioning on the selected wall by choosing:

- a **Reference for rotation**: by default, it is the sketch axis, but you can also select any line or edge on the wall.
- a **Rotation angle** value: you can either enter a value in the dialog box, or use the manipulator in the geometry to define this value.
- a new **Origin point** on the wall to coincide with the punch's point of origin.

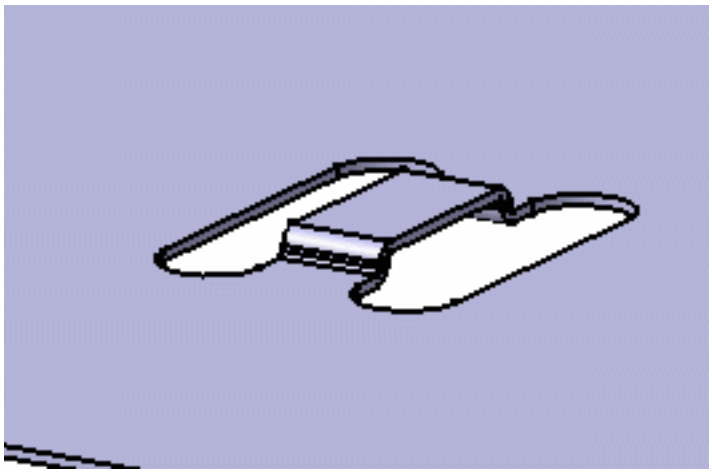
This is especially useful for non-circular stamps, but you can very well create the stamp as is, without further positioning.

14. Click OK to validate and create the stamping.

The stamp is automatically set in No Show mode.



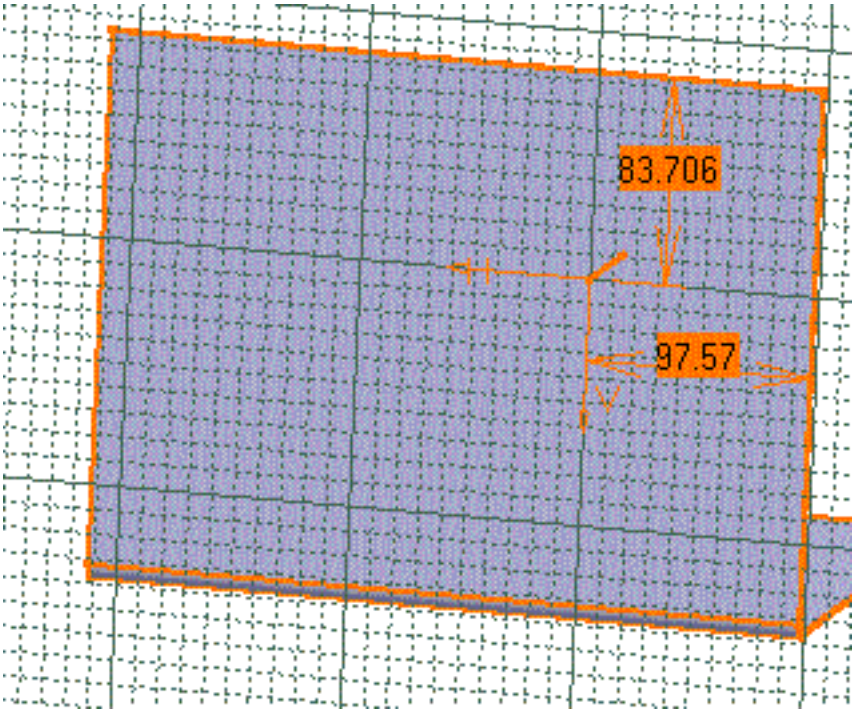
Stamping with opening faces



Stamping with opening and cutting faces

- **Radius** is the radius of the bend between the stamping and the wall.
- **Punch** is the body you have defined previously. If the punch is in another CATPart document, activate this document before clicking the punch.
- The **Faces for cut-out** and **Faces for opening faces** must be picked on the punch, not on the wall.
  If the punch is located into another .CATPart document, these faces must be picked on the copy of the punch where the wall to be stamped is located.

If you select two reference lines in addition to the plane, this will create two editable constraints to position the stamping. These constraints are editable.



- A user-defined stamping can be edited (punch, die, position, constraints)
- Check the **No radius** option to deactivate the Radius R1 value, and to create the stamp without a fillet.



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Editing User-Defined Stamps

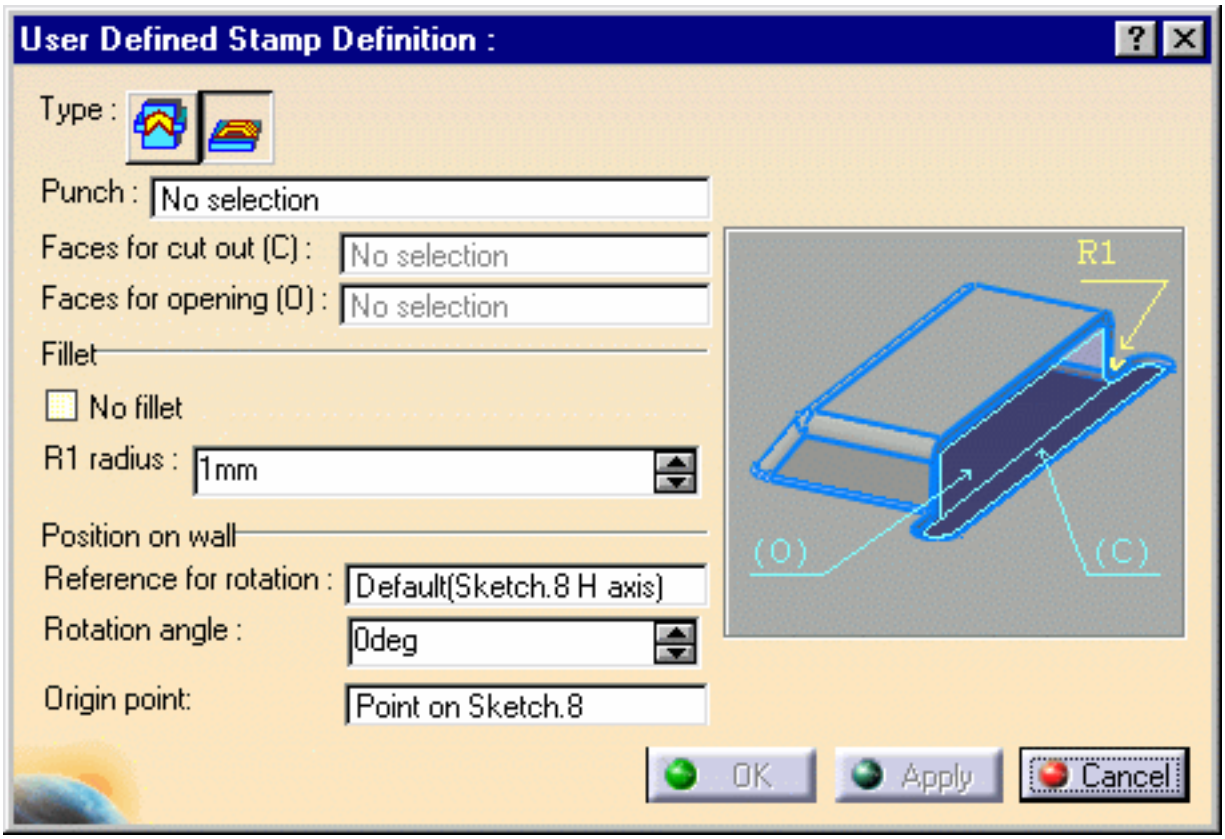
This task explains how to edit a user-defined stamp, that is:

- to change its type
- add or remove cutting and opening faces

Any .CATParts containing a user-defined stamp.

1. Double-click the existing user-defined stamp from the specification tree.

   The User Defined Stamp Definition dialog box is displayed.



2. Change stamp type using the icons:

- If you change from **With die** to **With cut-out and opening**, the Die feature no longer is selected, and you need to select **Faces for cut-out** and/or **Faces for opening**.
- If you change from **With cut-out and opening** to **With die**, the punch faces no longer are selected and you may select a die feature if you wish (it is not compulsory).

Basically, only the punch remains selected.

If you are working with a cutting punch (**With cut-out and opening** option) you may want to add or remove some cutting or opening faces:

3. Click in the **Faces for cut-out** field then:

- select a face in the geometry to add it to the already selected cutting faces
- select an already selected face to remove it from the cutting faces
- use the **Clear selection** contextual menu to remove all cutting faces that have been previously selected.

4. Similarly, click in the **Faces for opening** field then:

- select a face in the geometry to add it to the already selected opening faces
- select an already selected face to remove it from the opening faces
- use the **Clear selection** contextual menu to remove all opening faces that have been previously selected.

5. Modify any other parameter as needed.

6. Click OK in the User Defined Stamp Definition dialog box to take these modifications into account.

The stamp is updated accordingly.



Back Up

Patterning

This section explains and illustrates how to create various kinds of patterns on Sheet Metal parts.


Create rectangular patterns: select the element to be duplicated, set the patterning type, and its parameters, and the reference direction


Create circular patterns: select the element to be duplicated, set the axial reference parameters, the reference direction, and possibly the crown definition


Create user-defined patterns: select the element to be duplicated, and the positioning sketch and anchor point


To know more about patterns, refer to *Part Design User's Guide*.

Back Up Next

Creating Rectangular Patterns

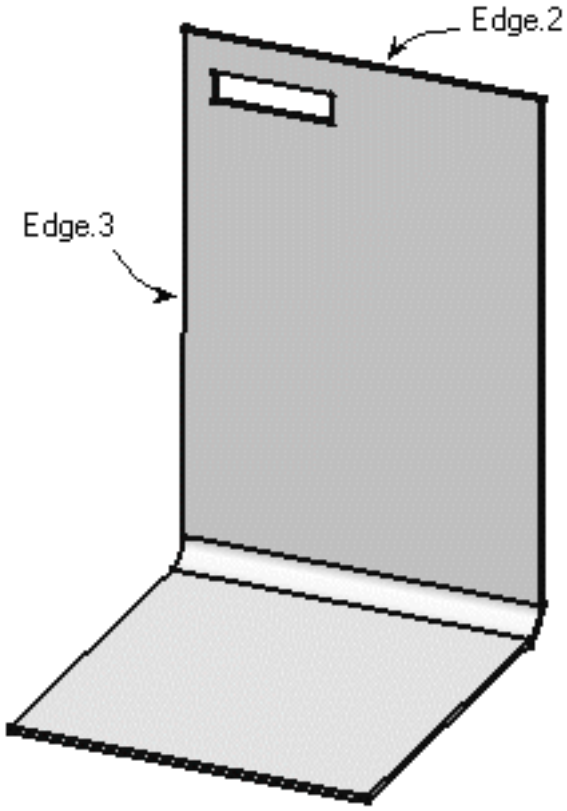
In this task, you are going to create rectangular cutouts according to a pattern. These features make the creation process easier.

In the NEW SheetMetal Design, you can only duplicate cutouts, holes, and stamps (except stiffening rib),

even stamps without fillet (radius=0)

These features must lie on a unique and planar wall.

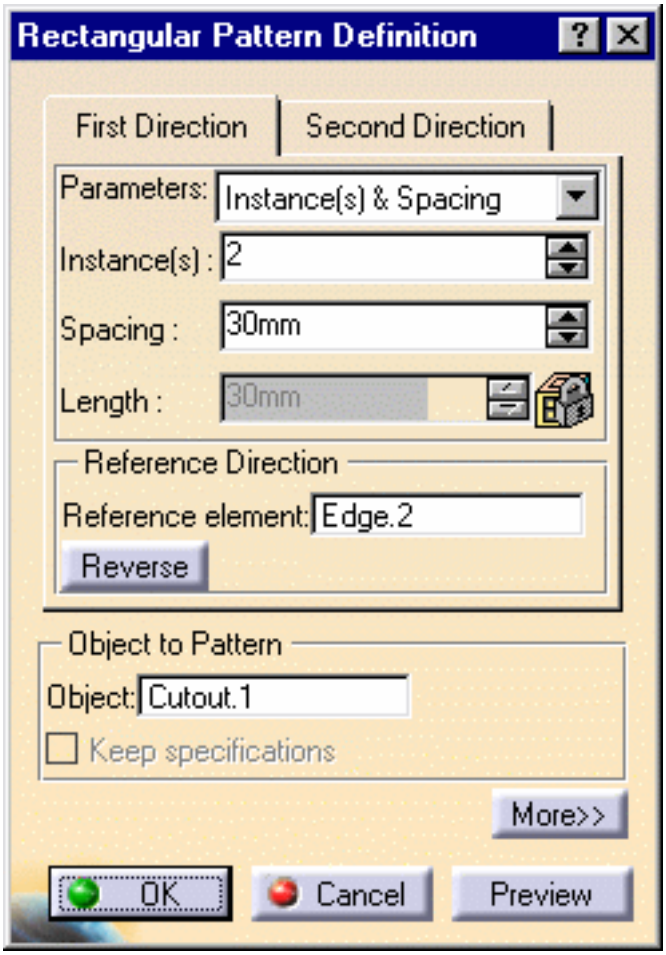
Open the RectangularPattern1.CATPart document from the Samples/sheet metal directory.
If you use the NEW Sheet Metal Design, open the NEWRectangularPattern1.CATPart document.
The Sheet Metal part looks like this:



1. Select the rectangular cutout you want to duplicate.

2. Click the **Rectangular Pattern** icon .

The **Rectangular Pattern Definition** dialog box is displayed. Each tab is dedicated to a direction to define the location of the duplicated feature.



3. Set the specification for the First Direction by selecting the first edge (**Edge.2**) as shown, to specify the first direction of creation.
An arrow is displayed on the wall.

   The **Reverse** button enables to modify the direction.
You can also click the arrow in the 3D geometry.

4. Keep the **Instances & Spacing** options to define the parameters.
Choosing these parameters types dims the **Length** field because the application no longer needs this specification to space the instances.

You can set the duplication parameters by choosing the number of instances, the spacing between instances, or the total length of the zone filled with instances. Three options are available:

1. **Instances & Length**: the spacing between instances is automatically computed based on the number of instances and the specified total length
2. **Instances & Spacing**: the total length is automatically computed based on the number of instances and the specified spacing value
3. **Spacing & Length**: the number of instances is automatically computed to fit the other two parameters.

For each of these cases only two fields are active, allowing you to define the correct value.

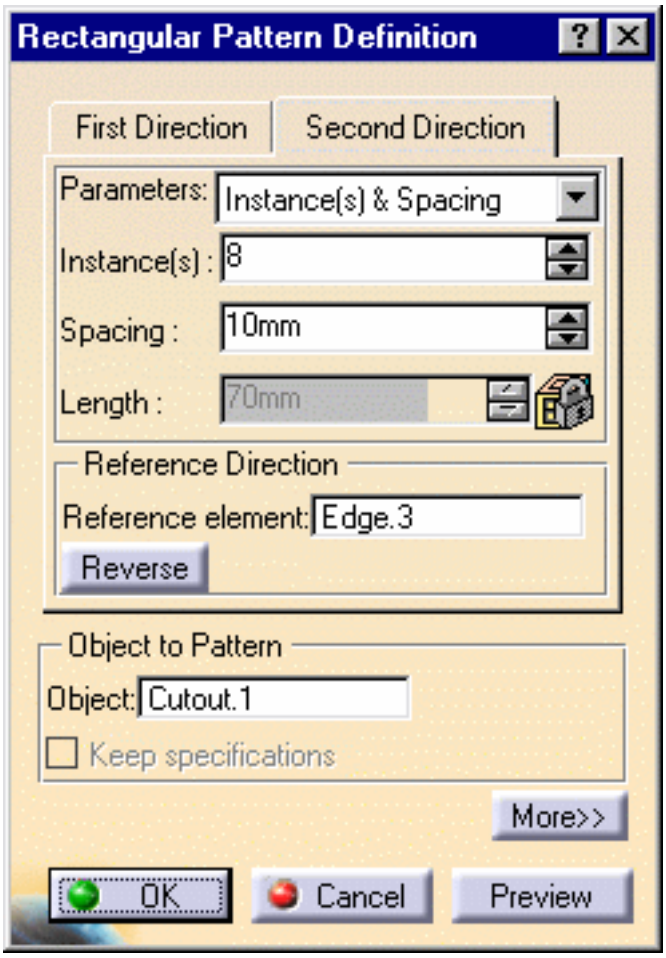
If you set **Instances & Length** or **Spacing & Length** parameters, note that you cannot define the length by using formulas.

Patterns should not go beyond the model.

5. Enter 2 as the number of instances you wish to obtain in the first direction.
6. Define the spacing along the grid: enter 30mm.

Defining the spacing along the grid and the length of your choice, would make the application compute the number of possible instances and space them at equal distances.

7. Now, click the **Second Direction** tab to define the other parameters.



Note that defining a second direction is not compulsory. Creating a rectangular pattern defining only one direction is possible.

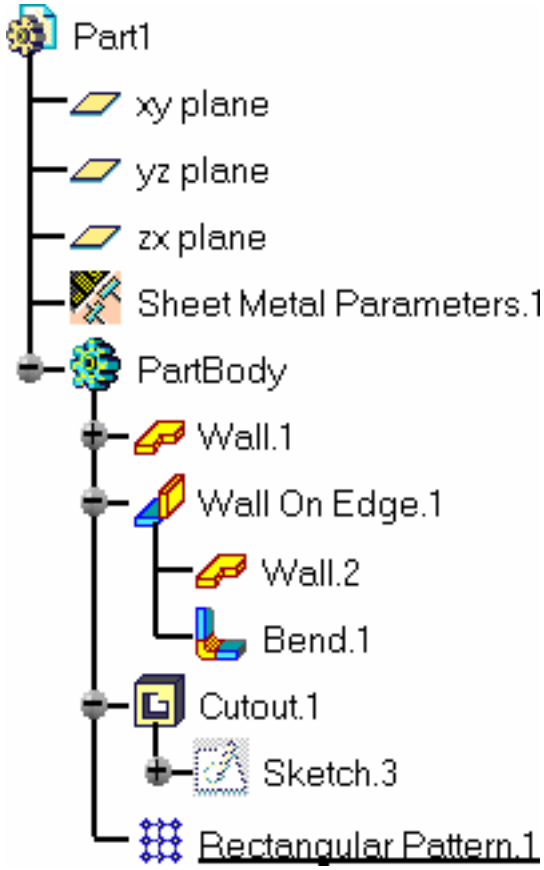
8. Select the second edge (**Edge.3**), as shown, to define the second direction.

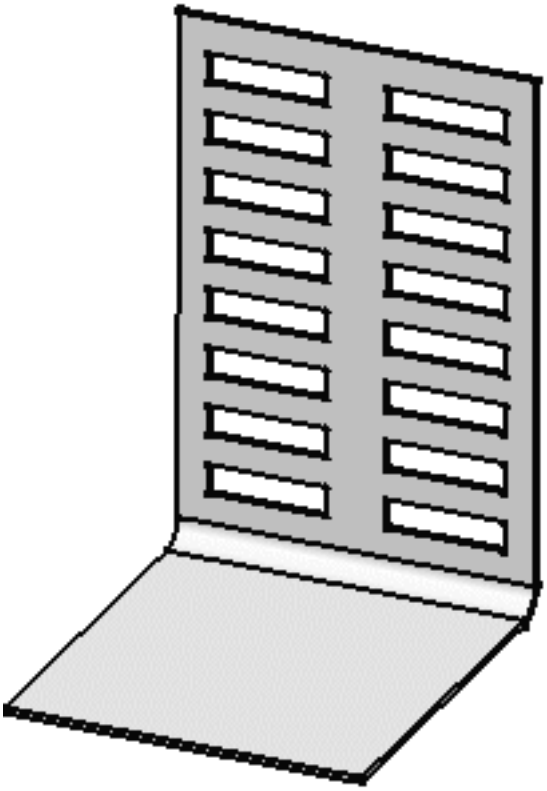
9. Keep the **Instances & Spacing** option: enter 8 and 10 mm in the appropriate fields.

   Additional cutouts have been aligned along this second direction.

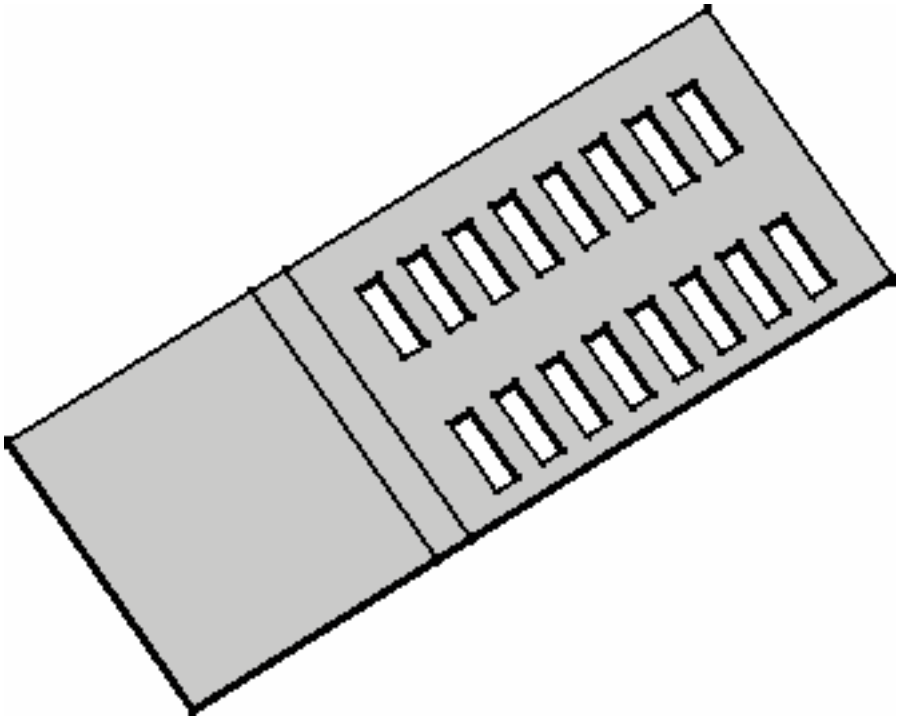
10. Click **OK** to repeat the cutouts.

After the update, the Sheet Metal part looks like this:



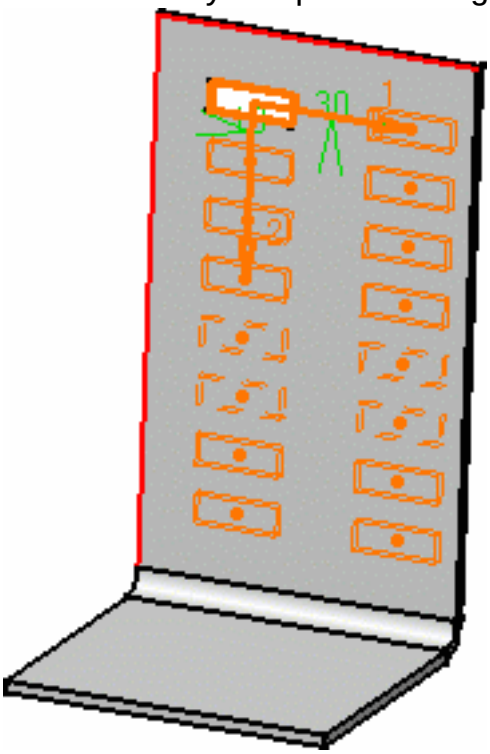


11. Select this icon  to unfold the part:
The pattern is updated on the unfolded view.



12. Click the More>> button to display further options.

The **Simplified representation** option lets you lighten the pattern geometry, when more than 10 instances are generated in one direction. What you need to do is just check the option, and click Preview. The system automatically simplifies the geometry:



You can also specify the instances you do not want to see by double-clicking the dots. These instances are then represented in dashed lines during the pattern definition and then are no longer visible after validating the pattern creation. The specifications remain unchanged, whatever the number of instances you view. This option is particularly useful for patterns including a large number of instances.

Up Next

Creating Circular Patterns

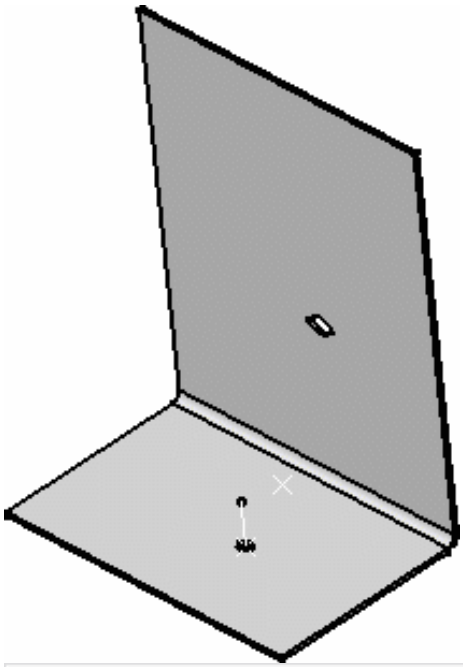
In this task, you are going to create circular cutouts according to a pattern.
These features make the creation process easier.

In the NEW SheetMetal Design, you can only duplicate cutouts, holes, and stamps (except stiffening rib),

even stamps without fillet (radius=0)

These features must lie on a unique and planar wall.

Open the CircularPattern1.CATPart document from the Samples/sheet metal directory.
If you use the NEW Sheet Metal Design, open the NEWCircularPattern1.CATPart document.
The Sheet Metal part looks like this:

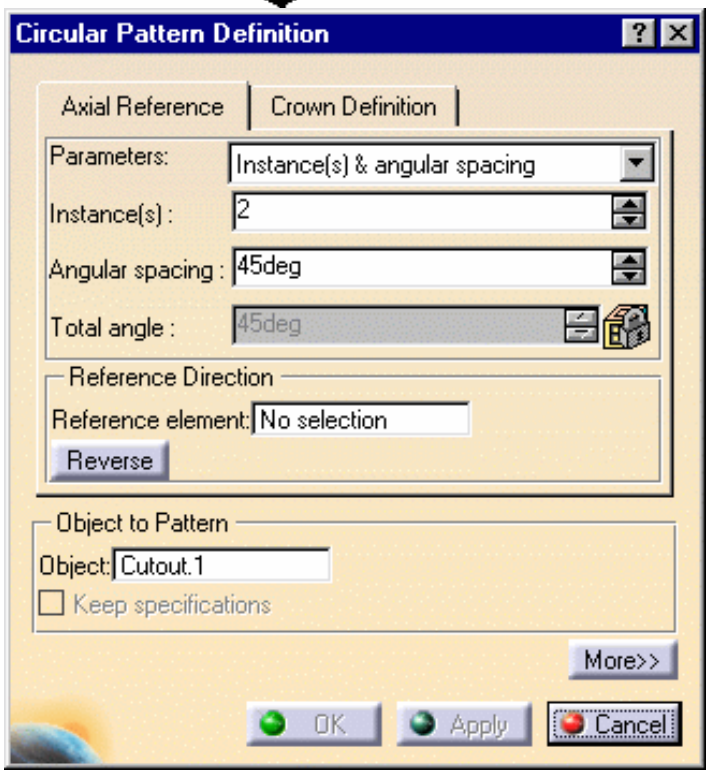


1. Select the circular cutout you want to duplicate.

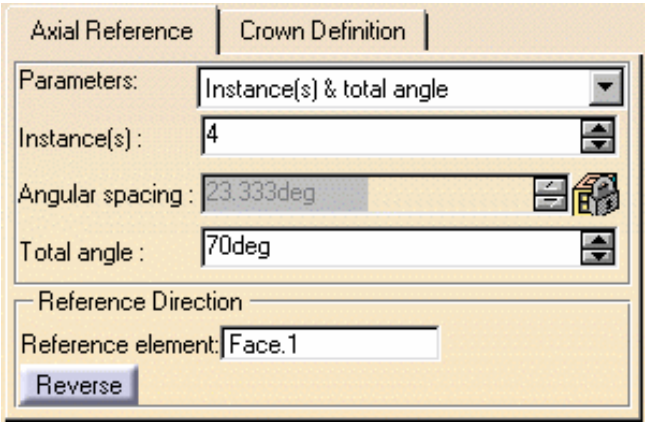
2. Click the **Circular Pattern** icon .

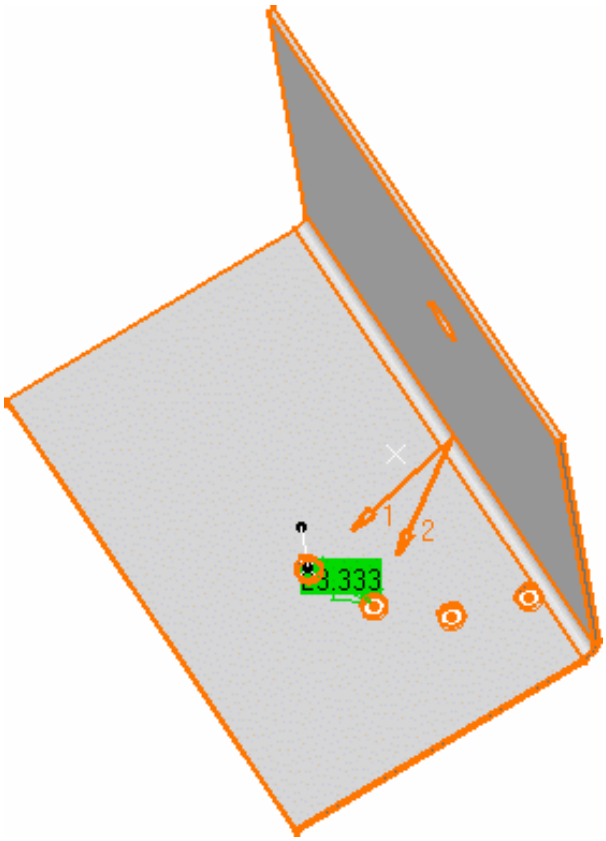
   The **Circular Pattern Definition** dialog box is displayed.

3. Define the **Axial Reference** by choosing the **Parameters** type, and reference direction.

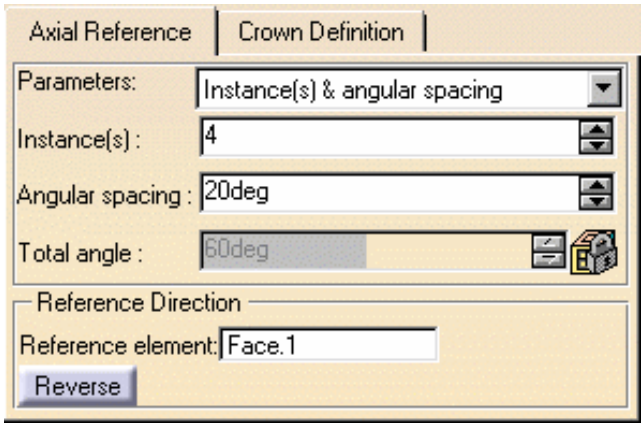


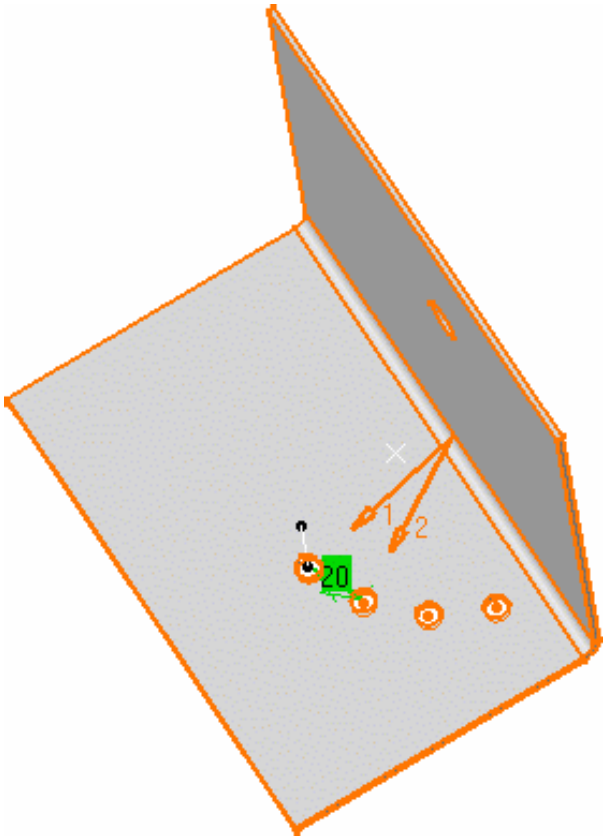
- **Instance(s) & total angle**: the number of patterns as specified in the instances field are created, in the specified direction, and evenly spread out over the total angle.



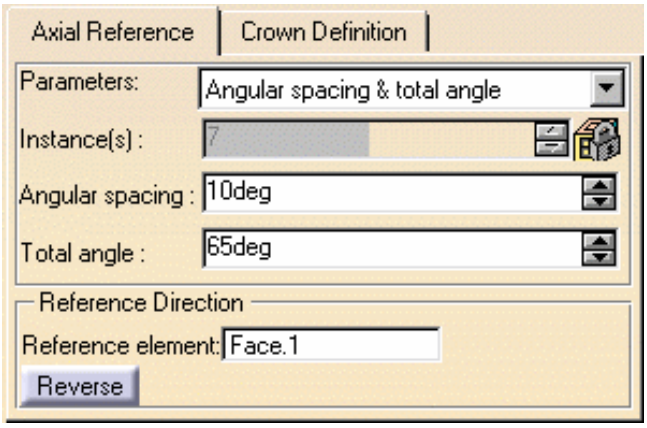


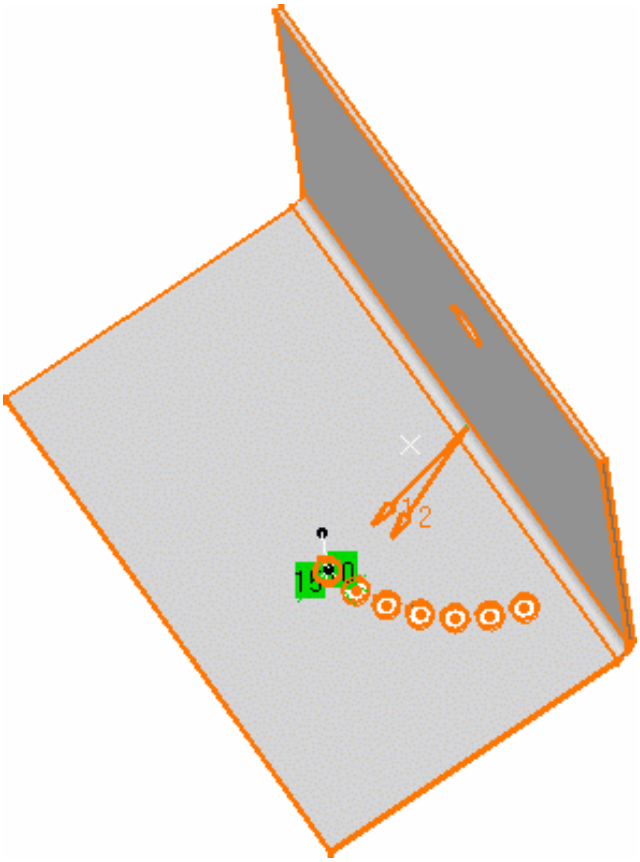
- **Instance(s) & angular spacing**: the number of patterns as specified in the instances field are created in the specified direction, each separated from the previous/next one of the angular angle value.



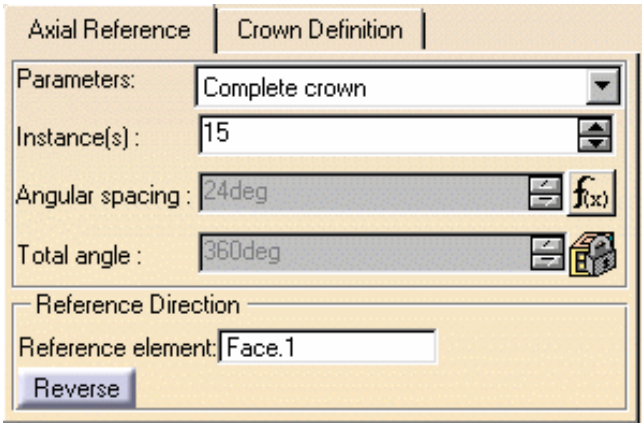


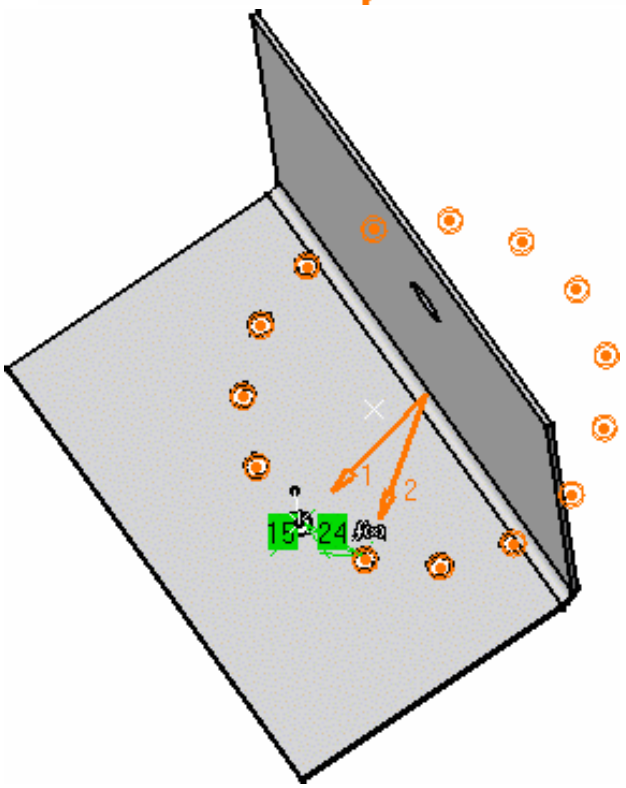
- **Angular spacing & total angle**: as many patterns as possible are created over the total angle, each separated from the previous/next one of the angular angle value.





- **Complete crown**: the number of patterns as specified in the instances field are created over the complete circle (360°).





If you set **Instance(s) & total angle** or **Angular spacing & total angle** parameters, note that you cannot define the length by using formulas.

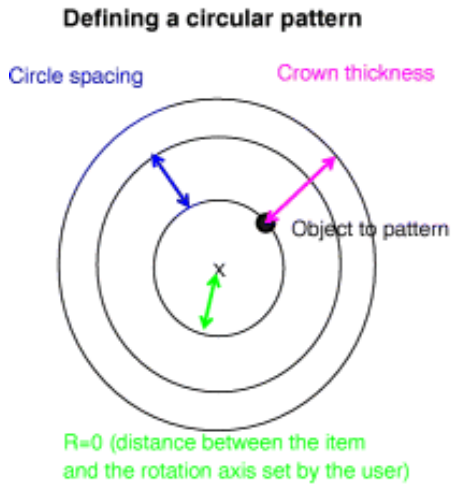
4. Click the Reference element and select the element defining the rotation axis.
   Here select the face on which lies the circular cutout.

- To define a direction, you can select an edge or a planar face.
  Should you select the face of a wall, the rotation axis would be normal to that face.
- Click the **Reverse** button to inverse the rotation direction.

Now you are going to add a crown to this pattern.

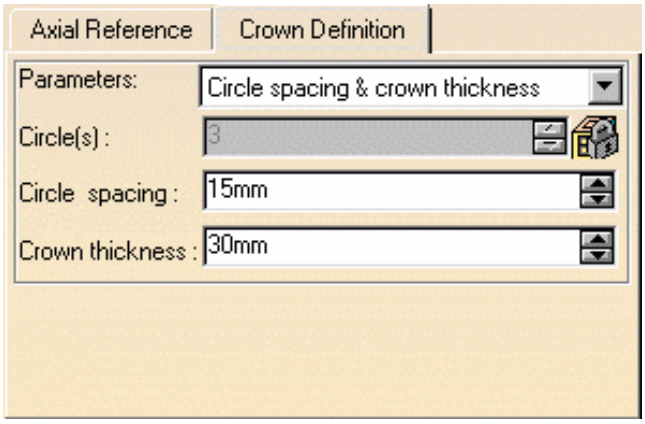
5. Click the **Crown Definition** tab, and choose which parameters you wish to define the crown.

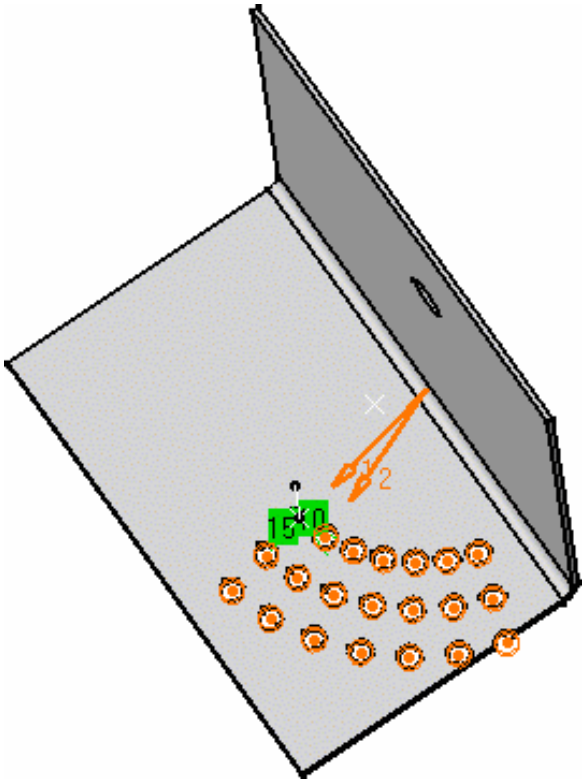
This figure may help you define these parameters:



- Circle(s) and crown thickness: you define the number of circles and they are spaced out evenly over the specified crown thickness
- Circle(s) and circle spacing: you define the number of circles and the distance between each circle, the crown thickness being computed automatically
- Circle(s) spacing and crown thickness: you define the distance between each circle and the crown thickness, and the number of circles is automatically computed.

For example, using the values described above for the **Angular spacing & total angle** option, you could define the crown as:

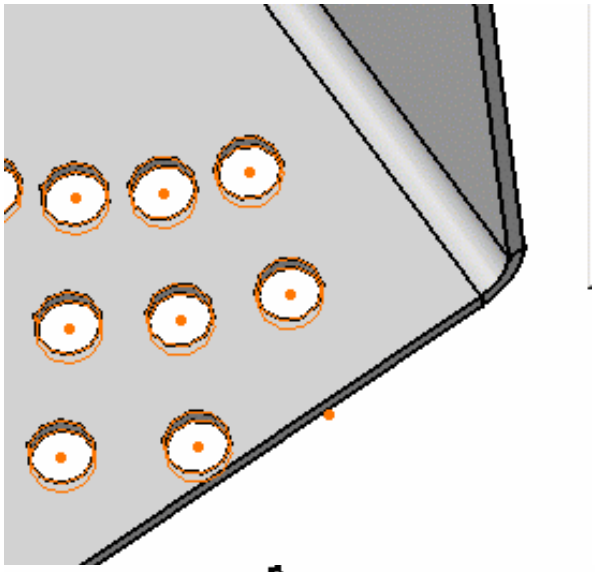




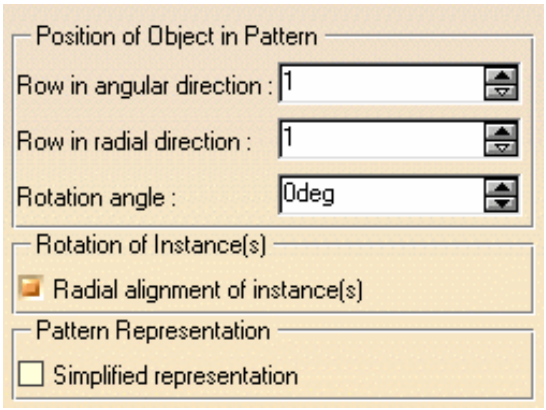
Note that one of the pattern is created beyond the wall.

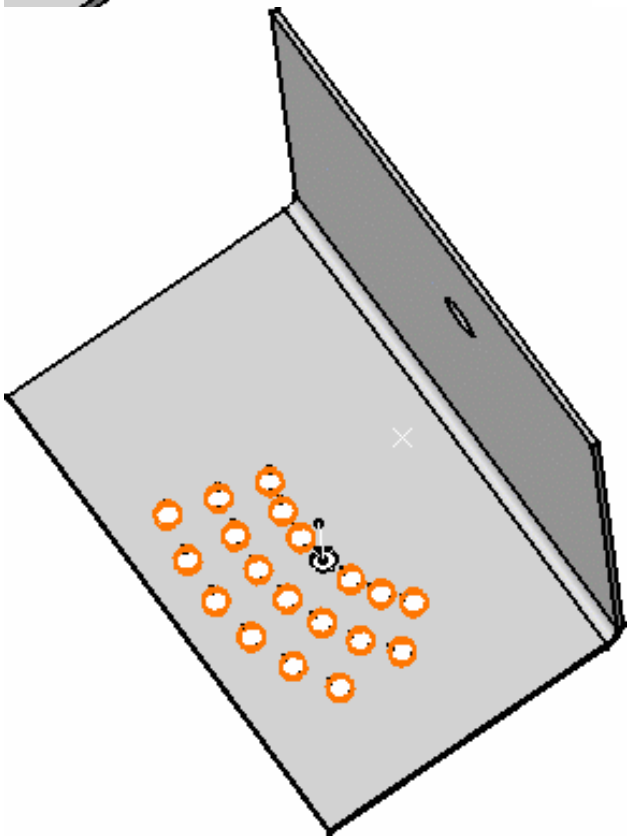
You can delete instances of your choice when creating or editing a pattern. To do so, just select the points materializing instances in the pattern preview.

The instance is deleted, but the point remains, as you may wish to click it again to add the instance to the pattern definition again.



6. Click the **More>>** button to display further options:





Using these options, you can change the position of the selected cutout within the crown. For example, if you set the **Row in angular direction** parameter to 4, this is what you obtain: the initially selected cutout is the fourth instance, based on the rotation direction, of the pattern.

Typically, in this case, you might want to edit the pattern and click again the instance that you removed above, to get a full pattern.

- The **Simplified representation** option lets you lighten the pattern geometry, when more than 10 instances are generated in one direction. What you need to do is just check the option, and click Preview. The system automatically simplifies the geometry:
  You can also specify the instances you do not want to see by double-clicking them . These instances are then represented in dashed lines during the pattern definition and then are no longer visible after validating the pattern creation. The specifications remain unchanged, whatever the number of instances you view. This option is particularly useful for patterns including a large number of instances.
- When checking the **Radial alignment of instances**, all instances have the same orientation as the original feature. When unchecked, all instances are normal to the lines tangent to the circle.

7. Click OK to create the pattern.

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Creating User-Defined Patterns

The User Pattern command lets you duplicate a feature, such as a sketch, a cutout, a stamp, or any other feature as many times as you wish at the locations of your choice. Locating instances consists in specifying anchor points. These points are sketches.

In the NEW SheetMetal Design, you can only duplicate cutouts, holes, and stamps (except stiffening rib),

even stamps without fillet (radius=0)

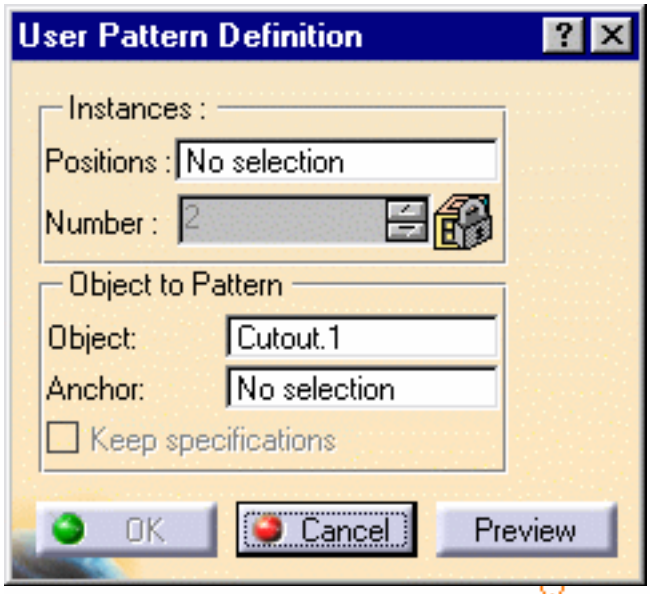
These features must lie on a unique and planar wall.

Open the UserPatterns1.CATPart document from the samples directory.
If you use the NEW Sheet Metal Design, open the NEWUserPatterns1.CATPart document.

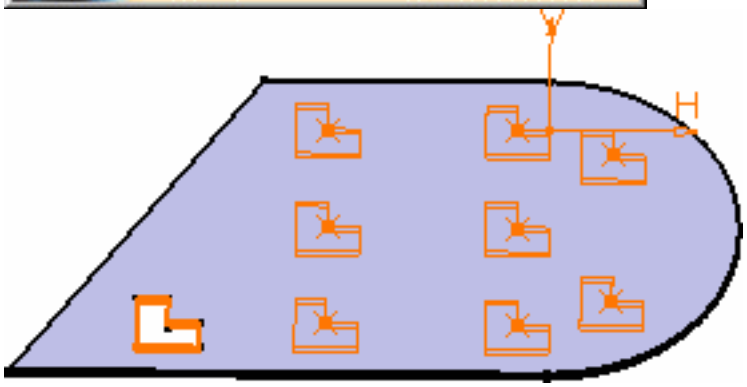
1. Select the feature to be duplicated.
   Here we selected the cutout.

2. Click the **User Pattern** icon .

   The User Pattern Definition dialog box is displayed.

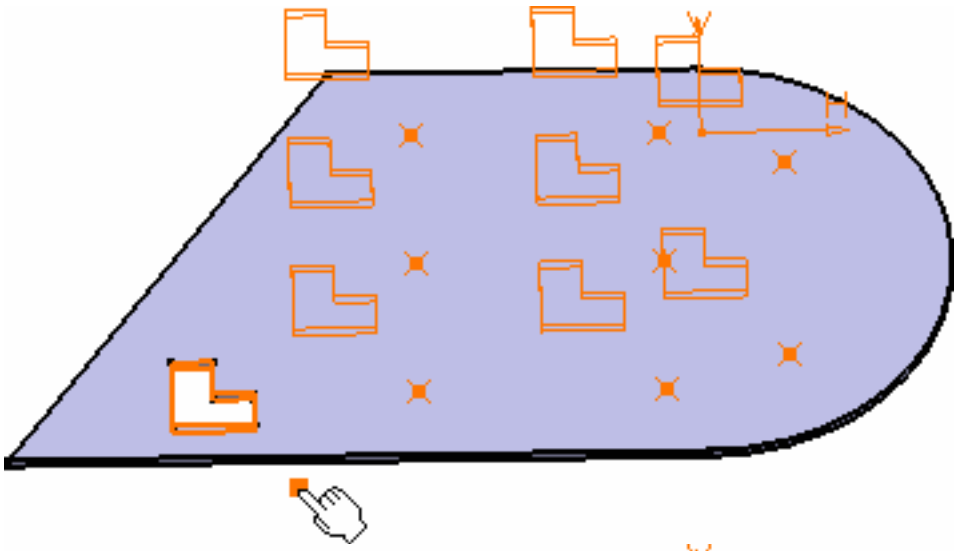


3. Select '**Sketch 9**' in the specification tree and click **Preview**.
   The sketch contains the points you need to locate the duplicated cutouts.

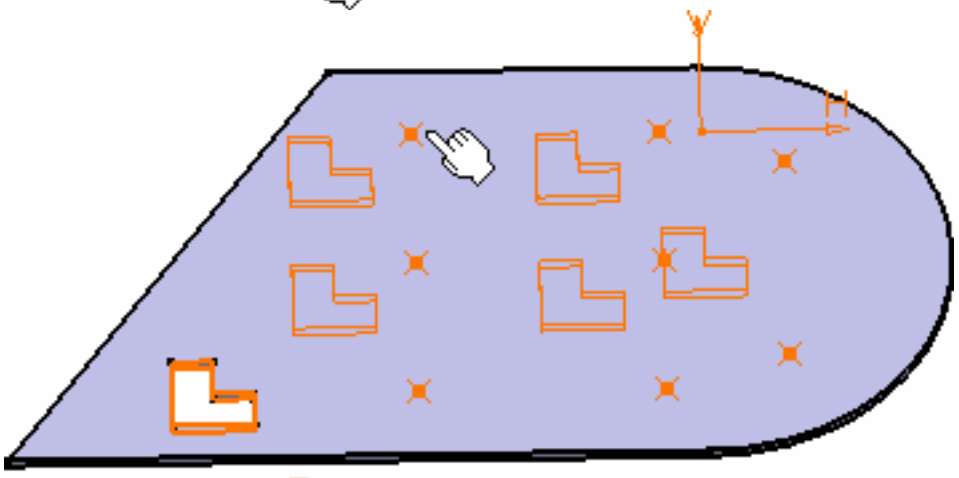


By default, the application positions each instance with respect to the center of gravity of the element to be duplicated. To change this position, use the anchor field: click the anchor field and select a vertex or a point.

4. Click inside the Anchor field and select the point (Point.1) to indicate a new reference location.

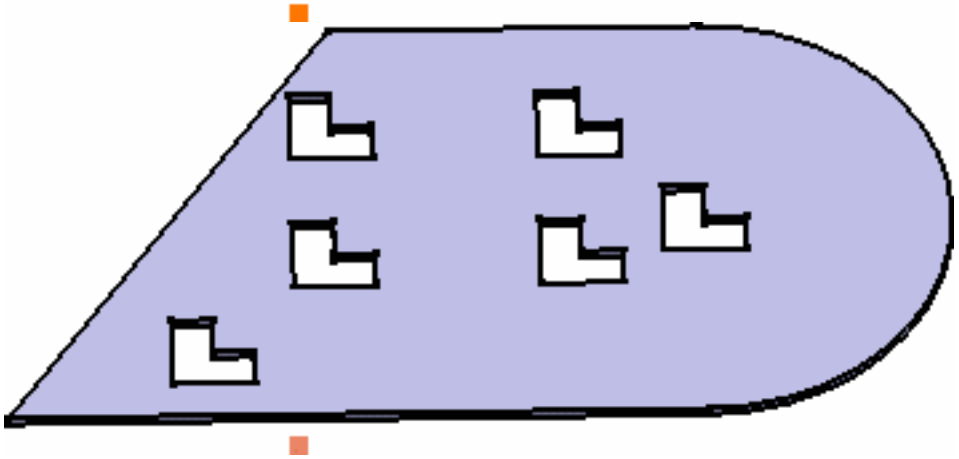


5. You can then click the points corresponding to the pattern instances to be removed.



6. Click OK in the User Pattern Definition dialog box.

   Cutouts are created at the points of the sketch.



Would you need to unfold the part using the  icon, you would notice that the pattern is updated.

Please be careful concerning the content of the sketch selected to locate the instances of the User Pattern. The sketch should only include the points locating the instances of the selected reference feature. Therefore it is useful to create a point as "Construction Element" corresponding to the reference feature (in case of constraints for example), so that the instances are different from this feature.

Back Up

Corner Relief

This section explains and illustrates different methods to create corner relieves on bends.

Redefine an automatic corner relief: double-click an automatic corner relief, edit its parameters in the dialog box


Create a local corner relief: select two or more bends, the corner relief type and parameters

See also Bend Corner Relief parameters settings.

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Redefining an Automatic Corner Relief



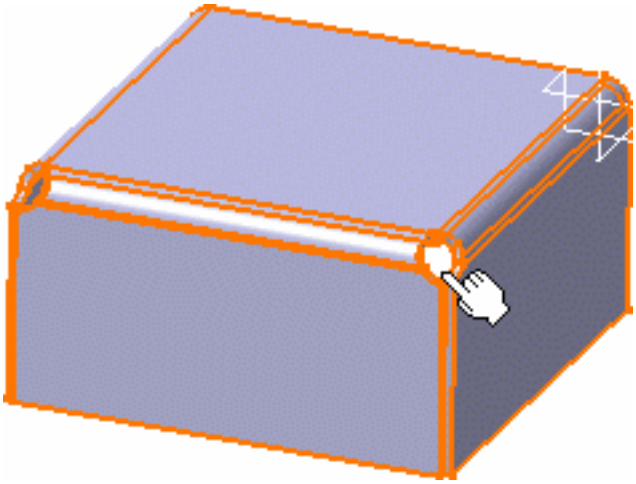
This task explains how to redefine automatic corner relieves on a Sheet metal part.

Open the CornerRelief01.CAPTPart model from the samples directory.
If you use the New SheetMetal Design workbench, open the NEWCornerRelief01.CATPart document.

1. Double-click the bend on which the corner relief you want to redefine is located.

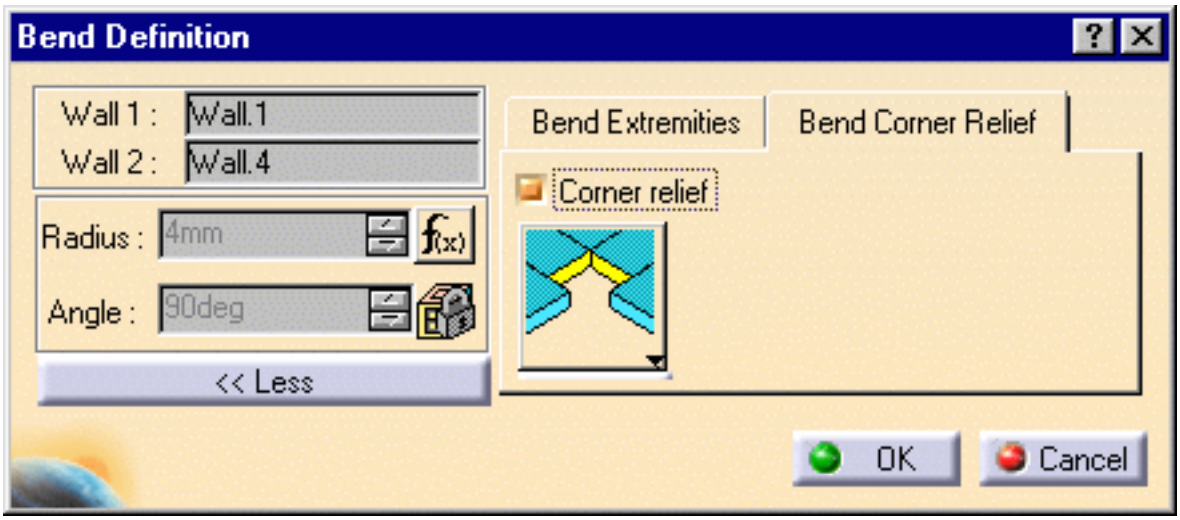
   It may be easier to double-click it from the specification tree.

   The Bend Definition dialog box is displayed.



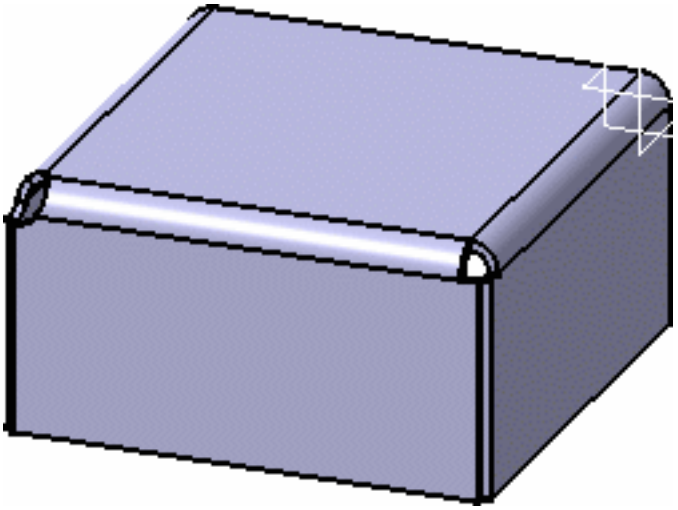
2. Click the More button and select the Bend Corner Relief tab.

   This tab is similar to the Corner Relief tab of Sheet Metal Parameters dialog box.



3. Choose a new corner relief type, and click **OK** to validate.

   This definition will apply to the current corner relief, and will prevail over any other corner relief definition you may enter through the Sheet Metal Parameter dialog box.



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Creating A Local Corner Relief

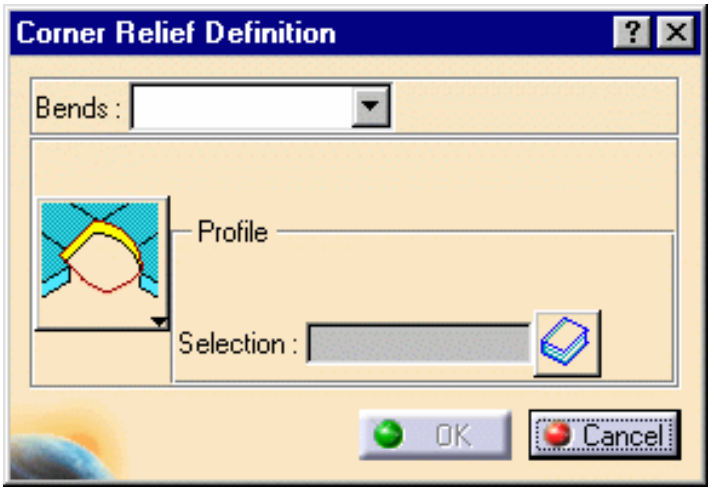
This task explains how to define a corner relief locally on a set of bends.
Depending on the number of bends involved, not all types of corner relief are available.

Open the CornerRelief02.CATPart document from the samples directory.

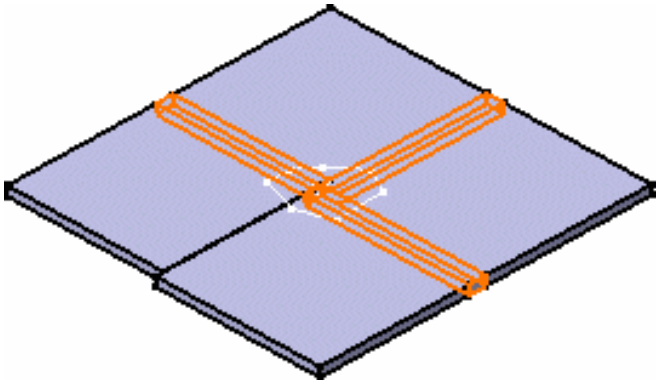
The part needs to be unfolded prior to creating the corner relief.

1. Click the **Corner Relief** icon .

   The Corner Relief Definition dialog box is displayed.

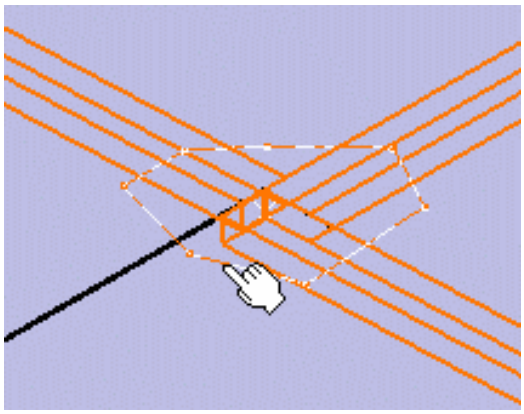


2. Select the supports on which a corner relief should be created.



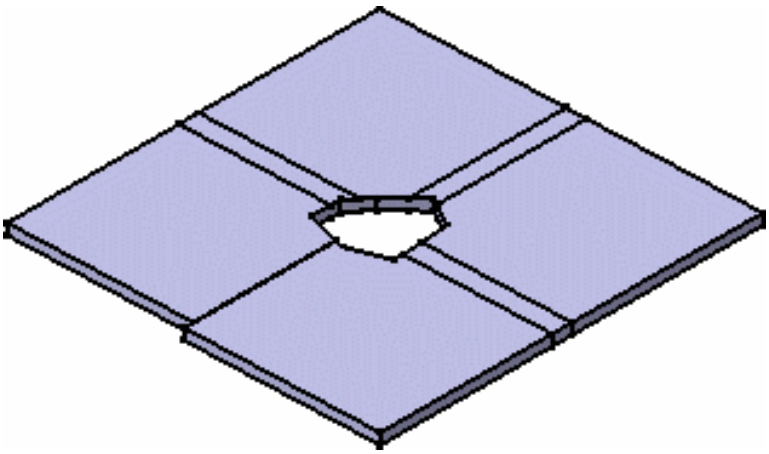
   By default the User Corner Relief  is active in the Corner Relief Definition dialog box.

3. Select the sketch, directly in the document.

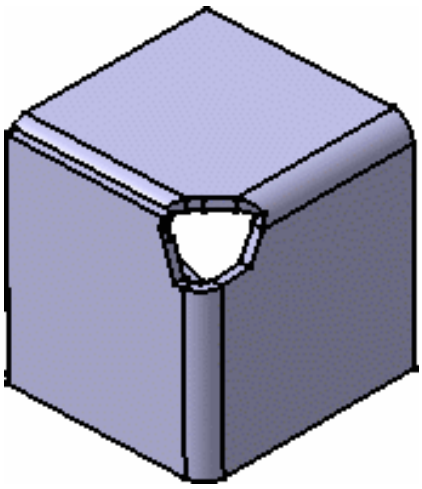


As soon as the sketch has been selected the sketcher icon is displayed in the dialog box allowing you to edit the selected sketch, if needed.

4. Click OK in the Corner Relief Definition dialog box.



5. Fold the part to check the corner relief in 3D.

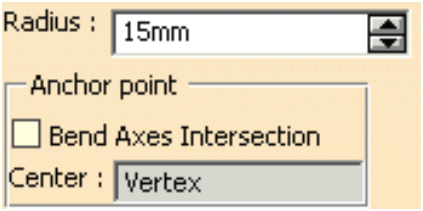


You can use the **Catalog** icon  to open the Catalog Browser.

For more information on catalogs, please refer to the *Component Catalog Editor* documentation.

If you choose another corner relief type, the scenario maybe slightly different:

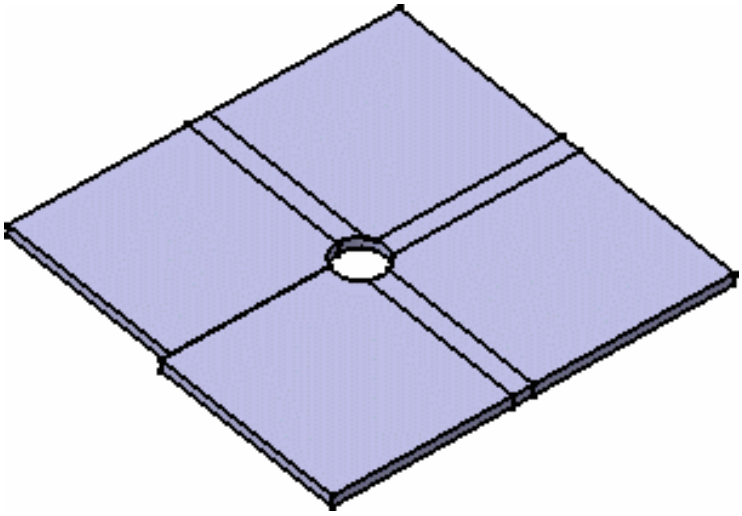
- **circular**: by default the corner relief center is located at the intersection of the bend axes.



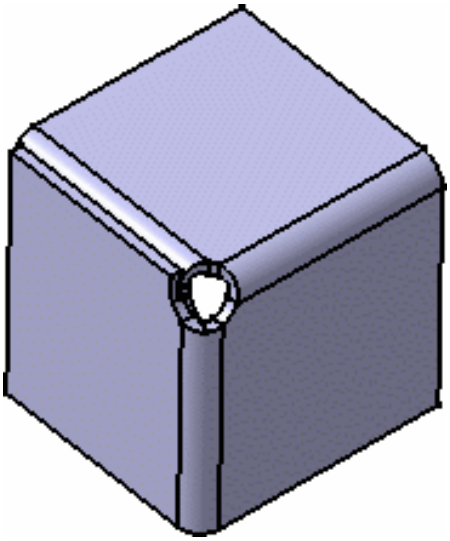
You can select a point as the circle's center.

A radius is proposed by default. It is equal to the bend radius + the thickness. You can change it by:

- **Selecting Formula -> Deactivate** from the contextual menu of the input field and enter a new value,
- or clicking on the  button and entering a new formula.



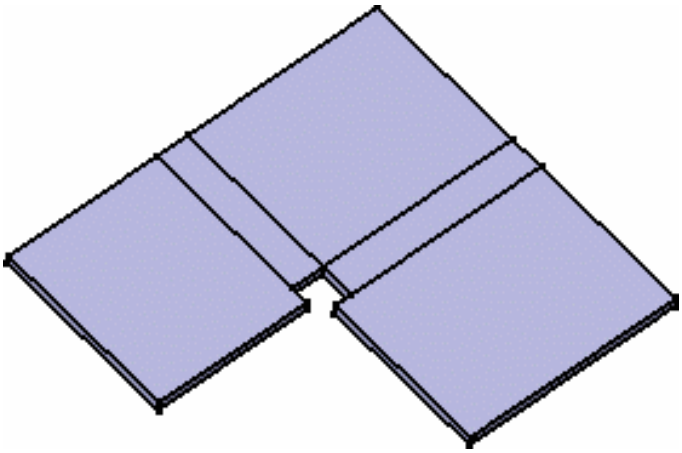
Unfolded circular corner relief



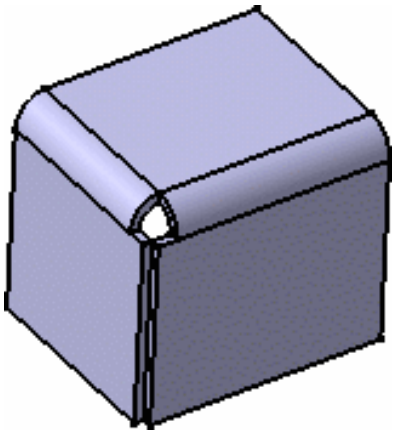
Folded circular corner relief

- **square**: the square corner relief  is created using the bend limits. Its dimensions are defined by the width of the unfolded bends.

  Available between two bends only.



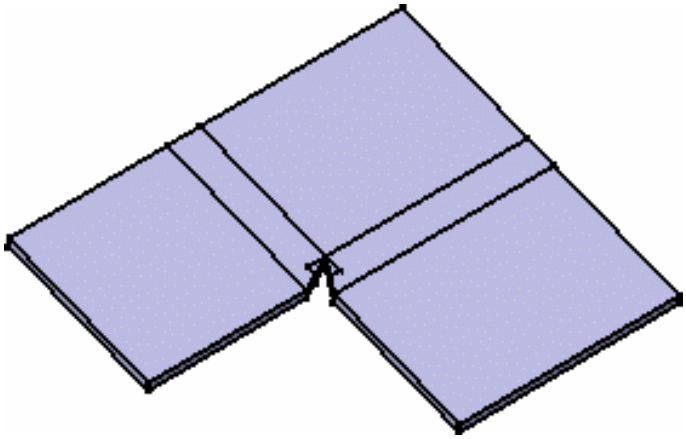
Unfolded square corner relief



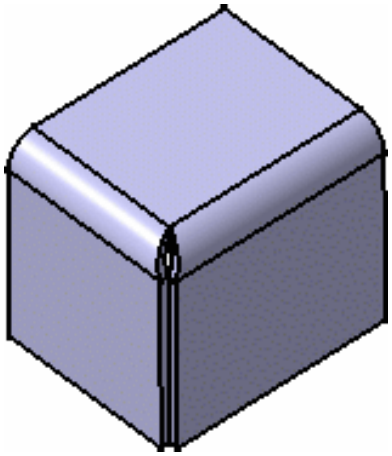
Folded square corner relief

- **triangular**: the triangular corner relief  is created from the intersection point of the inner bend limits towards the intersection points of the outer bend limits with each wall.

  Available between two bends only.



Unfolded triangular corner relief



Folded triangular corner relief

Only the User-defined and Circular corner relief type allow the selection of more than two bends.

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Creating Corners

This task shows how to create one or more corner(s) on a Sheet Metal part, that is to round off sharp edges, much like a fillet between two faces of a Part Design Body. This corner creation operation can be performed indifferently on the folded or unfolded view, and only one support (i.e. the corner when previewed should not lie over two supports).

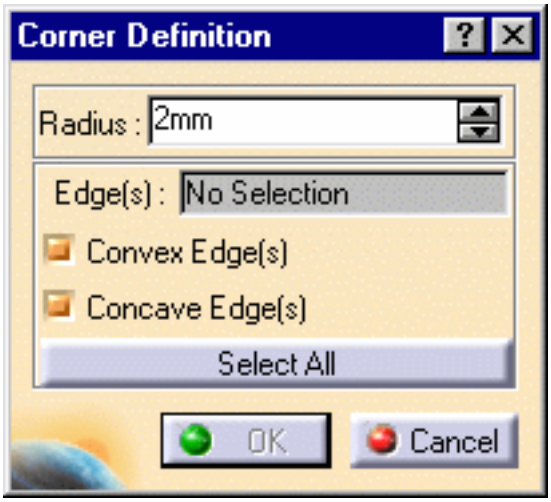
Open the Corners1.CATPart document.

1. Click the **Corner** icon .

   The Corner Definition dialog box is displayed.

2. Set the radius value.

3. Choose the type of edge you wish to round off:



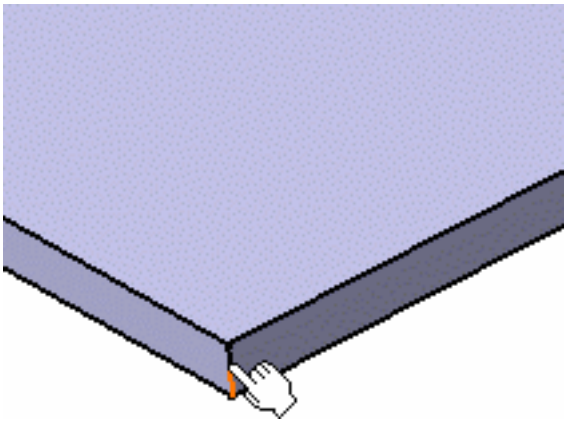
- using the **Select All** button: all convex or concave edges, or all edges of both types
- any edge manually selected

By default both buttons are checked, to allow the selection of any edge type whether manually or automatically.

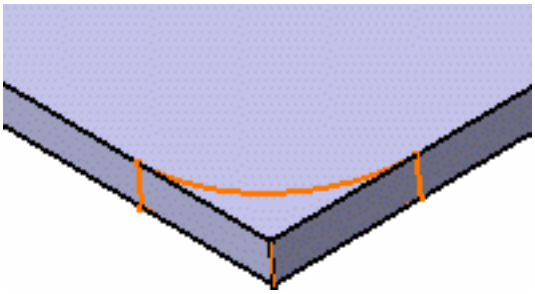
- Once you have selected an edge, you can no longer modify this option, unless you cancel the selection.
- If you check the **Convex Edge(s)** button and you select a concave edge, a warning is issued indicating that you did not select an edge corresponding to the active type.

4. With only the **Convex Edge(s)** button checked, select a sharp edge on a part.

   As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.

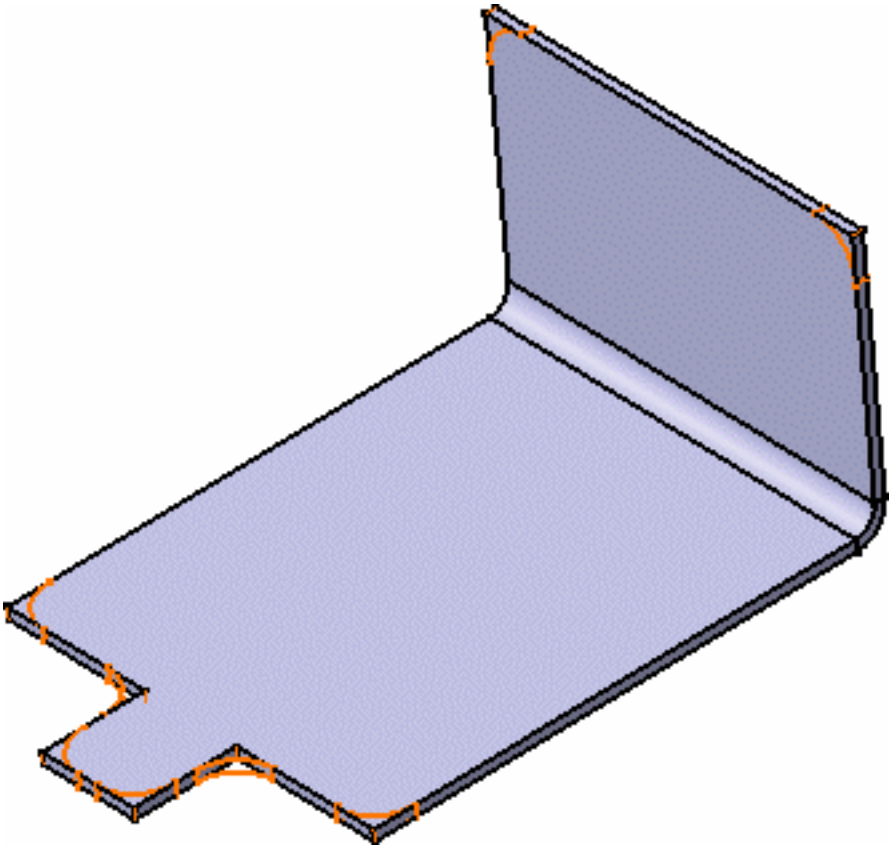


   The corner is previewed on the edge, with the current radius value.



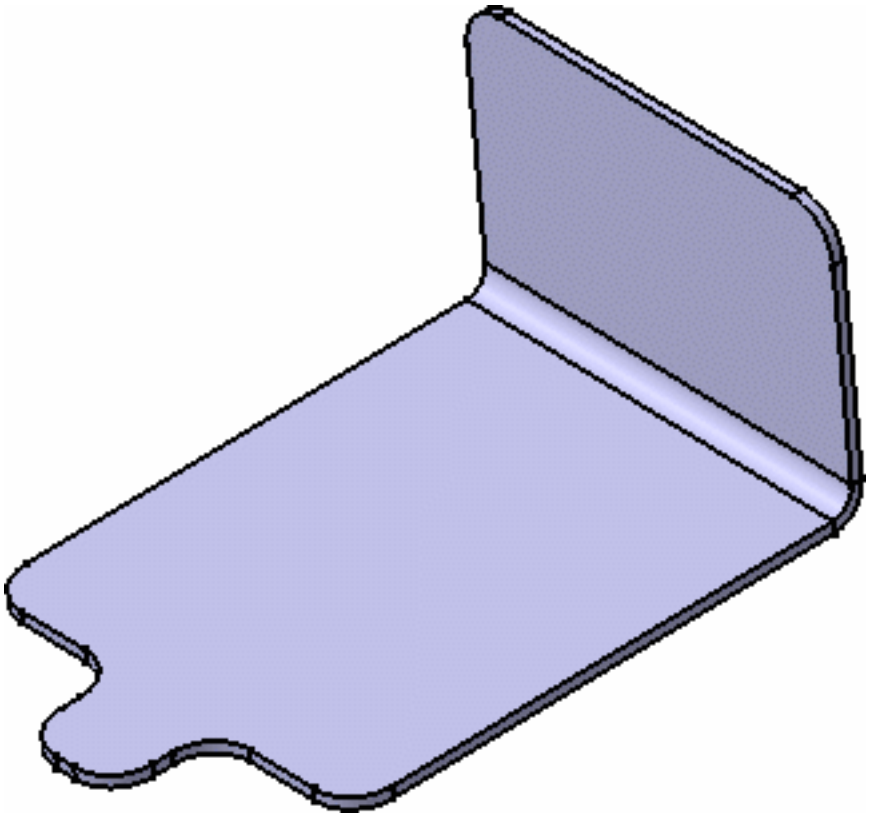
5. Click **Cancel Selection**, make sure that both **Convex Edge(s)** and **Concave Edge(s)** buttons are checked, then click the **Select All** button.

   All sharp edges of the part are selected, the **Select All** button taking into account the chosen type (convex, concave, or both) and the corners previewed.



6. Click OK in the dialog box.

   All sharp edges of the part are rounded off to create smooth corners.



To deselect an edge, simply click it again. For quick selection in a complex part, you can select all edges with the **Select All** check button, then deselect one or two edges.

- When you select an edge that is not sharp, such as the edge between a wall and a bend for example, a warning is issued.
- As you select more edges, the **Edge(s)** field of the dialog box is updated.
- When using the **Select All** button, you select all edges (whether concave, convex, or both) present at the time. If when modifying the Sheet Metal part, new edges are created, these will not be automatically rounded off.



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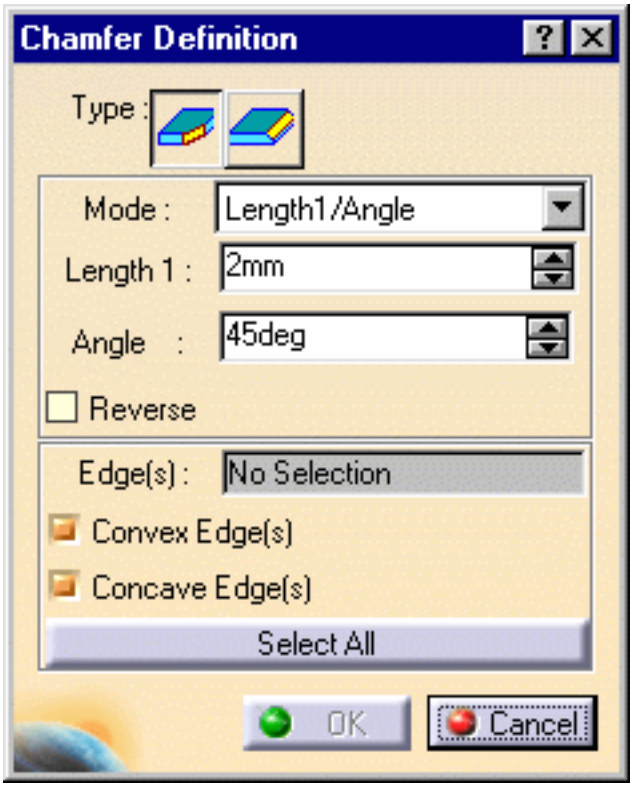
Creating Chamfers

This task shows how to create one or more chamfer(s) on a Sheet Metal part, that is to cut off, or fill in sharp edges of Sheet Metal parts.
This chamfer creation operation can be performed indifferently on the folded or unfolded view, and only one support (i.e. the chamfer when previewed should not lie over two supports).

Open the Corners1.CATPart document.

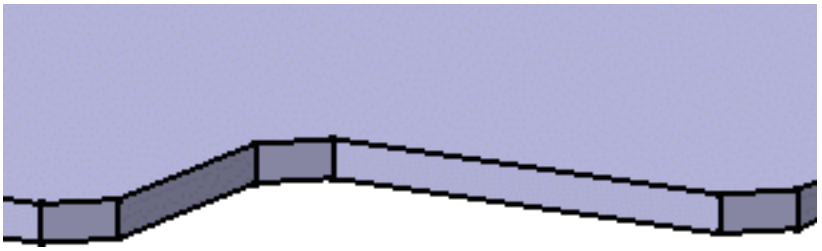
1. Click the **Chamfer** icon .

   The Chamfer Definition dialog box is displayed.

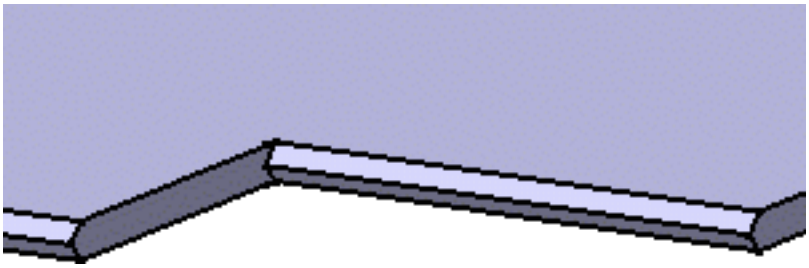


2. Choose the chamfer **Type**:

- **Thickness chamfer** : to be able to select edges that represent the thickness of the part



- **Welding chamfer** : to be able to select edges that represent the area of the part where it can be welded to another part.



With the Thickness chamfer type only, you can choose the type of edge you wish to chamfer:

- using the **Select All** button: all convex or concave edges, or all edges of both types
- any edge manually selected

By default both buttons are checked, to allow the selection of any edge type whether manually or automatically.

- Once you have selected an edge, you can no longer modify this option, unless you cancel the selection.
- If you check the **Convex Edge(s)** button and you select a concave edge, a warning is issued indicating that you did not select an edge corresponding to the active type.

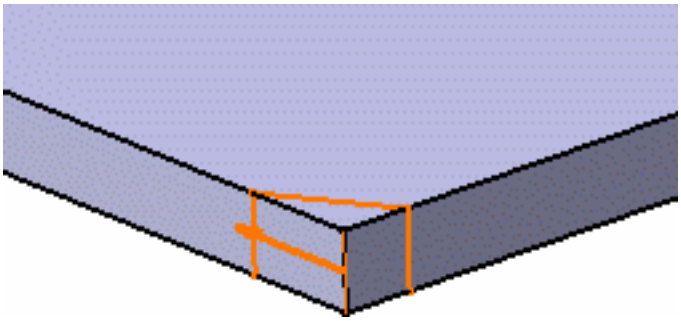
3. Select a sharp edge on a part.

   The chamfer is previewed on the edge.

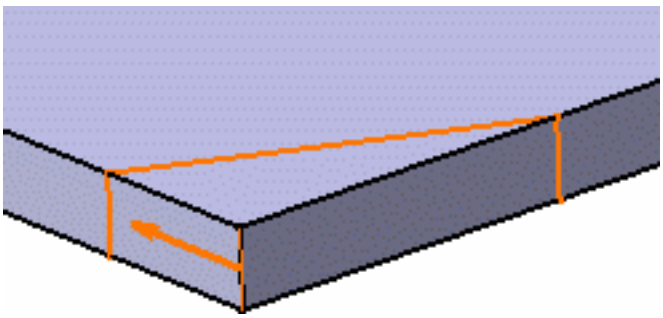
As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.

4. Choose a chamfer **Mode**. You can either enter:

- a length value and an angle: the length is computed on one side of the edge and the angle from the chamfer's limit on the same side



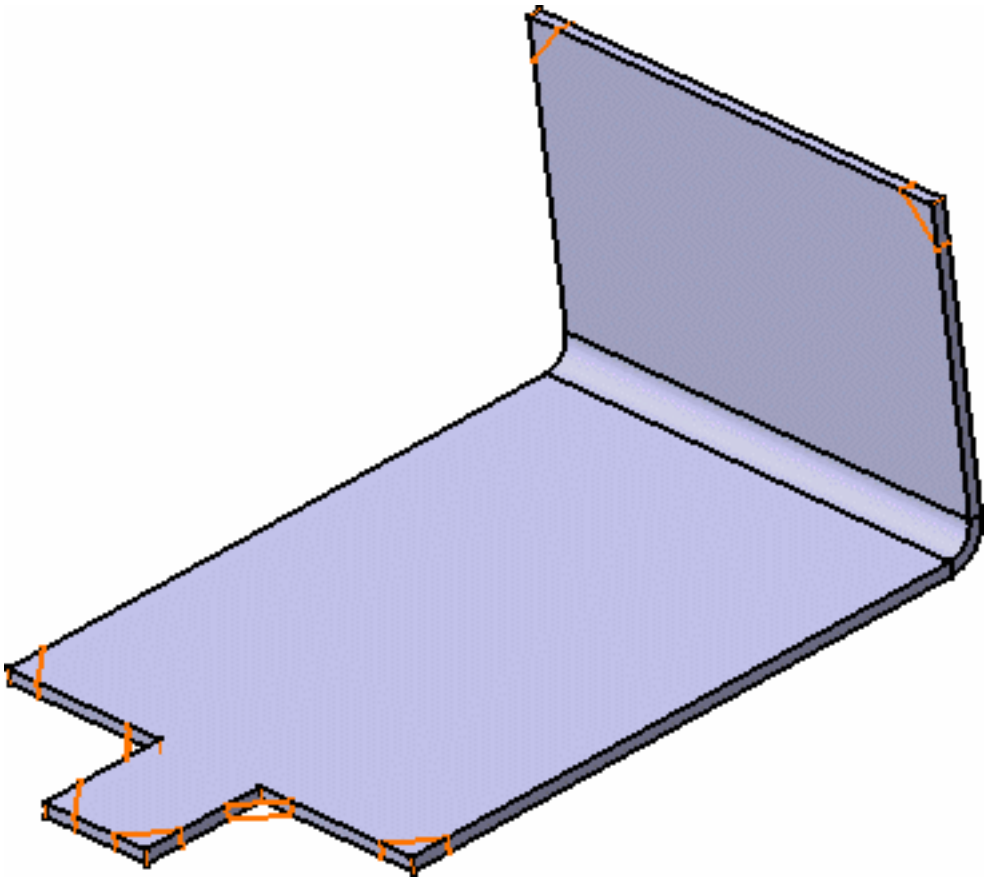
- two lengths: these lengths are computed from the selected edge on both sides.



You can use the Reverse button to inverse all edges' side, on which the values are taken into account;
Use the arrow displayed on each edge to locally invert only one edge.

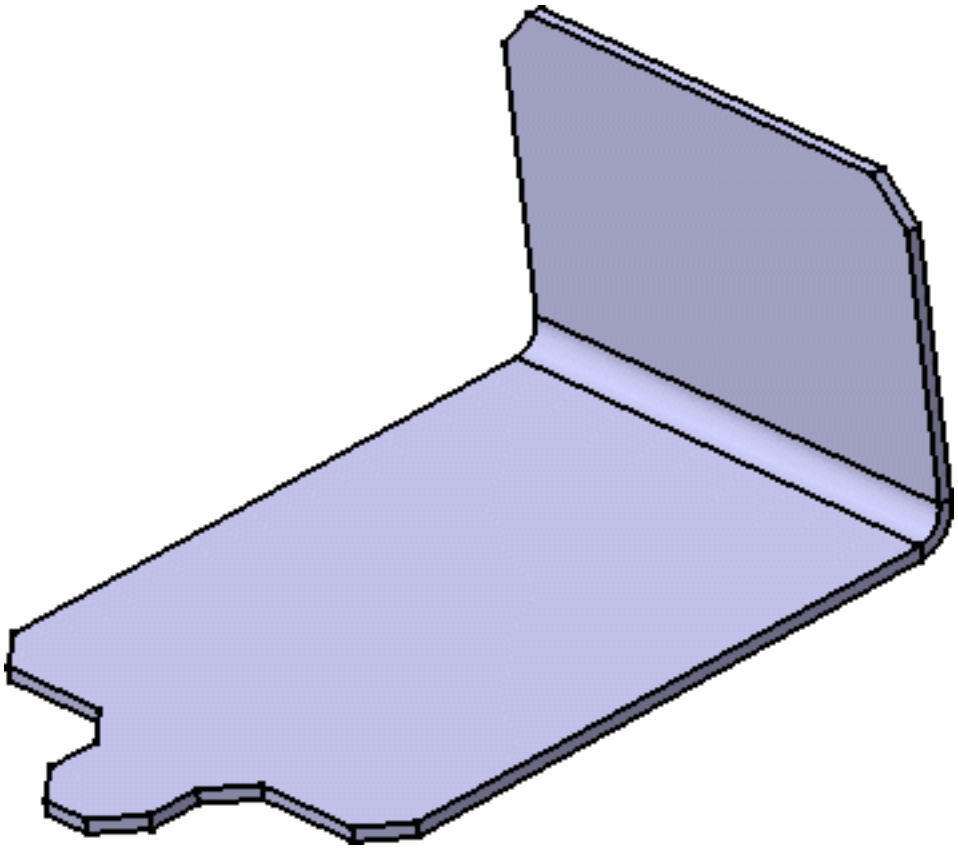
5. Click **Cancel Selection** then, make sure that both **Convex Edge(s)** and **Concave Edge(s)** buttons are checked, and click the **Select All** button.

   All sharp edges of the part are selected, the **Select All** button taking into account the chosen type (convex, concave, or both) and the chamfers previewed.



6. Click OK in the dialog box.

   All sharp edges of the part are cut off or filled in.



To deselect an edge, simply click it again. For quick selection in a complex part, you can select all edges with the **Select All** button, then deselect one or two edges.

- When you select an edge that is not sharp, such as the edge between a wall and a bend for example, a warning is issued.
- As you select more edges, the **Edge(s)** field of the dialog box is updated.
- When using the **Select All** button, you select all edges (whether concave, convex, or both) present at the time. If when modifying the Sheet Metal part, new edges are created, these will not be automatically chamfered.

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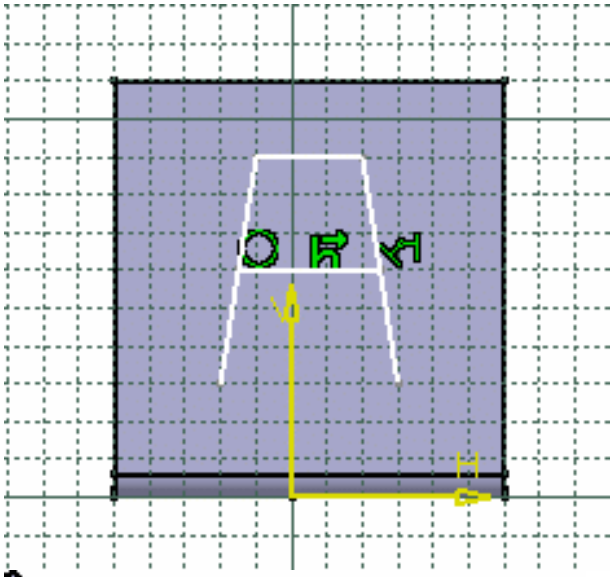
Mapping Elements

This task shows how to create curves or points from a sketch (as designed using the Sketcher) or from existing curves or points, onto a Sheet Metal part; and to fold/unfold it, just as other Sheet Metal elements.
This is especially useful when:

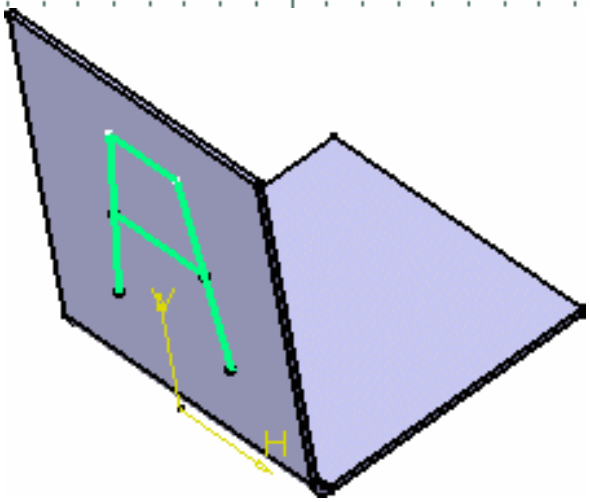
- you want to generate a logotype
- you want to define an area for chemical milling
- you want to create a cutout (pocket) to solve the overlapping of walls for example (the overlapping can be checked with the Sheet Metal Production product).

Open the Mapping1.CATPart document.
If you use the New SheetMetal Design, open the NEWMapping1.CATPart document.

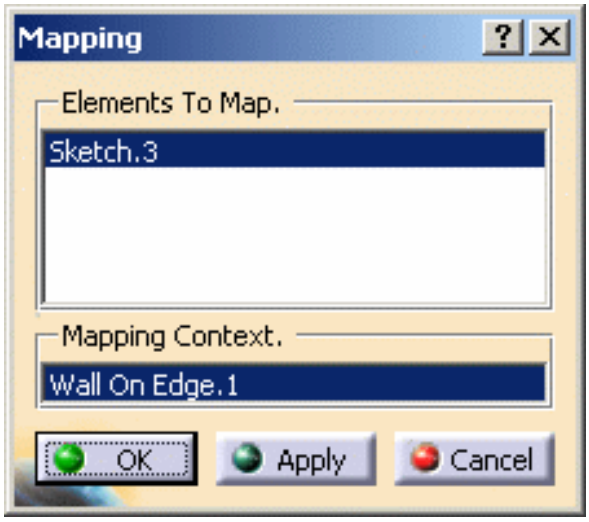
1. Click the **Sketcher** icon , select the wall onto which the curve should lie, and draw the sketch you wish.
   This is the sketch that will be mapped onto the part.
2. Exit the Sketcher .



The 3D part looks like this:



3. Make sure the sketch is selected, and click the **Fold / Unfold Points or Curves** icon .



The Elements To Map definition dialog box is displayed.
It indicates which elements have been selected for mapping.

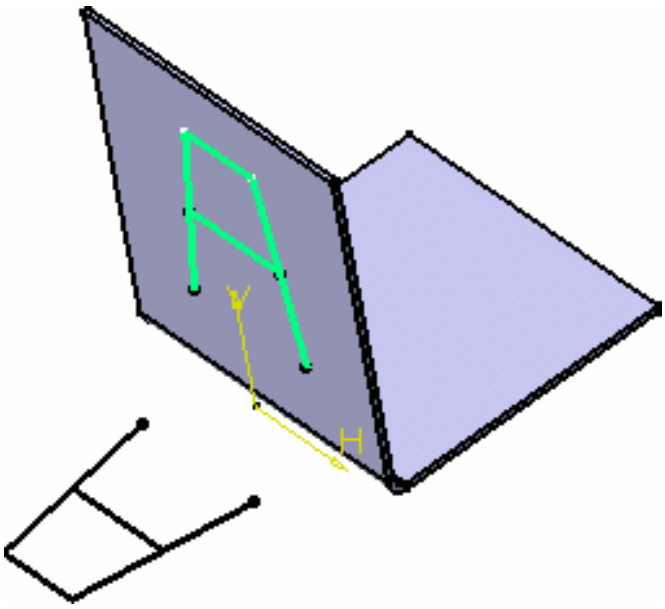
You can manage the list of elements:

- to remove an element, select it from the list and use the **Clear selection** contextual menu
- to add an element, select it directly in the geometry.
  Order in the list does not matter.

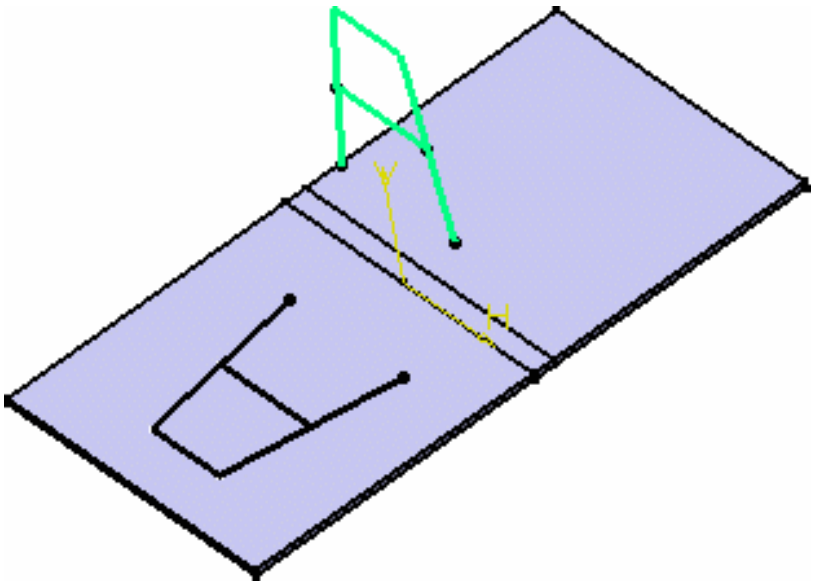
4. Select the **Mapping Context**, that is the element of the part on which the curve should be generated when folding or unfolding.

The **Mapping Context** is not necessarily the support element on which the element to be mapped has been drawn. Indeed, by default, the **Mapping Context** is the last Sheet Metal feature that has been created or modified, that is the current feature in the specification tree.

5. Click OK.
   A curve is created and added in the specification tree.



Folded view of the curve mapping



Unfolded view of the curve mapping

- You can select several sketches/curves/points to be mapped at a time.
- Mapped curves can be created across several walls and bends.



Back Up Next

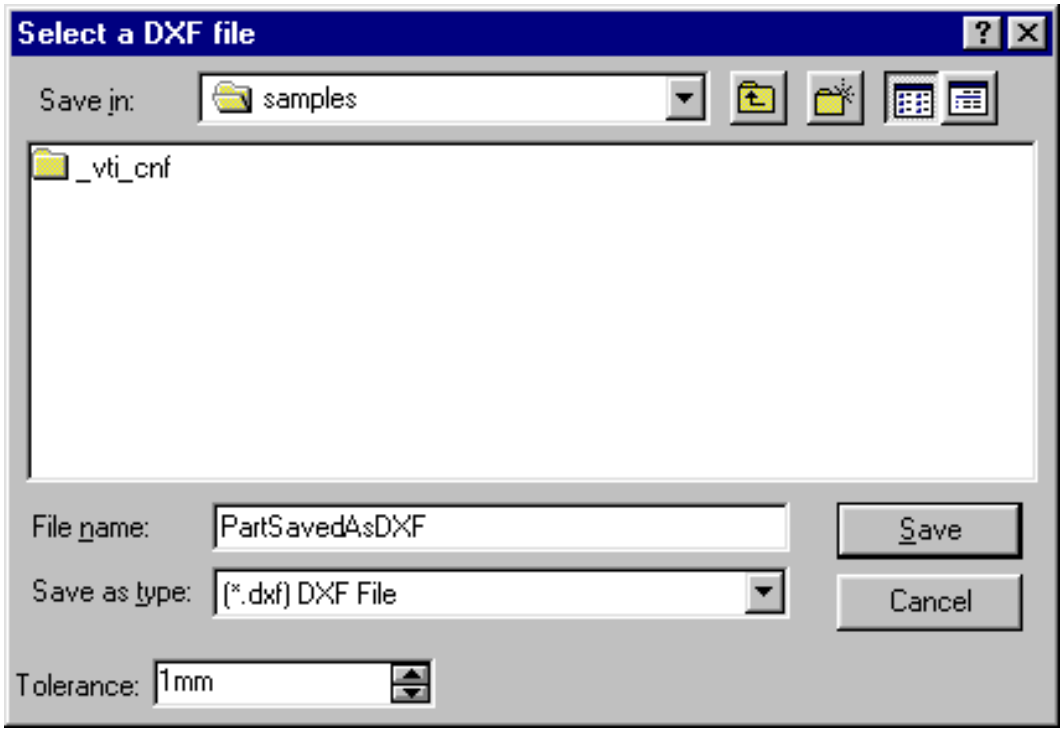
Saving As DXF

This task shows how save the generated geometry as a DXF document.

If working in the NEW SheetMetal Design workbench, this task is new.

1. Click the **Save As DXF** icon .

   The **Select a DXF file** dialog box is displayed allowing you to navigate to the correct location.



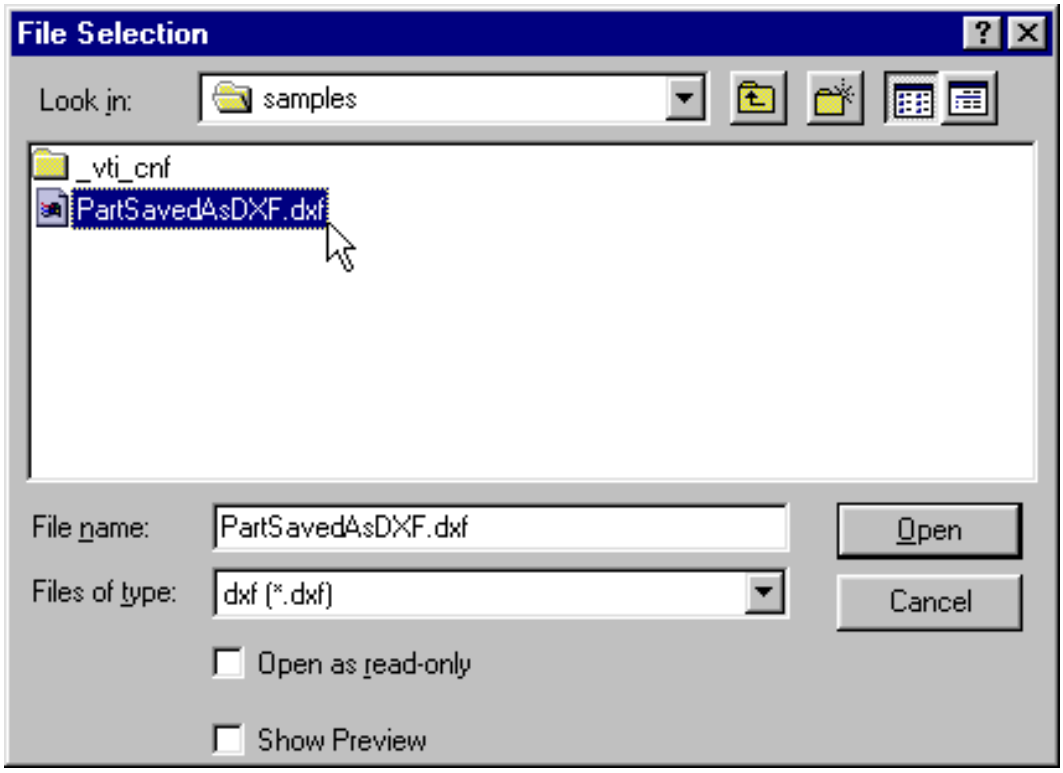
2. Indicate the correct path and file name (we saved it as PartSavedAsDXF.dxf).
3. Click **Save**.

The geometry has been saved, and can be imported as a DXF file in any system supporting this type.

4. Close the CATPart Document.
5. Click **File -> Open**.

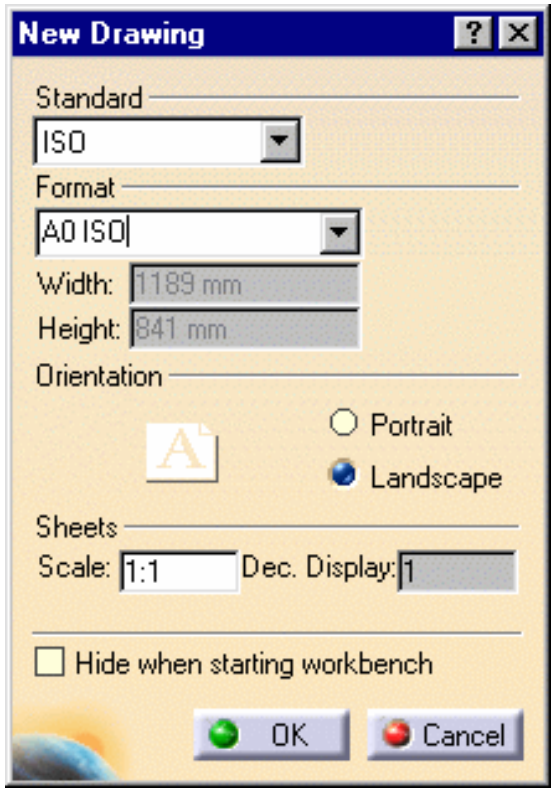
6. From the File Selection dialog box, choose the .dxf file type, then select the saved part (PartSavedAsDXF.dxf).

7. Click Open.

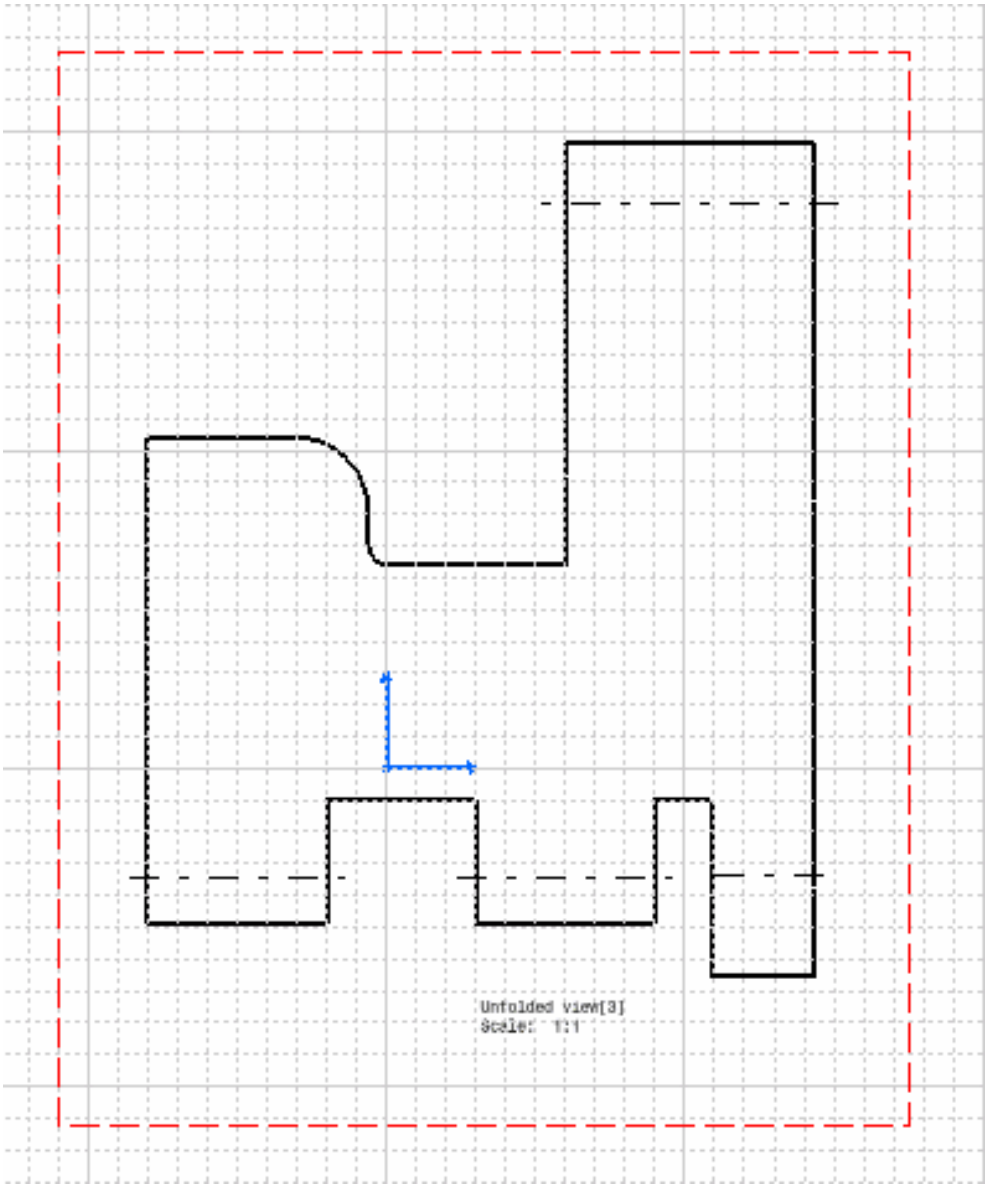


The New Drawing dialog box is automatically displayed because the .dxf type is recognized as being a drafting type.

8. Click OK in the New Drawing dialog box.



The unfolded view of the part is created within the CATDrawing document.



Note that the axes of bends and planar hems, tear drops, or flanges are automatically displayed on the drawing.

Refer also to DXF/DWG Settings.

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Reference Elements

You can create wireframe elements within the Sheet Metal Design workbench:

 Create points: click this icon, choose the point creation type, and specify parameters

 Create lines: click this icon, choose the line creation type, and specify parameters

 Create planes: click this icon, choose the plane creation type, and specify parameters

Back Up

Advanced Tasks

The Advanced Tasks section explains how to use further functions that may not be as common as the ones described in the Basic Tasks section, as well as the integration of the Sheet Metal Design workbench and elements with other workbenches.

Integration with Part Design
Integration With Weld Design
Designing in Context
PowerCopies Management
Browsing the Sheet Metal Catalog
Looking For Sheet Metal Features

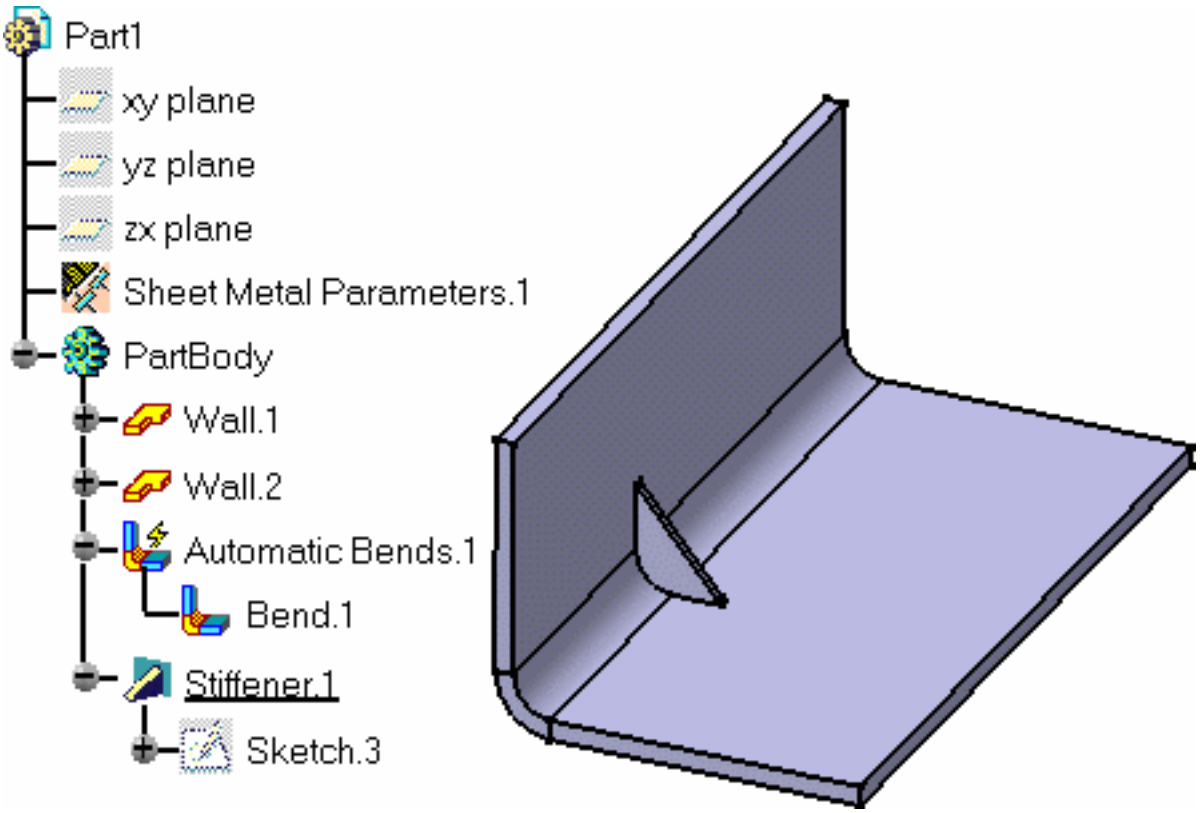
Integration With Part Design

You can open the Stiffener1.CATPart document from the samples directory to replay the scenario.
If you use the NEW SheetMetal Design workbench, open the NEWStiffener1.CATPart document.

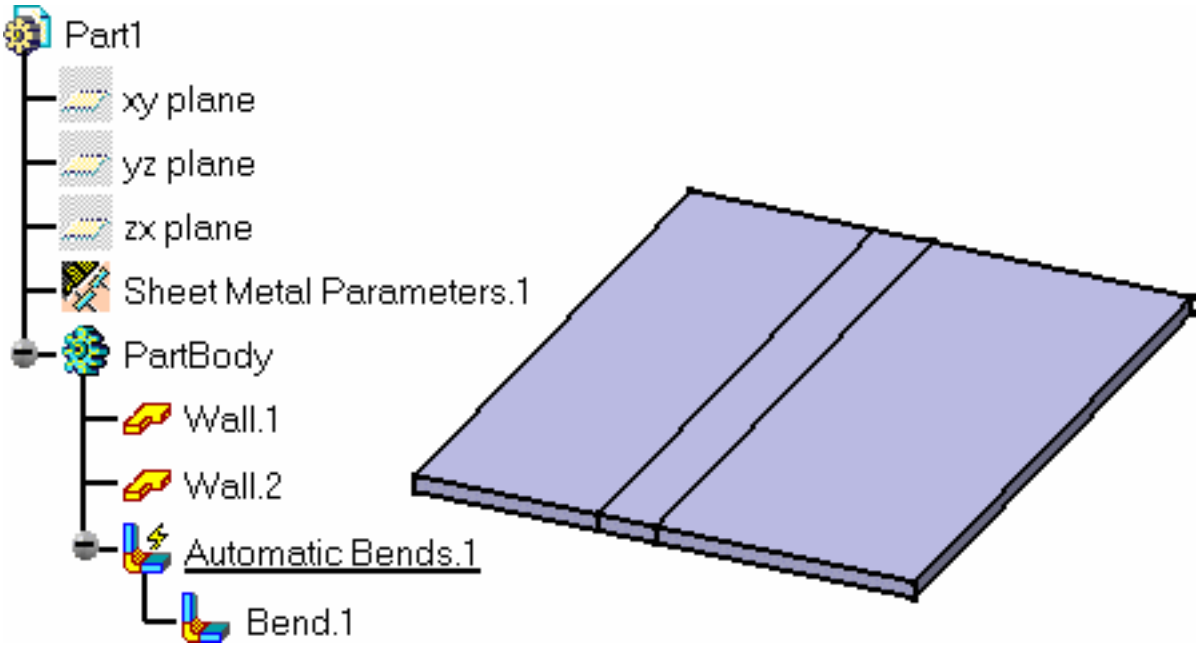
In a CATPart document, you may have Part Design features and Sheet Metal features according to the following rules:

- Part Design features can be created before Sheet Metal features.
- a Part Design feature can also be created after Sheet Metal features as long as the part is in folded view.
- in the unfolded view, the Part Design feature will not be displayed.
- it is no longer possible to create Sheet Metal features after this last Part Design feature in folded view.

1. Create two walls with an Automatic or Cylindrical Bend.
2. Switch to Part Design workbench.
3. Launch the Sketcher and draw an oblique line in the yz plane.
4. Click the icon  to create a **Stiffener**.

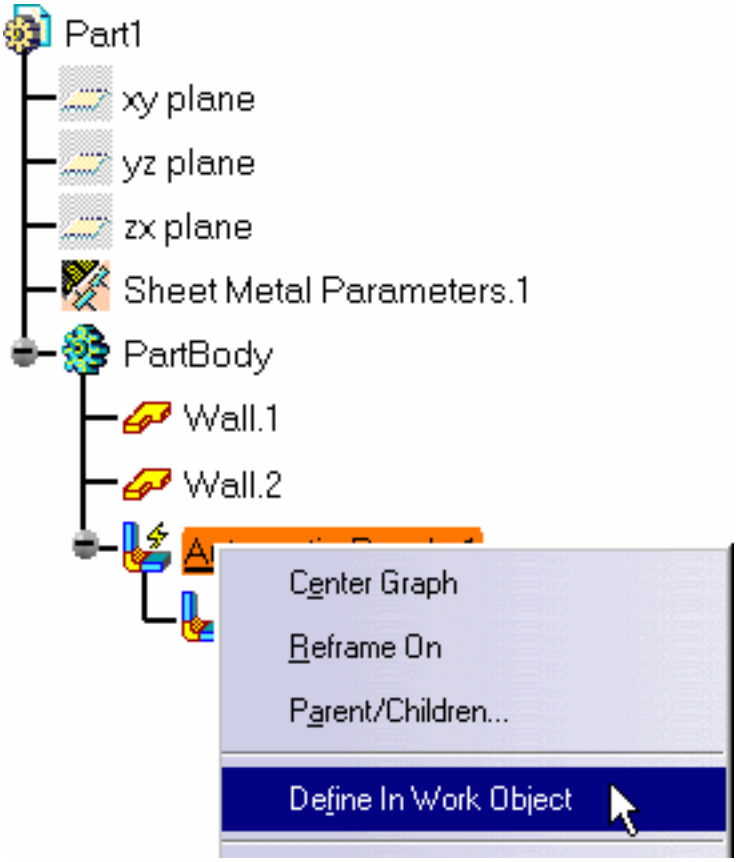


5. Switch to the Sheet Metal workbench.
6. Click the **Unfold** icon .



The stiffener is not displayed on the unfolded view.

To add a new Sheet Metal feature and switch to the Sheet Metal Design workbench, select the Bend for example and right-click the **Define In Work Object** item.



The new Sheet Metal feature will be added after the Bend but before the Stiffener.



Up Next

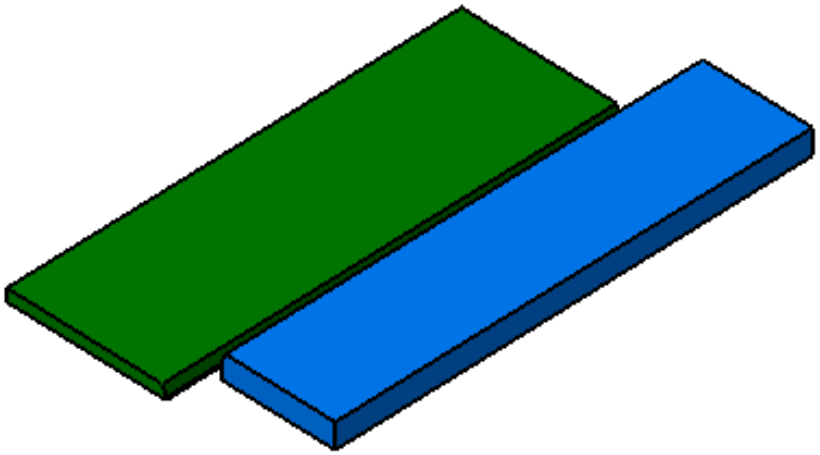
Integration With Weld Design

You can open the Integration.CATProduct document from the samples directory to replay the scenario.
If you use the New SheetMetal Design workbench, open the NEWIntegration.CATProduct document.

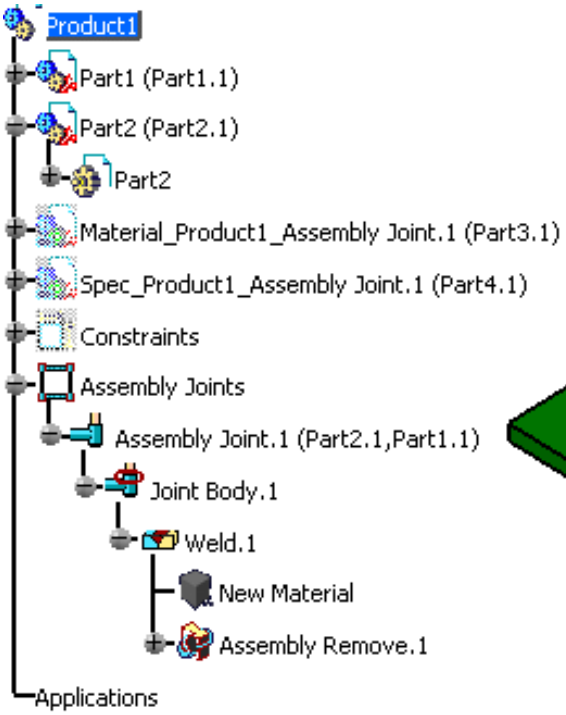
In a CATProduct document, you may have Weld Design features and Sheet Metal features according to the following rules:

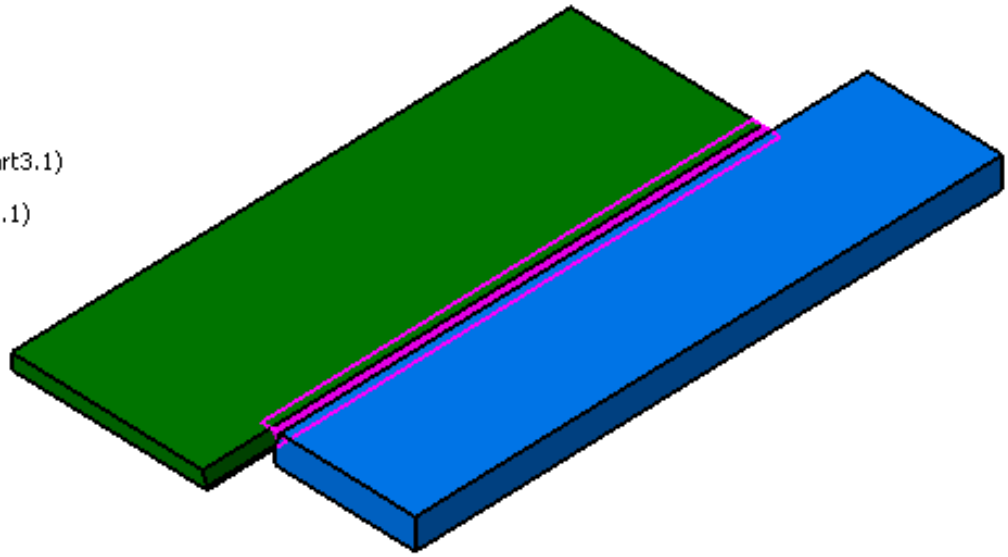
- Weld Design features are created after Sheet Metal features.
- in the unfolded view, the Weld Design feature will not be displayed.
- it is no longer possible to create Sheet Metal features after this last Weld Design feature.

1. Create two walls.
2. Switch to Weld Design workbench.



3. Create a weld (in our scenario, we create a Single-V Butt weld)





To add a new Sheet Metal feature and switch to the Sheet Metal Design workbench, select any of the two walls and make it current.

The new Sheet Metal feature will be added after the Bend but before the Stiffener.



Designing in Context

This section explains and illustrates how to create and use various kinds of features.
The table below lists the information you will find.

Design in context: from a CATProduct document, create a new CATPart document, and create a complete SheetMetal Part.

Modify the design: modify another part from another document and update the SheetMetal Part.

Back Up Next

Designing in Context

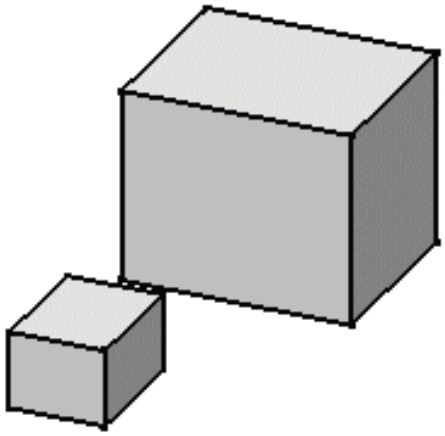
This task explains how to create a Sheet Metal part in an Assembly context.

Open the Scenario2.CATProduct document from the samples directory.
If you use the New SheetMetal Design, open the NEWScenario2.CATProduct document.

You are in Assembly Design workbench.

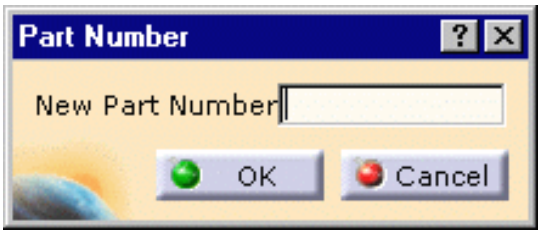


The document contains two parts.

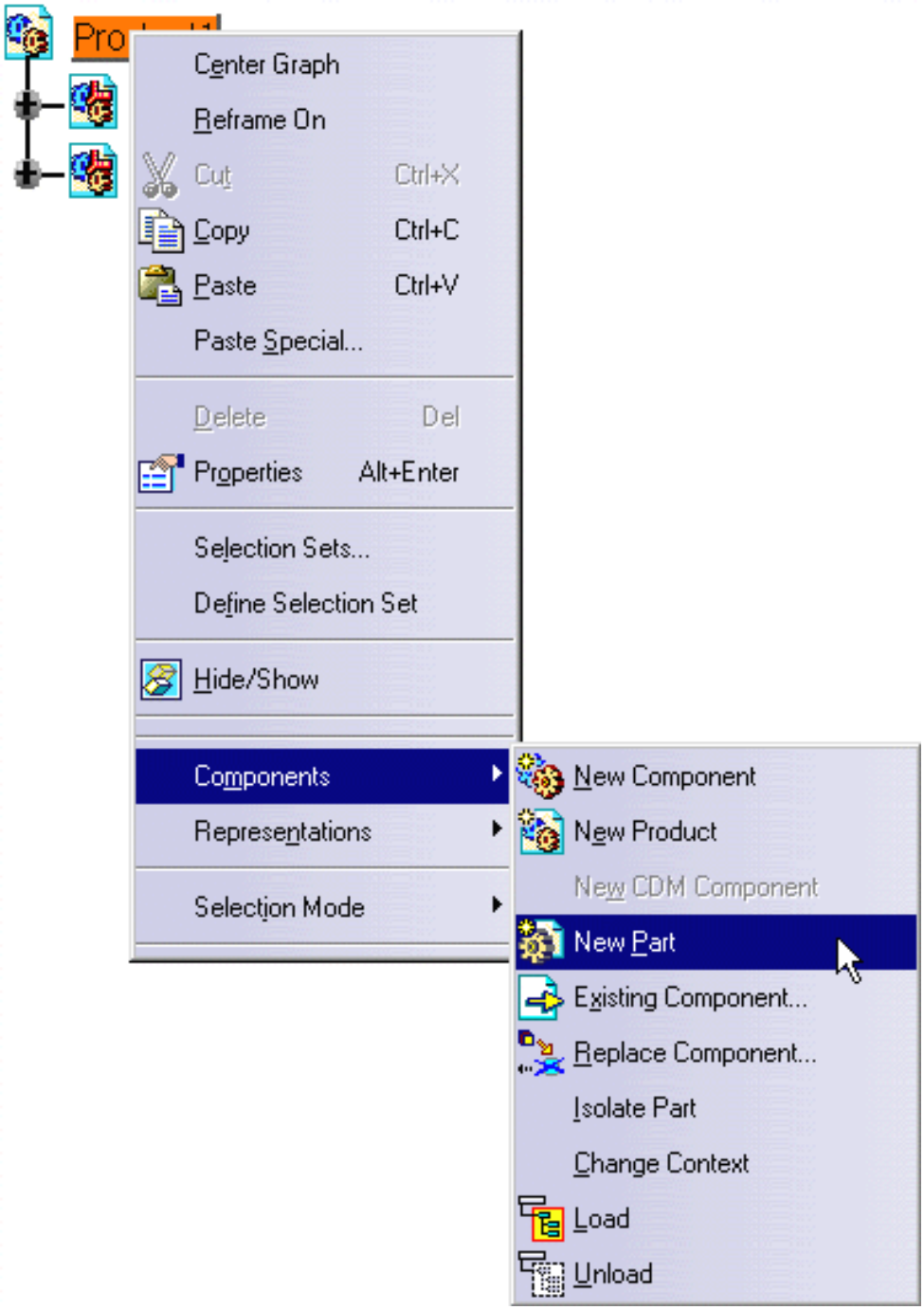


1. Right-click **Product1** in the specification tree and select **Components -> New Part...**

   Provided the **Manual Input** option is checked in **Tools -> Options -> Infrastructure -> Product Structure**, **Product Structure** tab, the Part Number dialog box is displayed:



2. Enter **Part3** in the New part Number field and click **OK.**
   A New Part dialog box proposes two locations to define the origin point.
   For more information, refer to Inserting a New Part, in the Basic Tasks of the *Product Structure User's Guide*.



3. Click **No** to locate the part origin according to the **Product1** origin point.

   Make sure you are in **Design Mode:**
   - Select **Product1**
   - Choose **Edit -> Representations ->Design Mode**

4. Expand the tree and activate the **Part3** Part body by double-clicking.

5. Switch to Sheet Metal Design workbench.



6. Select the **SheetMetal parameters** icon  to create the Sheet Metal characteristics for the part:

   - 1mm for the Thickness
   - 3mm for the Bend radius
   - Linear for the Bend extremities

   and click OK.

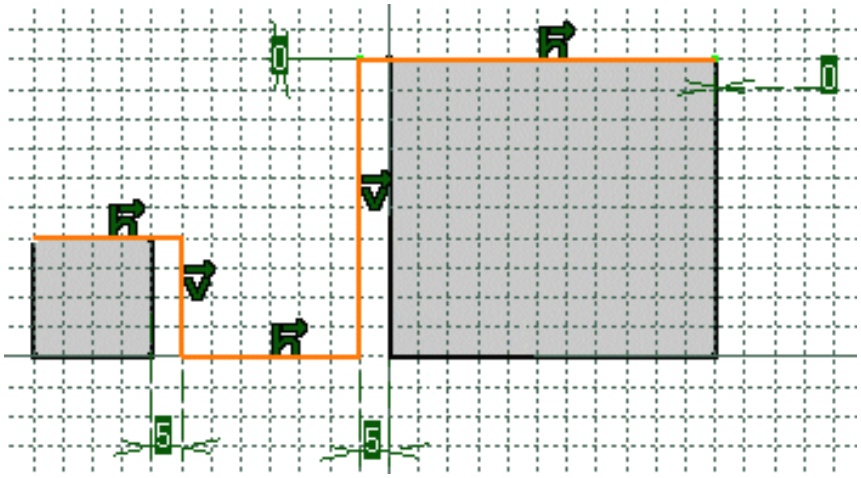
7. Choose the **Tools -> Options -> Mechanical Design -> Part Design, General** tab and check the **Keep link with selected object** option, then click OK.

8. Click the **Sketcher** icon  and select the **zx plane**.

9. Select the **Profile** icon .

10. Sketch the contour and set the constraints as shown below:

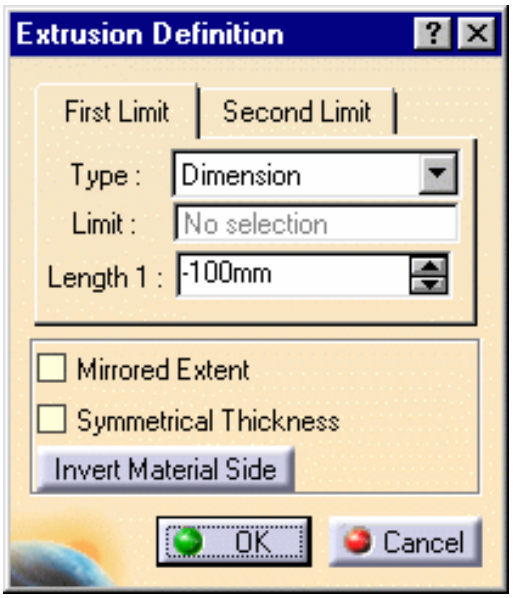
- 5mm between the Sheet Metal vertical walls and each pad
- 0mm between the Sheet Metal horizontal walls and each pad top
- 0mm between the last point of the Sheet Metal sketch and the right pad side.



11. Click the **Exit** icon  to return to the 3D world.

12. Select the **Extrusion** icon .

13. Select the Sheet Metal profile.
The Extrusion Definition dialog box appears.



14. Enter 70mm for **Length1** then click **OK**.

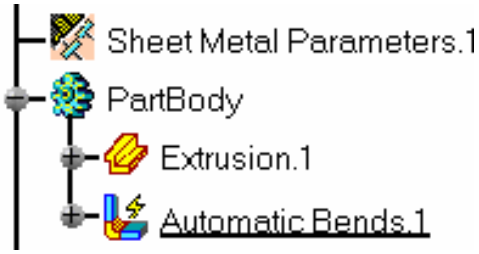
The Material Side should be set to the outside.

15. Select the **Automatic Bends** icon .

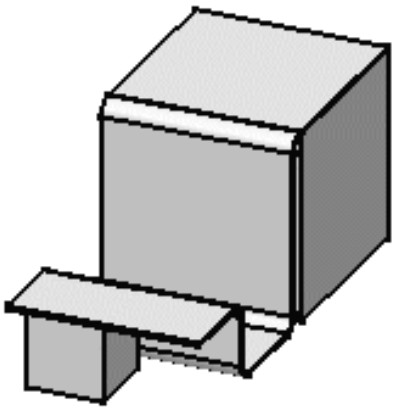
The bends are created.

The new features are shown in the specification tree:

- **Extrusion.1** with five walls
- **Automatic Bends.1** with four bends.



The Sheet Metal part looks like this:





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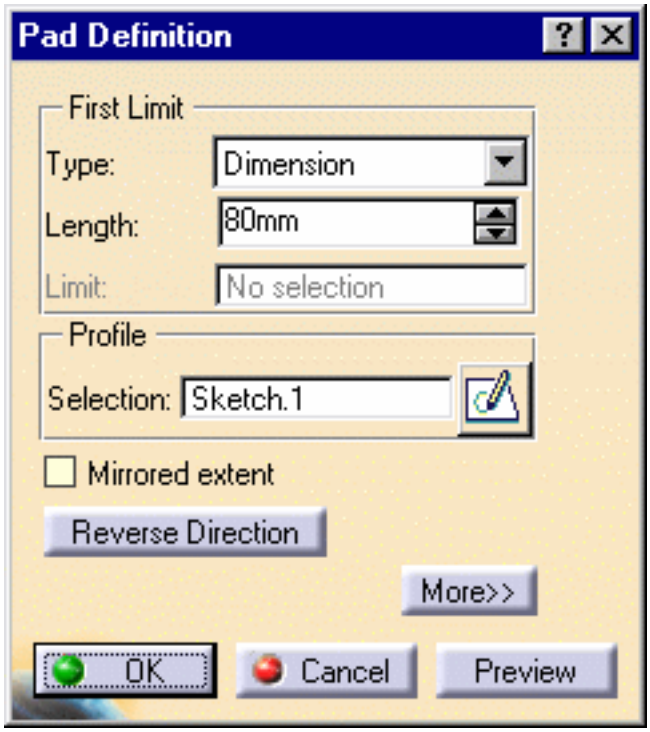
Modifying the Design

In this task, you are going to modify the height and the sketch of Pad.1.

The Scenario2.CATProduct document is open from the previous task.
If not, open the Scenario2_2.CATProduct document from the samples directory.

1. Double-click **Part1\PartBody\Pad.1** in the specification tree.

   The dialog box is displayed.



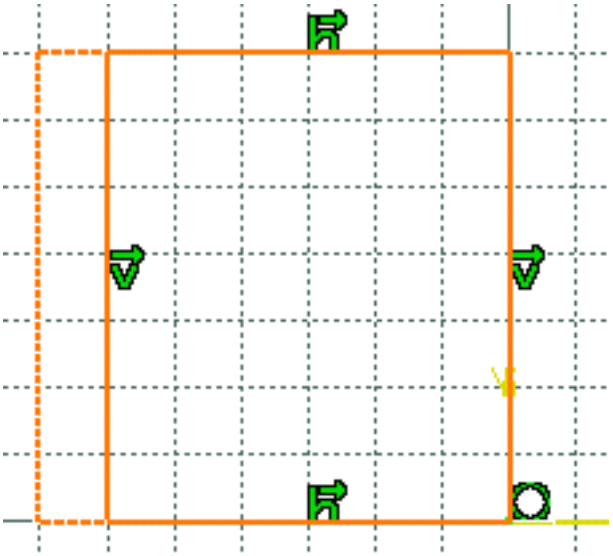
2. Enter 40mm for the **Length** and click **OK**.
   The pad is updated.

3. Double-click Part3 and **Update** the Sheet Metal part using the **Update** icon.

4. Double-click **Part1\Pad.1\Sketch.1**.

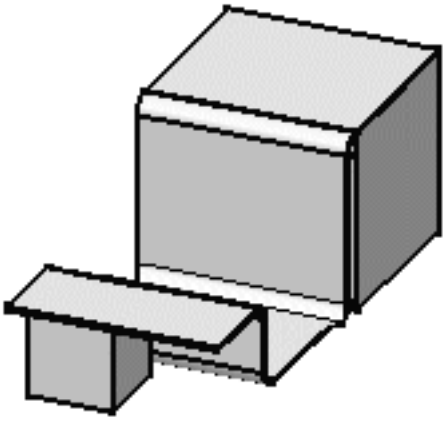
5. Modify the sketch:

6. Click the **Exit** icon  to return to the 3D world.



The constraints are respected.

After the Part3 update, the document looks like this:





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Managing Power Copies



Create PowerCopies: Select the **Insert ->Advanced Replication Tools -> PowerCopy Creation** command, select the elements making up the PowerCopy from the specification tree, define a name for the PowerCopy and its reference elements then choose an icon for identifying it.



Instantiate PowerCopies: Select the **Insert -> Instantiate From Document** command, select the document or catalog containing the powercopy, complete the **Inputs** within the dialog box selecting adequate elements in the geometric area.



Save PowerCopies into a Catalog: Select the PowerCopy from the specification tree, select the **Insert -> Advanced Replication Tools -> PowerCopy Save In Catalog...** command, enter the catalog name and click Open.

Back Up Next

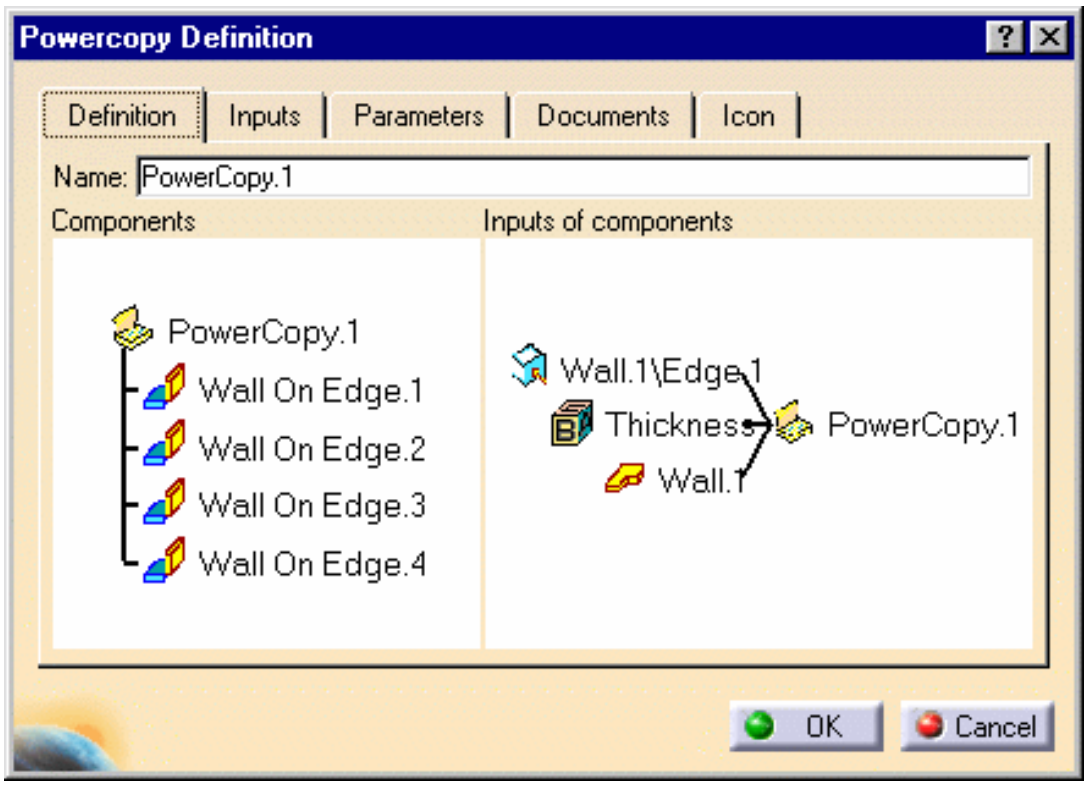
Creating PowerCopy Features

This task shows how to use create PowerCopy elements, to be reused later.
A PowerCopy is a set of features (geometric elements, formulas, constraints and so forth) that are grouped in order to be used in a different context, and presenting the ability to be re-specified according to the context when pasted.
This PowerCopy captures the design intent and know-how of the designer thus enabling greater reusability and efficiency.

Open the PowerCopyStart.CATPart document.

1. Select the **Insert ->Advanced Replication Tools -> PowerCopy Creation** menu item.
   The **PowerCopy Definition** dialog box is displayed.
2. Select, from the specification tree, the elements to be included in the PowerCopy.

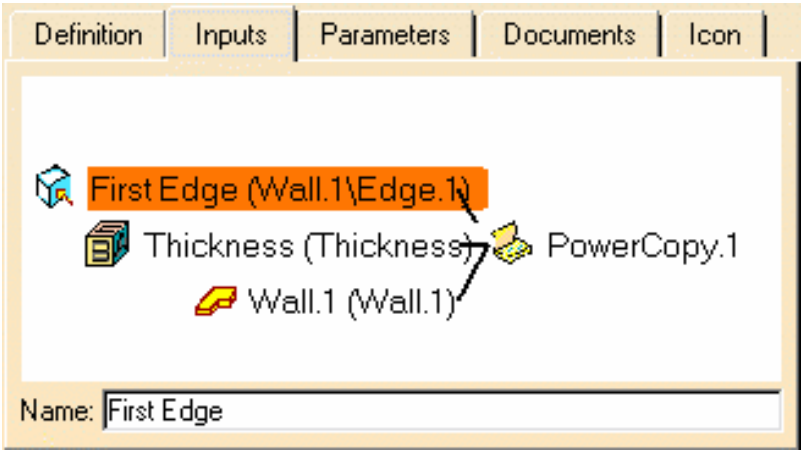
   The **PowerCopy Definition** dialog box is automatically filled with information about the selected elements.



3. Define the PowerCopy as you wish to create it:

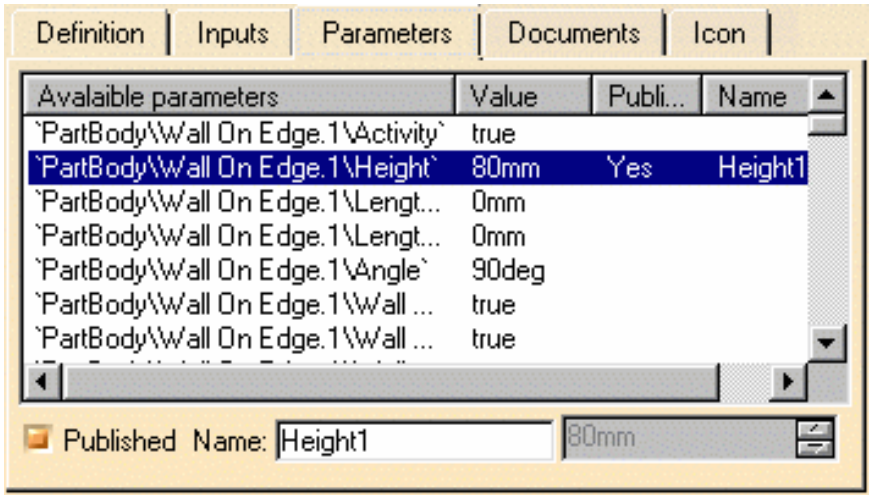
   The **Definition** tab lets you assign a name to the PowerCopy and presents its components in the 3D viewer.

The **Inputs** tab lets you rename the reference elements making up the PowerCopy.



You can do that for clarification purposes as to their roles, by selecting the elements in the viewer and entering a new name in the **Name** field.
In this example, we renamed all three elements and in brackets you still can read the elements' default name based on their type.

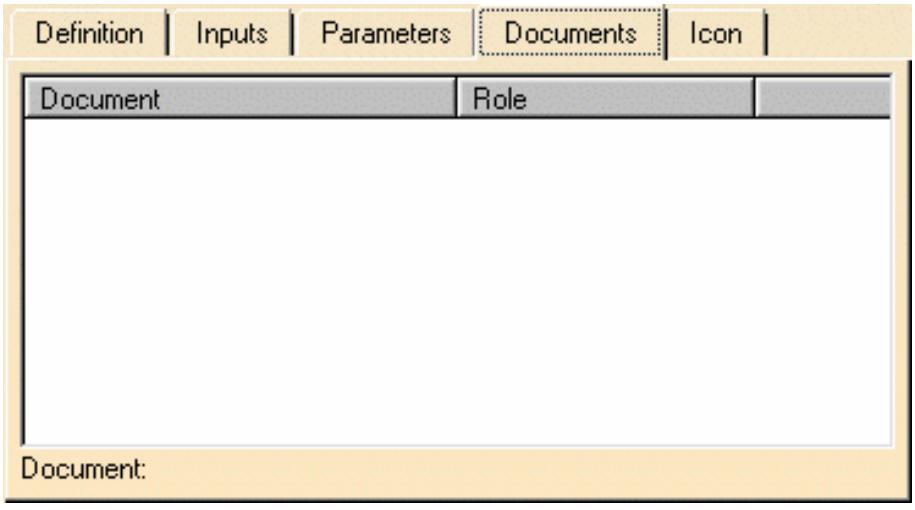
The **Parameters** tab lets you define which of the parameter values used in the PowerCopy you will be able to modify at instantiation time.



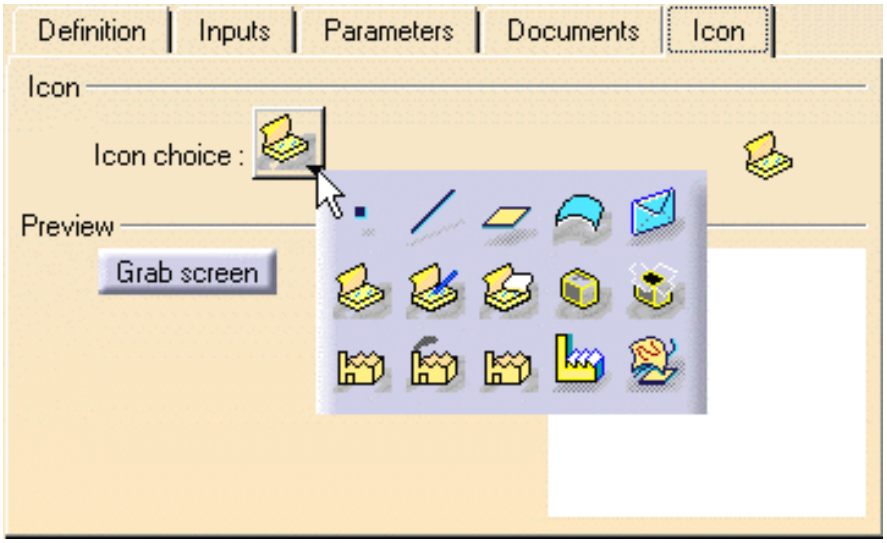
Simply check the Published button.

Use the **Name** field to give a more explicit name to the element.

The Documents tab shows the complete path and role of Design tables that are referenced by an element included in the Power Copy.

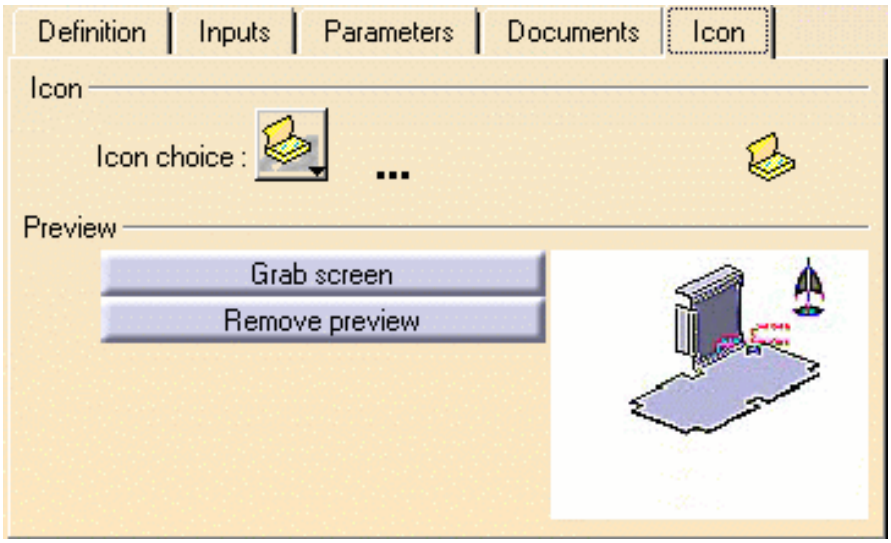


The **Icon** tab lets you modify the icon identifying the PowerCopy in the specifications tree.



A subset of icons is available from the Icon choice button.
If you click ... the Icon Browser opens, giving you access to all the graphic icons installed with the CATIA software.

Use the **Grab screen** button to capture an image of the PowerCopy to be stored with its definition in the catalog (see Saving PowerCopies into a Catalog).



Use the **Remove preview** button to delete the image captured with the **Grab screen** button.

4. Click OK to create the PowerCopy.

The PowerCopy is displayed close to the top of the specification tree.

- Double-click the PowerCopy in the specification tree to display the **PowerCopy Definition** dialog box and edit its contents.
- A formula is automatically included in a Power Copy definition when all its parameters are included.
  Otherwise, i.e. if at least one parameter is not selected as part of the Power Copy, you have to manually select the formula to make it part of the definition. If you do so, all the formula's parameters that have not been explicitly selected, are considered as inputs of the Power Copy.

Up Next

Instantiating PowerCopy Features

This task shows how to instantiate PowerCopies once they have been created as described in Creating PowerCopies.
There are three ways to do this:

1. using the contextual menu
2. using the **Insert -> Instantiate From Document** menu item
3. using a catalog

The PowerCopyStart.CATPart document is still open from the previous task, otherwise, 0pen the PowerCopyStartResults1.CATPart document.

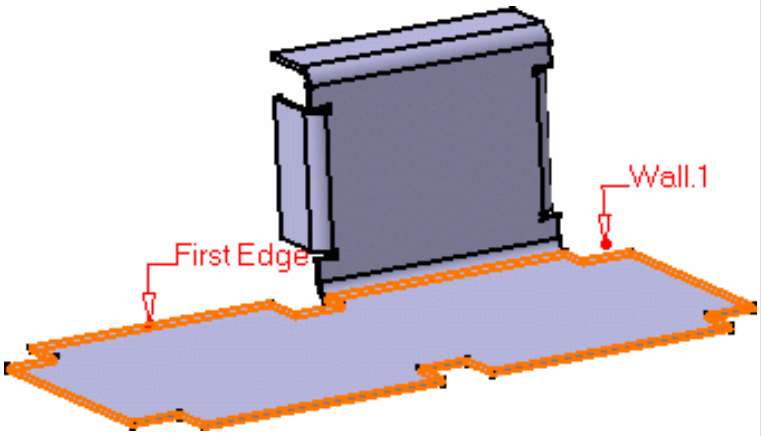
Using the Contextual Menu Item:

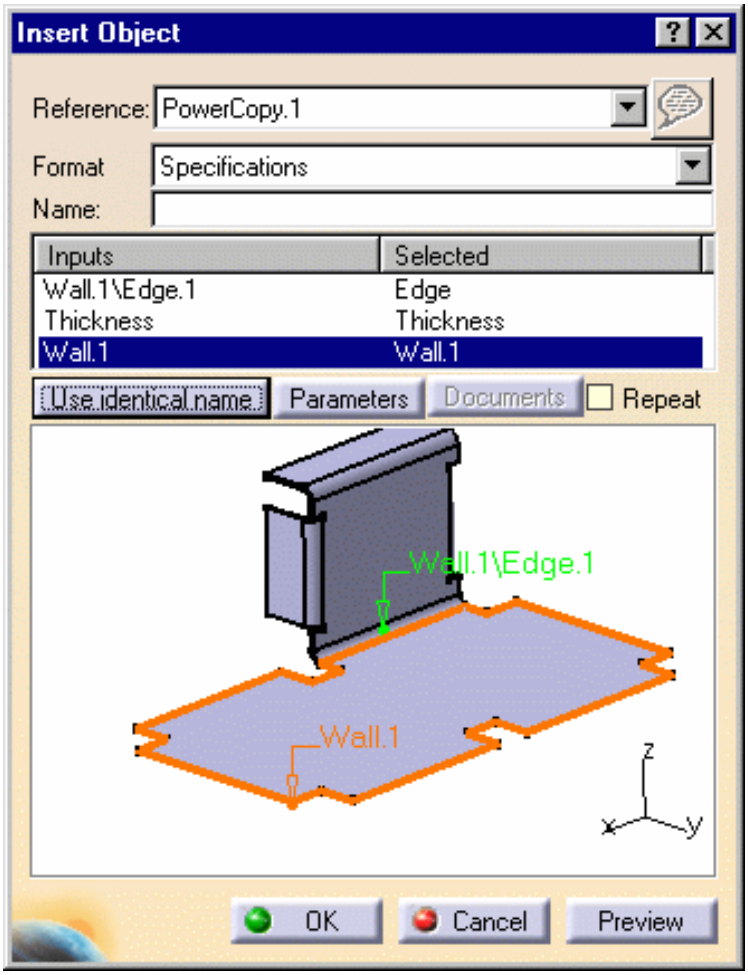
1. Select the PowerCopy feature from the specification tree.
2. Right-click to display the contextual menu, and choose the **PowerCopy.1 object -> instantiate** menu item.

   The **Insert Object** dialog box is displayed.

3. Complete the **Inputs** within the dialog box by selecting the adequate element in the geometric area.

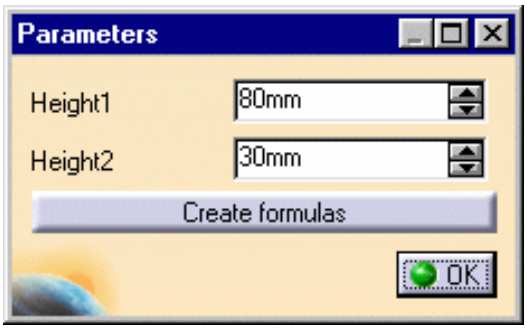
   You need to click the arrow to invert the First Edge orientation.





Check the **Repeat** option to be able to repeat the instantiation.
In this case, once you have clicked OK in the Insert Object dialog box, the latter remains open, the PowerCopy's **Inputs** are listed and ready to be replaced by new inputs, as described above.
Modified parameters using **Parameters** button are retained as well for the next instantiation.
To exit the command, you then need to uncheck the **Repeat** button or click Cancel.

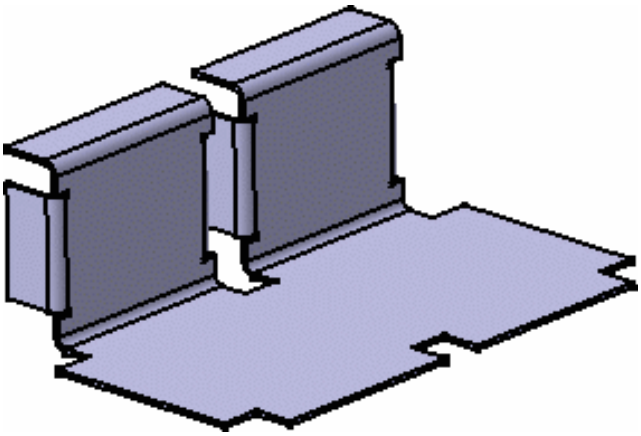
5. You can also click on the **Parameters** button to display the **Parameters** dialog box and modify values, if needed.
6. Use the Create formulas button to automatically create a formula on every parameters with the same name provided there are any.
7. Click OK in the **Parameters** dialog box.



The Documents button lets you access the list of documents (such as design tables) pointed by one of the elements making up the Power copy.
If there are documents, the Documents dialog box opens and you can click the Replace button to display the File Selection dialog box and navigate to a new design table to replace the initial one.
When no document is referenced, the Documents button is grayed within the Insert Object dialog box.

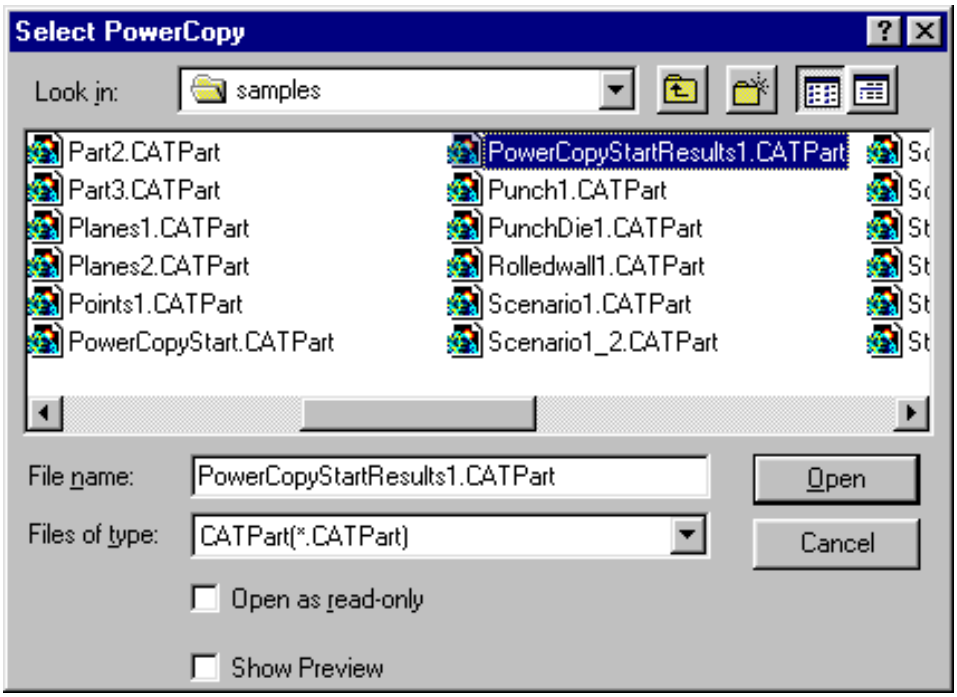
8. Click OK to create the PowerCopy instance.

   The PowerCopy is instantiated in context, meaning its limits are automatically re-defined taking into account the elements on which it is instantiated.



Using the Insert Menu:

1. Select the **Insert -> Instantiate From Document** menu item.
   The **Select PowerCopy** dialog box is displayed allowing you to navigate to the document or catalog where the power copy is stored.



2. Select the document containing the Powercopy.

   The **Insert Object** dialog box is displayed.

   Use the **Reference** list to choose the correct PowerCopy when several have been defined in the document.

3. Continue the instantiation as described in Using the Contextual Menu item, step 3.

Using the catalog:

You need to have a catalog available, created either:

- using the Catalog capability, see the *Infrastructure User's Guide.*
- using the **Insert -> Advanced Replication Tools -> PowerCopy Save In Catalog...** menu item.

1. Click the icon.
   If accessing a catalog for the first time, you need to navigate to the catalog location. This location is stored in the settings for faster access later on.
2. Select the catalog containing the PowerCopy you wish to instantiate.
3. Select the PowerCopy to be instantiated, then you can:

- drag and drop it onto the reference element
- double-click the PowerCopy
- or right-click on the PowerCopy in the dialog box and use the **Instantiate** contextual menu.

From then on, you instantiate the PowerCopy as described Using the Contextual Menu item, step 3.

- You can only instantiate a PowerCopy if the PowerCopy itself and the features making it up have been created in the current view mode: i.e. you will be able to instantiate a PowerCopy created in 3D mode, only on a feature in 3D mode, not on an unfolded feature.
- The  icon is always grayed when instantiating Power Copies. It is available with User Features and allows you to create and modify URLs.



Back Up Next

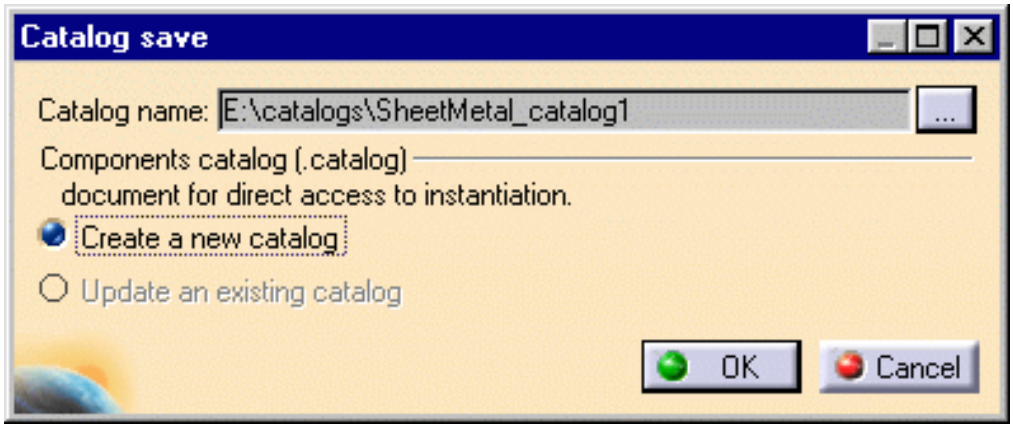
Saving PowerCopy Features

This task shows how to use store Power Copy elements into a catalog, for later use as described in Instantiating a PowerCopy Feature.

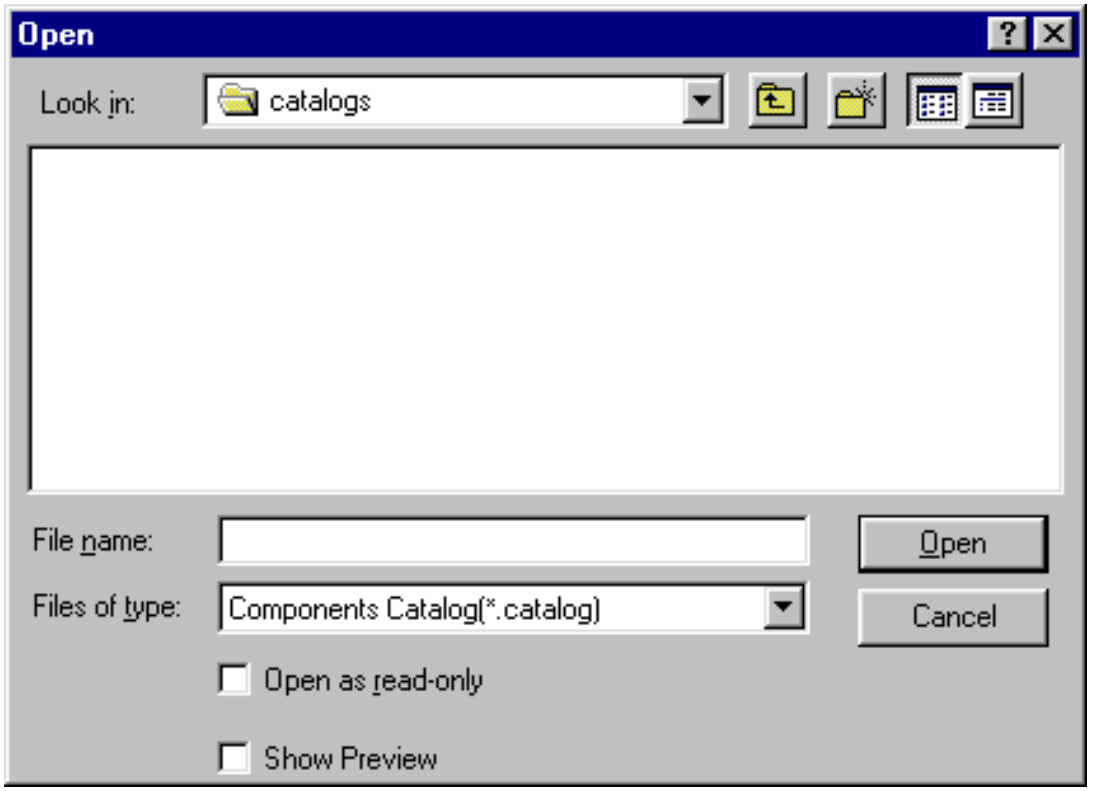
Open the PowerCopyStartResults1.CATPart document.

1. Select the PowerCopy from the specification tree for example.

2. Choose the **Insert -> Advanced Replication Tools -> (PowerCopy) Save In Catalog...** menu item.
   The **Catalog Save** dialog box is displayed:



- 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.

3. Click OK.

The PowerCopy has been stored in the catalog.

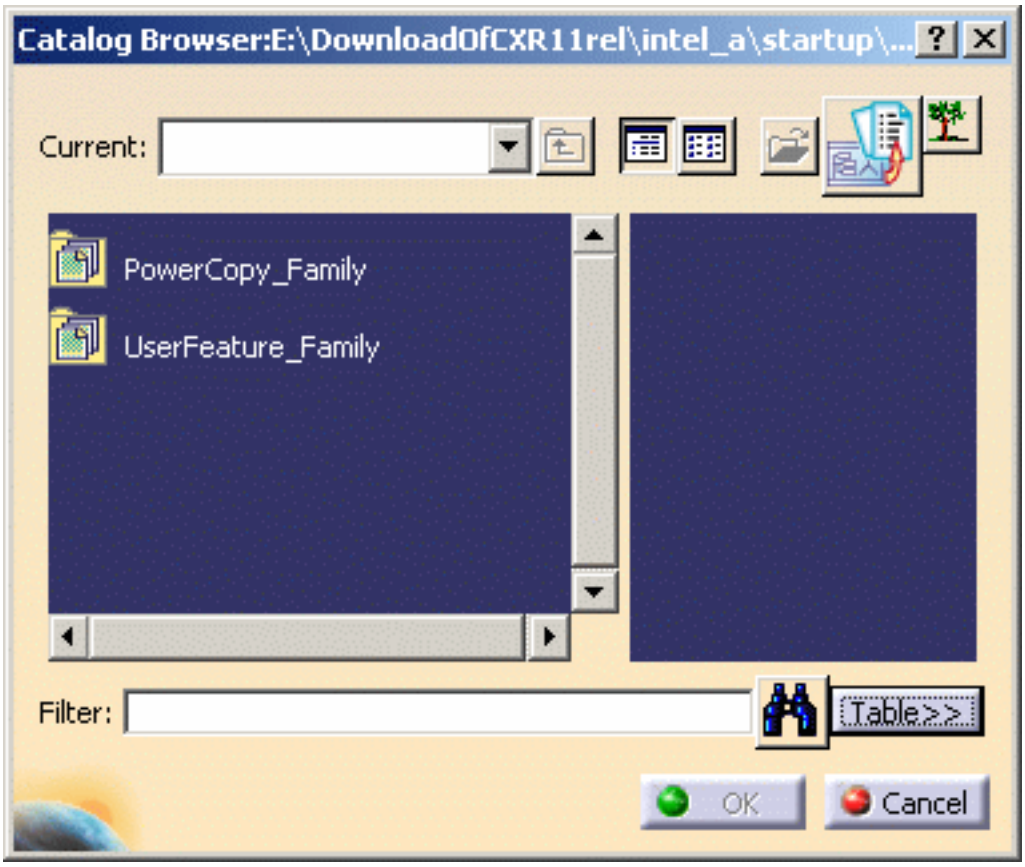


Back Up

Browsing the Sheet Metal Catalog

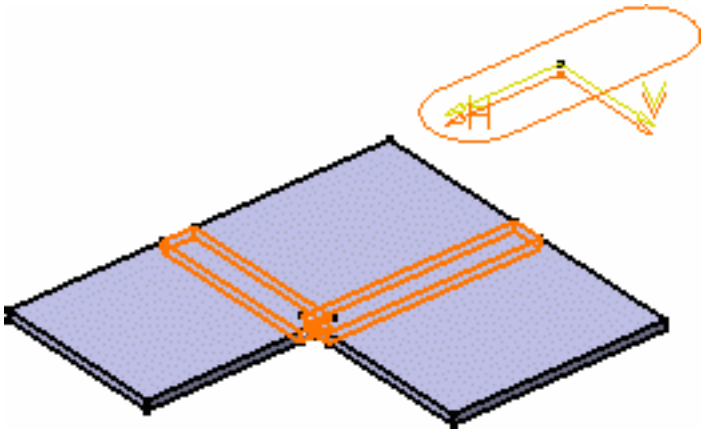
Open the CornerRelief03.CATPart document from the samples directory.

In this case, the Catalog Browser opens, and lets you choose from a number of sketches.

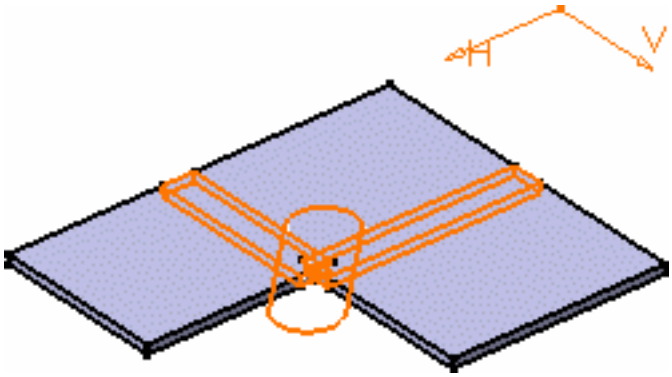


1. Select a sketch and click OK in the Catalog Browser.

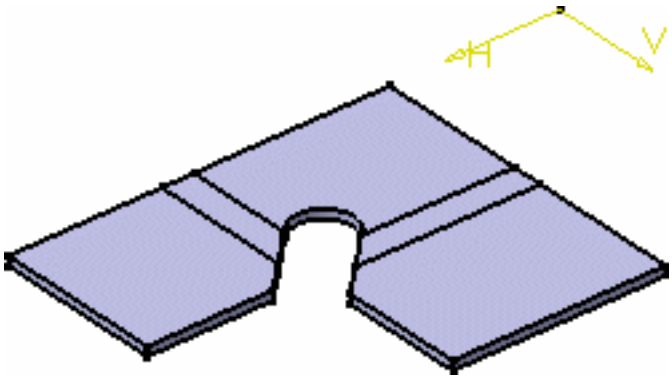
You return to the Corner Relief Definition dialog box, that has been updated with the Sketcher icon.

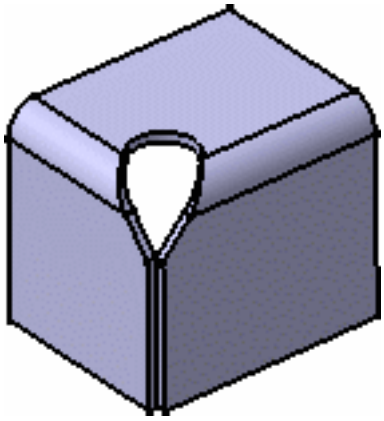


2. Click the sketcher icon, and move the sketch to position it properly in relation to the part.



3. Click OK, and check the part in a folded view.





Back Up Next

Looking For Sheet Metal Features

This task shows how to use the Search capabilities on Sheet Metal Features, in order to detect any specific kind of feature.

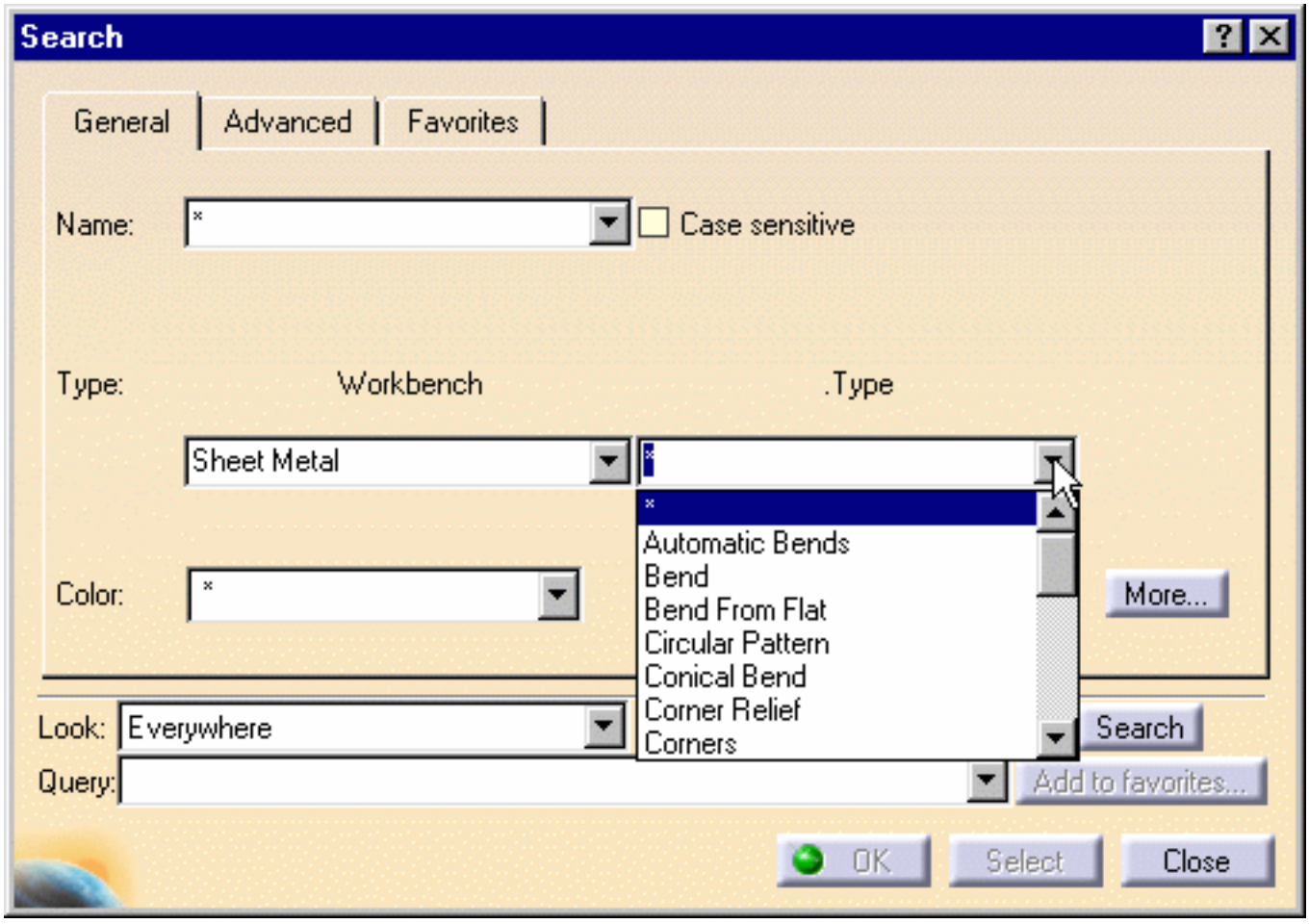
Open the PowerCopyStart.CATPart document.

1. Select the **Edit -> Search** menu item.

   The Search dialog box is displayed.

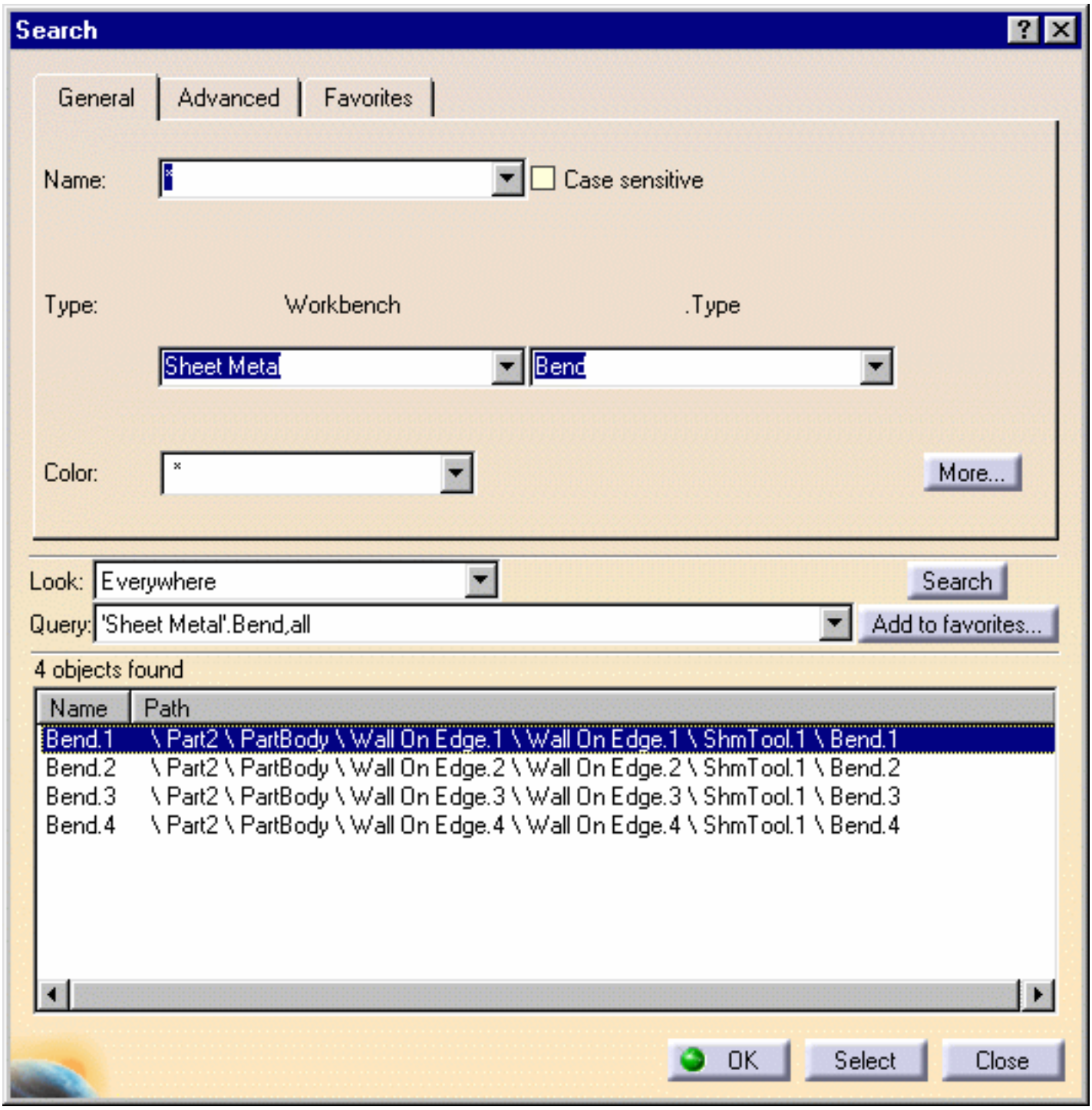
2. From the Type Workbench list choose Sheet Metal.

   You can then display the list of Sheet Metal features from the **Type** list:



3. Select the type of feature you wish to find within the open .CATPart document, and click Search.

   The list of all elements of the selected type is displayed in the Objects found field:



You can select an element from the list, it will be highlighted in the geometry area.

To find out more on the search capabilities, refer to Selecting Using the Search... Command (General Mode) and Selecting Using the Search... Command (Favorites and Advanced Modes) from the *Infrastructure User's Guide*.



Back Up

Working with the New SheetMetal Design Workbench



On the current level, you can access SheetMetal Design, SheetMetal Production, and NEW SheetMetal Design workbenches. However you can only access to the SheetMetal Design and SheetMetal Production functionalities.
To access the NEW SheetMetal Design functionalities, you need to set the following variables:

- SMD_NEW_WORKBENCH_ACCESS set to 1 enables the access to the NEW SheetMetal Design functionalities (SheetMetal Design and SheetMetal Production functionalities are still available),
- SMD_OLD_WORKBENCH_HIDE set to 1 disables the access to both SheetMetal Design and SheetMetal Production functionalities, therefore allowing the access only to the NEW SheetMetal Design functionalities.



This chapter lists the functionalities available with the NEW SheetMetal Design workbench whose behavior or interface slightly differs from the Sheet Metal Design workbench.

When functionalities are strictly identical to the SheetMetal Design workbench, a link to the corresponding CATPart or CATProduct to be opened in the NEW SheetMetal Design workbench is added in the file.

Creating Walls
- Creating Walls from a Sketch
- Creating Tangent Walls
- Creating Walls From An Edge

Extruding

Creating Rolled Walls

Creating Bends on Walls
- Manually Creating Bends from Walls
- Creating Bends From a Line

Aggregating Bodies

Creating Swept Walls
- Creating a Flange
- Creating a Hem
- Creating a Tear Drop
- Creating a Swept Flange

- Redefining Swept Wall Limits
- Display swept walls axes in drafting sheets
- Creating a Hole
- Creating a Cutout
- Stamping
  - Creating Standard Stamping Features
    - Creating a Flanged Hole
    - Creating a Bead
    - Creating a Circular Stamp
    - Creating a Surface Stamp
    - Creating a Bridge
    - Creating a Flanged Cutout
    - Creating a Stiffening Rib
    - Creating a Curve Stamp
    - Creating a Louver
  - Creating User-defined Stamping Features
    - Punch and Die
    - Opening and Cutting faces
    - Editing User-Defined Stamps
- Patterning
  - Creating Rectangular Patterns
  - Creating Circular Patterns
  - Creating User-Defined Patterns
- Creating a Local Corner Relief
- Creating Corners
- Creating Chamfers
- Mapping Elements
- Saving As DXF
- Integration with Part Design
- Integration With Weld Design

Creating Walls

This section explains and illustrates different methods to create walls.


Create a wall from a sketch: use the sketcher to define the profile, and set the material side.


Create a wall tangent to another one: select a profile coincident with an existing wall, and select the wall to which it should be tangent


Create a wall from an edge: select a wall edge, set the height, limits, angle, then the sketch and material sides

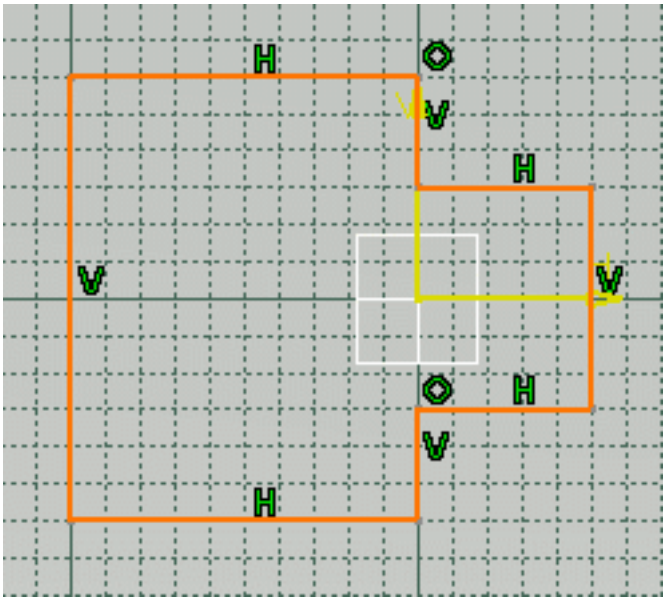
Up Next

Creating Walls from a Sketch

This task shows how to create a wall from a sketch.

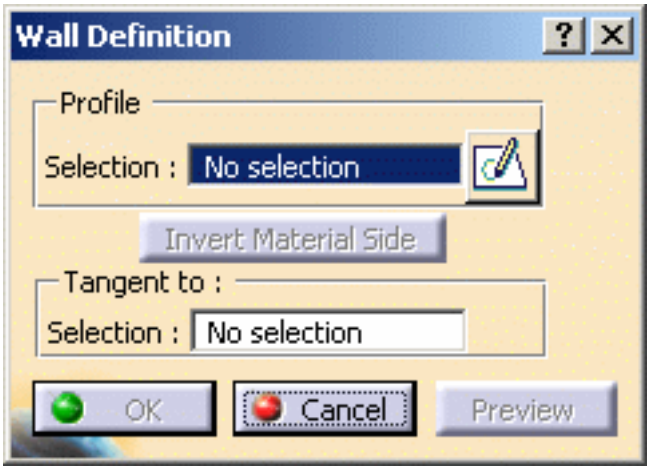
You must be in the Sheet Metal Workbench, with a .CATPart document open, and you must have defined the Sheet Metal parameters.
Set the sketcher grid to **H = 100mm** and **V = 100mm**, using the **Tools -> Options**, **Mechanical Design -> Sketcher**, **Sketcher** tab.

1. Click the **Sketcher** icon then select the **xy plane**.
2. Select the **Profile** icon .
3. Sketch the contour as shown below:



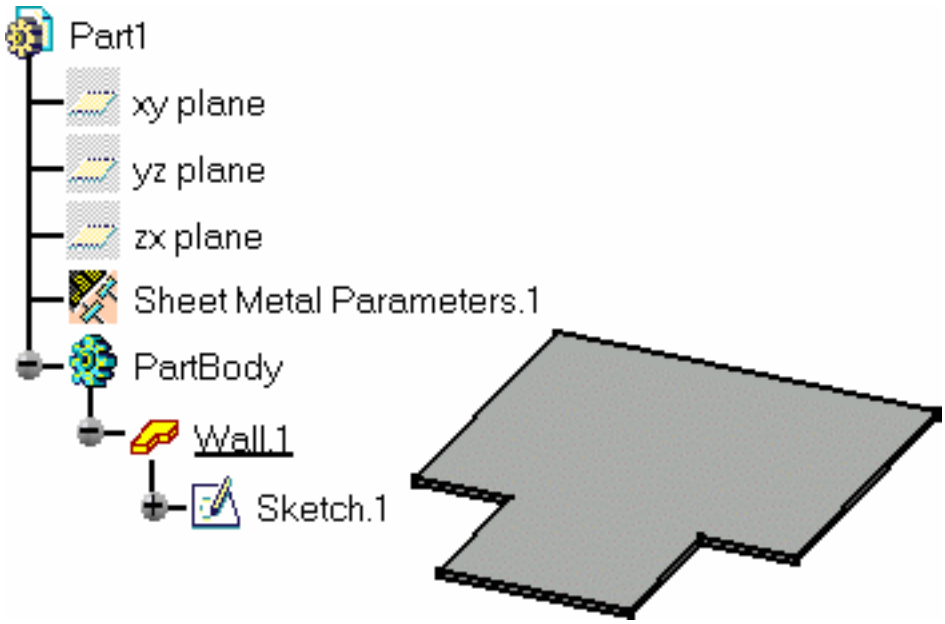
4. Click the **Exit workbench** icon to return to the 3D world.

5. Click the **Wall** icon .
The Wall Definition dialog box opens.



By default, the Material Side is set to the top.

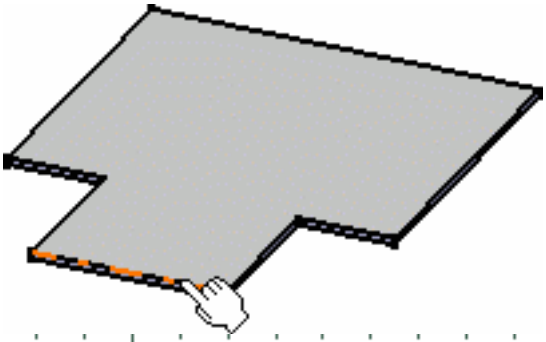
6. Click OK.
The **Wall.1** feature is added in the specification tree.



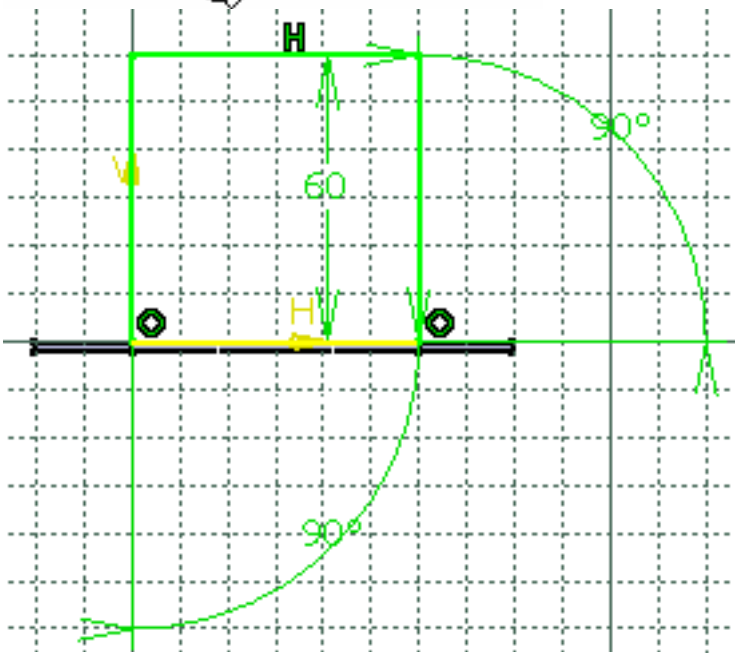
The first wall of the NEW Sheet Metal Part is known as the Reference wall.

- Click the **Sketcher** icon from the Wall Definition dialog box, if you wish to directly edit the selected sketch. When exiting the sketcher, you then go back to the wall creation step, without having to reactivate the **Wall** icon.

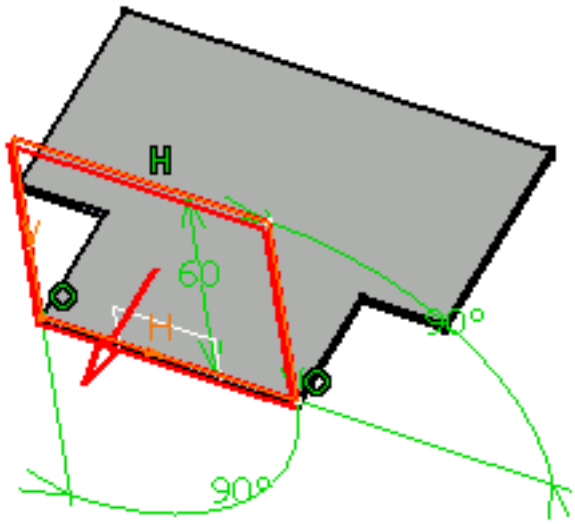
This is also very useful if you have selected an edge from a wall and clicked the **Wall** icon .



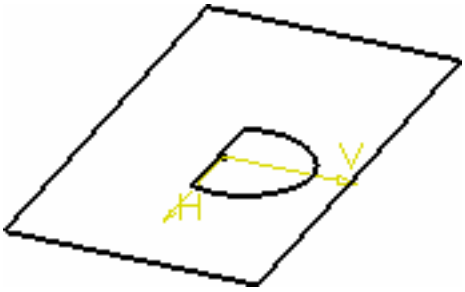
In this case, the sketcher is automatically activated displaying a square sketch constrained to the plane perpendicular to the support of the selected edge.



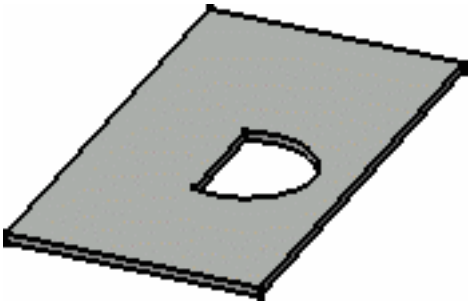
You can then directly draw a sketch, then exit the sketcher and return to the wall creation step.



- You can directly create a wall with a hole, by selecting a sketch with an inner contour (the contours must not intersect):



Sketch with inner contour



Resulting wall

Note however, that the emptied area is part of the wall and is not a separate cutout that can be edited.

Creating Tangent Walls

This task shows how to create a wall tangent to a planar part of another wall or of a flange. This capability will then allow this tangent wall to be seen when unfolding the part, even though there is no bend linking it to its tangent support, provided this support is unfoldable too.

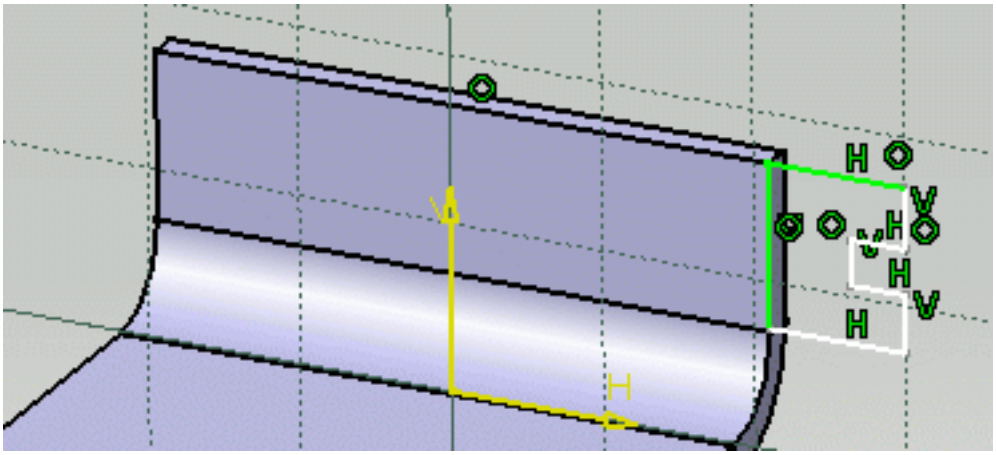
Open the NEWTangentWall1.CATPart document.

1. Select a face of an existing wall and click the Sketcher icon .
   Here we selected the flange's planar face.

2. Select the **Profile** icon and sketch the contour as shown below.

You can also select Sketch.2 from the specification tree.

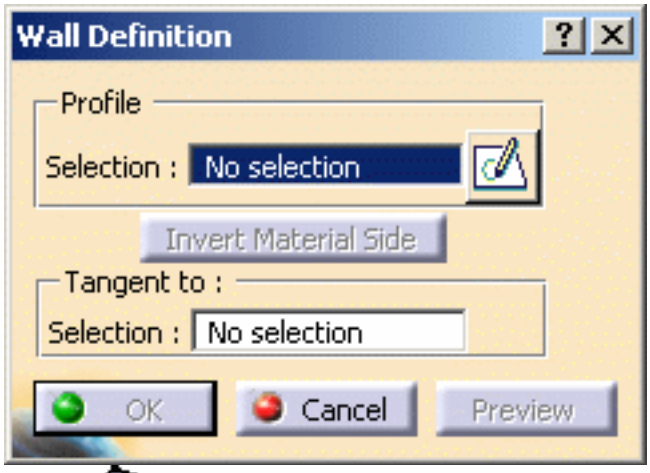
3. Using the Constraint Defined in Dialog box icon, set coincidence constraints between the edges where the support and sketch are to coincide.



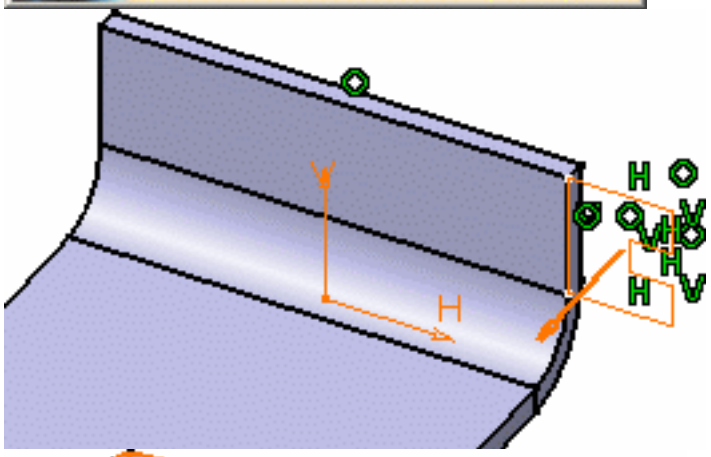
4. Click the **Exit workbench** icon to return to the 3D world.

5. Make sure the sketch is still active, then click the **Wall** icon .

   The Wall Definition dialog box opens.

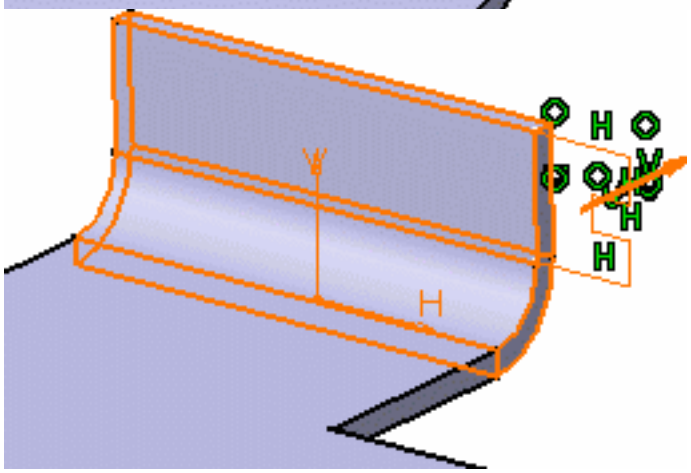


Note the orientation of the wall to be created.



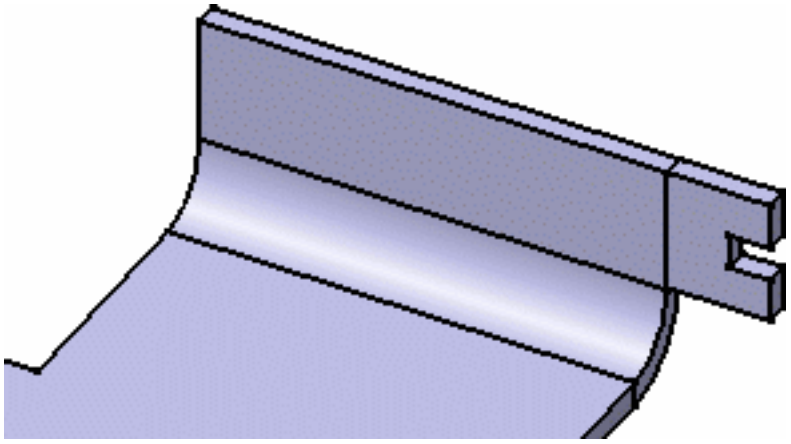
6. Click inside the **Tangent to** field, then select the wall to which the new wall has to be tangent.

   Here, you need to select the planar face of the flange.

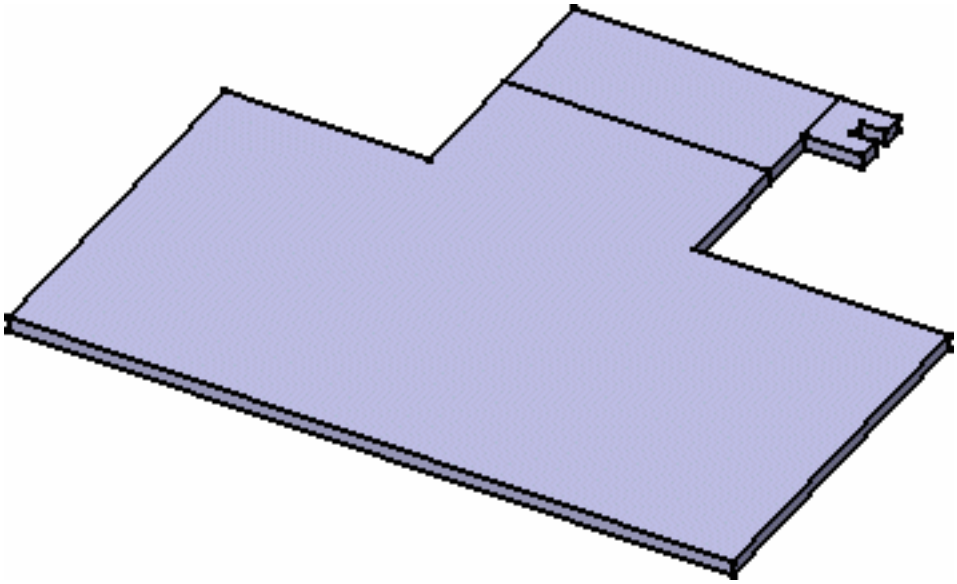


Note that the orientation automatically changes to conform to the material orientation already defined on the support wall.

7. Click **OK**.
   The wall is created and a **Wall.xxx** feature is added in the specification tree.



8. Click the **Unfold** icon .
   The tangent wall is unfolded as a wall linked by a bend to another wall, would be.



If you want to create the wall on a wall that cannot be unfolded, the system issues a warning and prevents you from creating the tangent wall.

Creating Walls From An Edge

This task shows how to create walls from edges of an existing wall. This function is used to create a box in an easy and quick way from an existing reference wall.

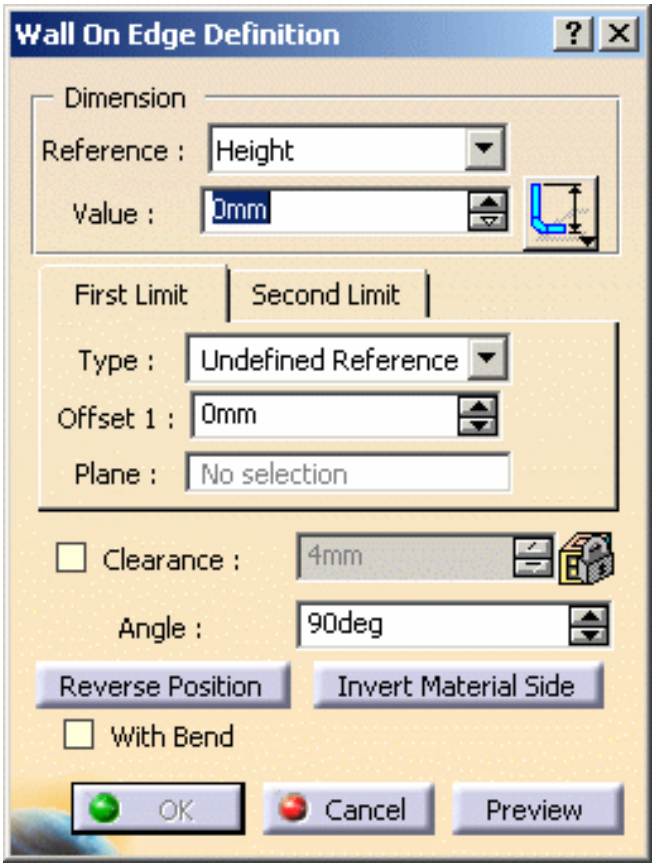
At least one wall must already exist.

Open the NEWWall1.CATPart document from the samples directory.

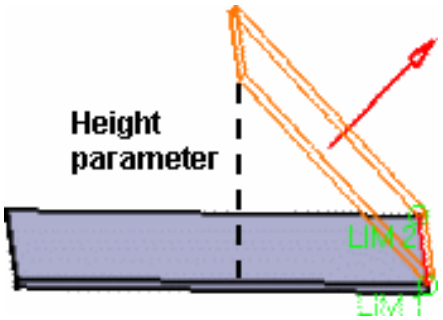
1. Click the **Wall on Edge** icon and select an edge of an existing wall.

   The Wall On Edge Definition dialog box is displayed together with a preview of the wall.

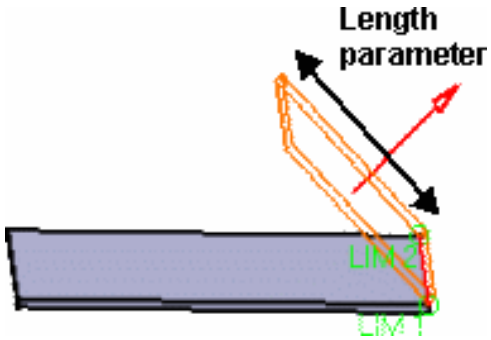
2. Define the type of wall to be created by specifying the Reference, that is:



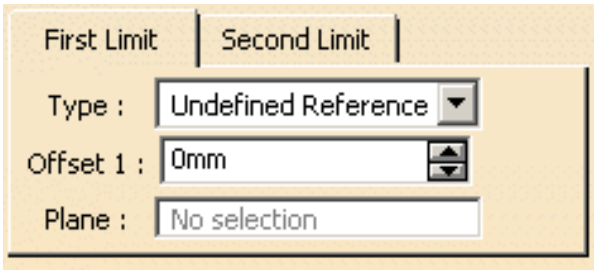
- the **Height** of the wall: that is the orthogonal projection from the top of the wall on edge to the reference wall.
  Select the icon to define the height of the wall from the bottom of the reference wall or the icon to define the height of the wall from the top of the reference wall.



- the **Length** of the wall: that is the absolute value of the wall on edge without bend.



3. You can now relimit the wall by selecting planes for the First and Second Limit:



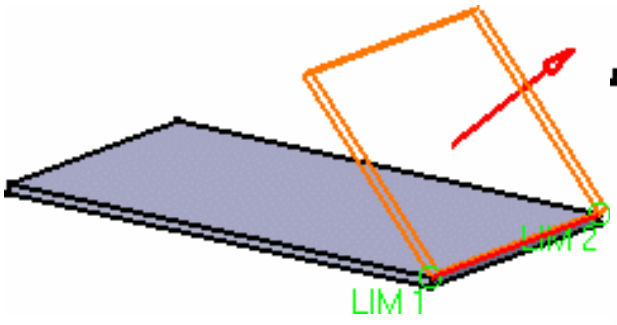
- type:
  - Undefined reference
  - Up To Plane
- Offset
- Plane

As a consequence, the generated wall is not necessarily rectangular.

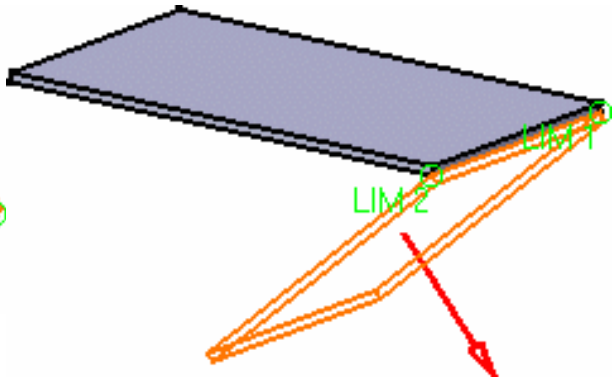
4. Define the angle of the wall: by default it is perpendicular to the plane containing the edge. You may modify it according to your needs. it is updated dynamically on the screen.

This preview gives information about:

- the Sketch Profile: by default, if you pick an edge on the top of the reference wall, the direction of the wall is upwards, if you pick an edge on the bottom of the reference wall the direction of the wall is downwards.

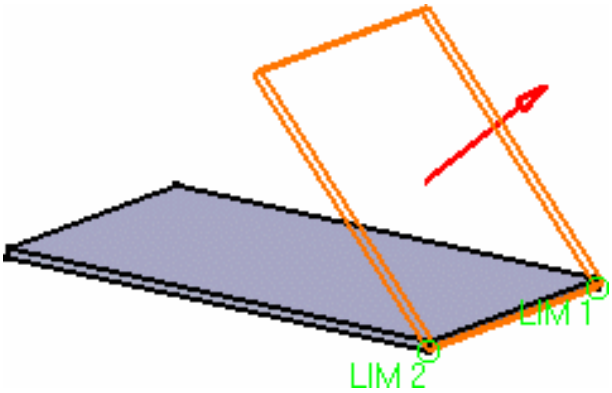


Preview with top edge selected



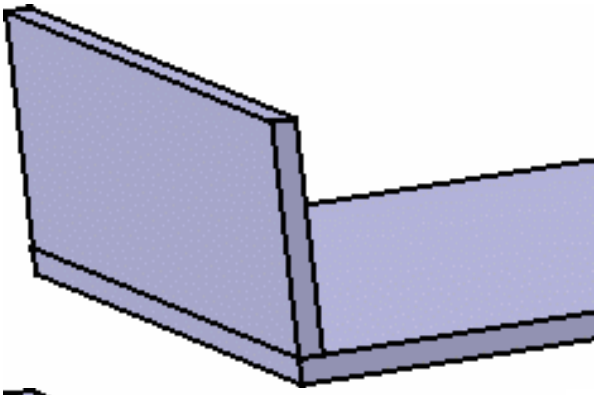
Preview with bottom edge selected

You can invert the sketch's position, and therefore the wall's, using the **Reverse Position** button.

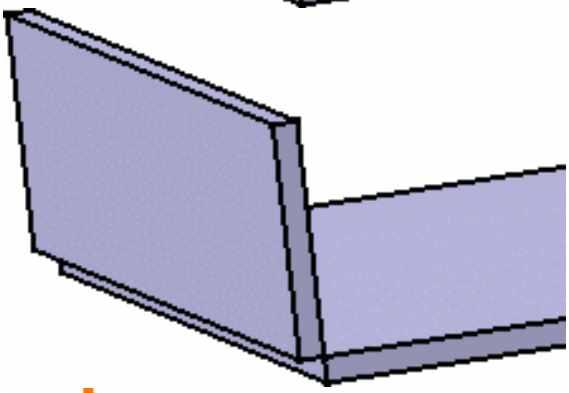


Preview with bottom edge selected and sketch profile inverted

- the material side: by default the material side is proposed so as to keep a continuity with the reference profile.

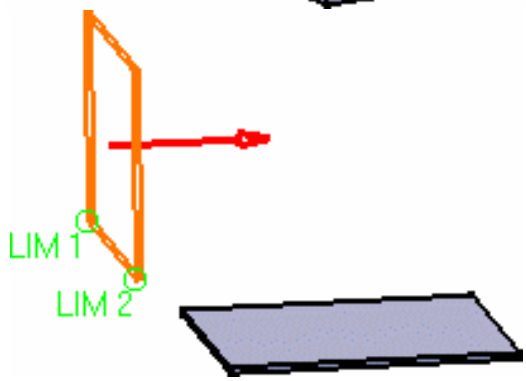


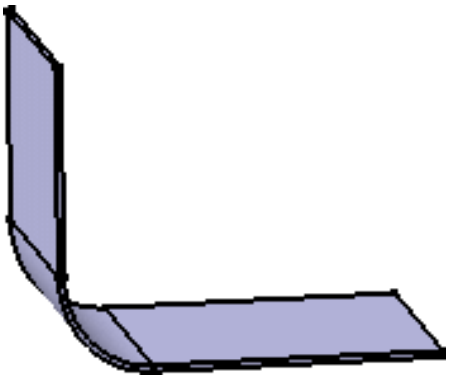
However, you can change it by clicking the red arrow or the **Reverse Side** button.



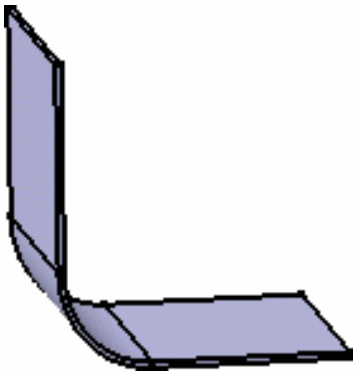
5. Check the **Clearance** option to offset the wall on edge from the selected edge.

   The entered value is the radius of the bend on this edge.





Wall on edge with clearance



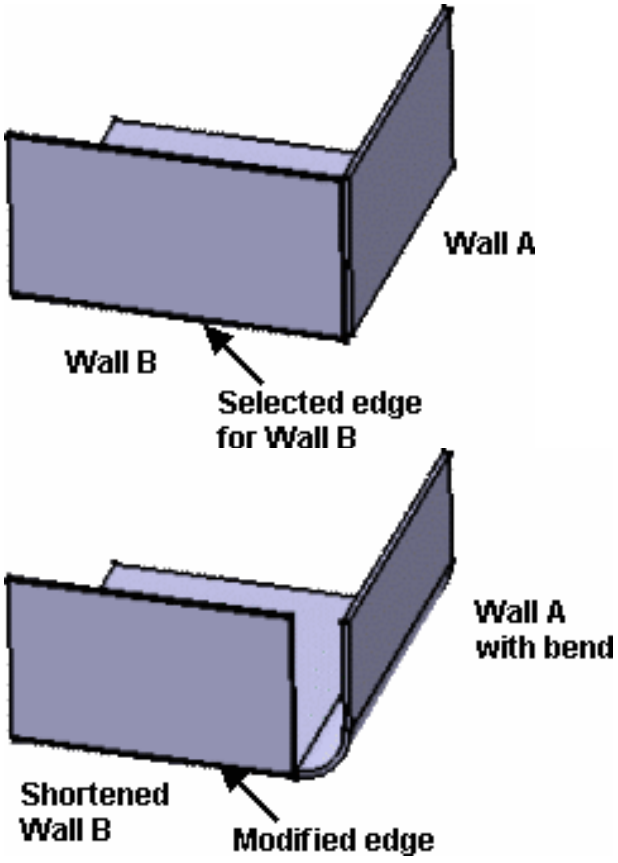
Wall on edge without clearance

Note that the reference wall remains unchanged when changing the bend radius value. It is the Wall on Edge's length that is affected.

6. You can also choose to create the wall with or without a bend by checking the **With Bend** option.

- If there is no wall from which a limit can be computed, the reference element is the edge of the reference wall.

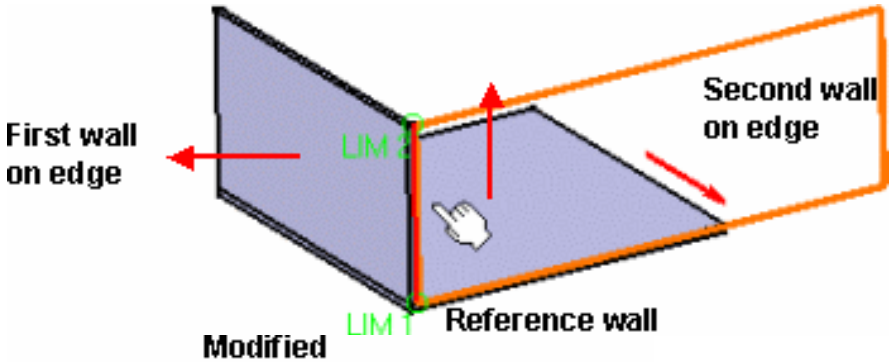
- A wall on edge is defined by the selected edge (reference edge). When the reference edge is modified, by adding any feature that shortens the edge (a bend to an adjacent wall on edge or a cutout for example) the wall on edge based on this reference edge is recomputed.
  To avoid this you may:
  - create the bend manually on the wall modifying the edge used as the reference to create the other wall
  - reorder the creation of walls to postpone the creation of the modifying feature



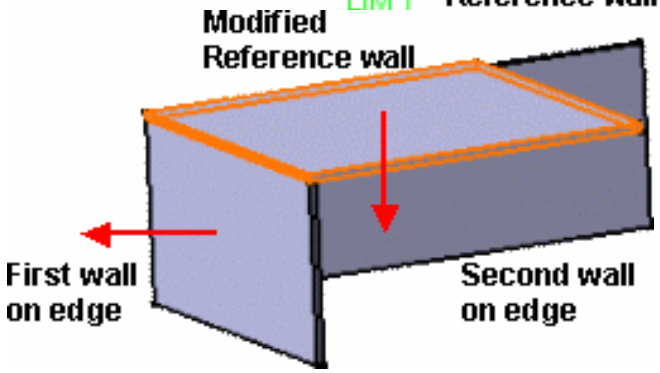
- Both limits are computed with the same reference icon.
- The bend is not previewed, even if the option **With Bend** is checked. However it will be created.
- The selected options are modal and will be proposed to create the next wall.

- Walls on edge being contextual features, if you break the profiles continuity by inverting the material side of a wall, you may have to manually re-specify all features based upon the modified one, even if they are not directly connected to the modified wall, in order to update the part afterwards.

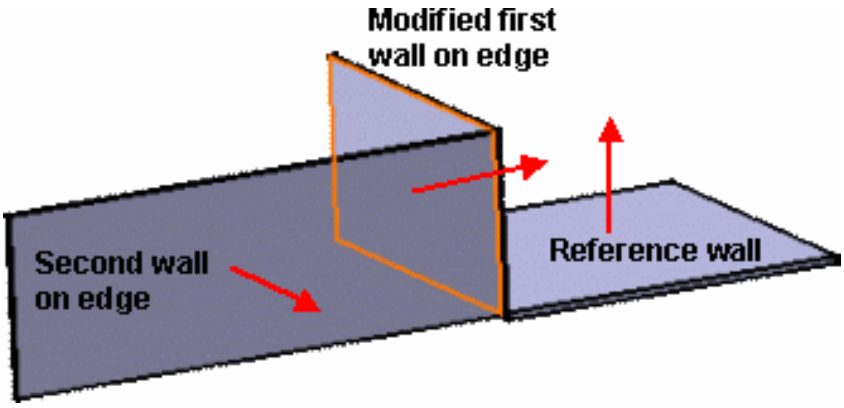
  This is the case when creating a wall on edge from another wall on edge, for example:



- If you invert the material side of the reference wall on which the first wall on edge has been created, both walls on edge are relocated (as if you were flipping the geometry):



- If you invert the material side of the first wall on edge (and not the material side of the reference wall), the second wall on edge is relocated. Indeed, its specification being relative to the first wall on edge, when its input data (i.e. the edge selected on the first wall on edge) is modified, the second wall on edge is rebuilt at a new location.



7. Click OK in the Wall On Edge Definition dialog box.

   A WallOnEdge.xxx element is created in the specification tree.

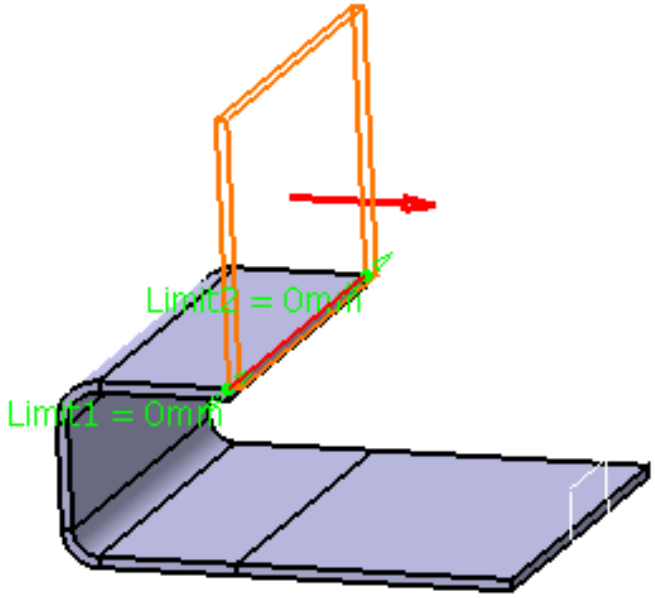
- A WallOnEdge.x element can be edited: double-click it in the graphic area or in the specification tree to display its creation dialog box and modify the parameters described above, including the edge from which it is created. However, the sketch of a wall on edge cannot be edited directly.
- Would you need to edit the sketch of a wall on edge, you have to isolate it first. See Isolating Walls.
- You can cut or copy and paste a wall on edge.

- If you cut and paste a wall on edge with children elements, these children elements are lost. This may result in update errors.
- You cannot undo an **Isolate** action after having modified the wall.
- Isolating a wall on edge erases all updating data.

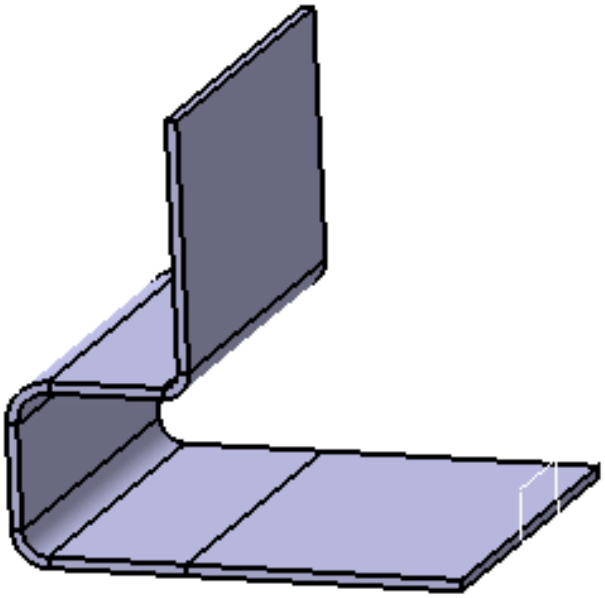
Creating a Wall On Edge on a flange or a stamp

Open the NEWWall3.CATPart document.
Let's take an example with a user flange.

1. Click the **Wall on Edge** icon and select the edge of the user flange.



2. Click OK in the Wall on Edge Definition dialog box to create the wall on edge on the user flange.



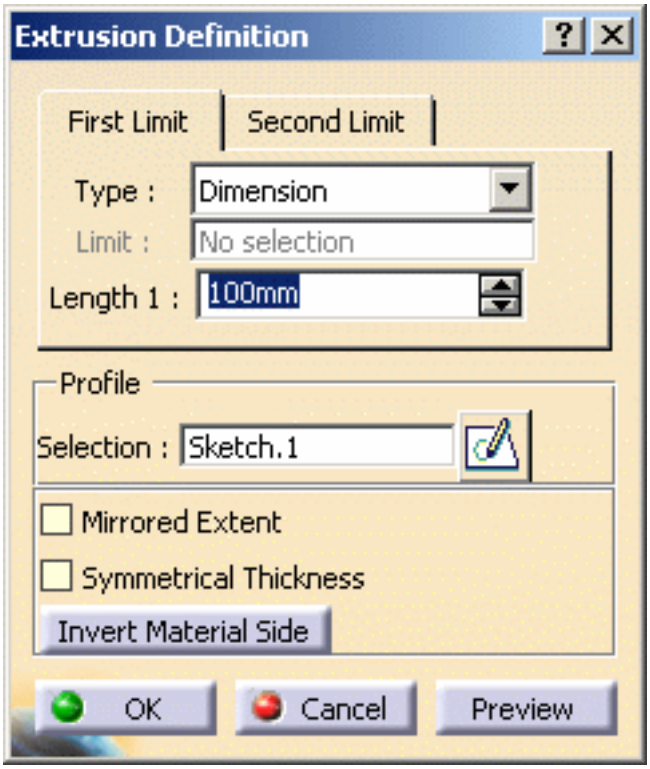
Extruding

This task explains how to create a wall by extrusion.

You must have defined the Sheet Metal parameters.
A model is available in the NEWExtrude1.CATPart from the samples directory.

1. Click the **Extrusion** icon .
2. Select a sketch.

   The Extrusion Definition dialog box is displayed.

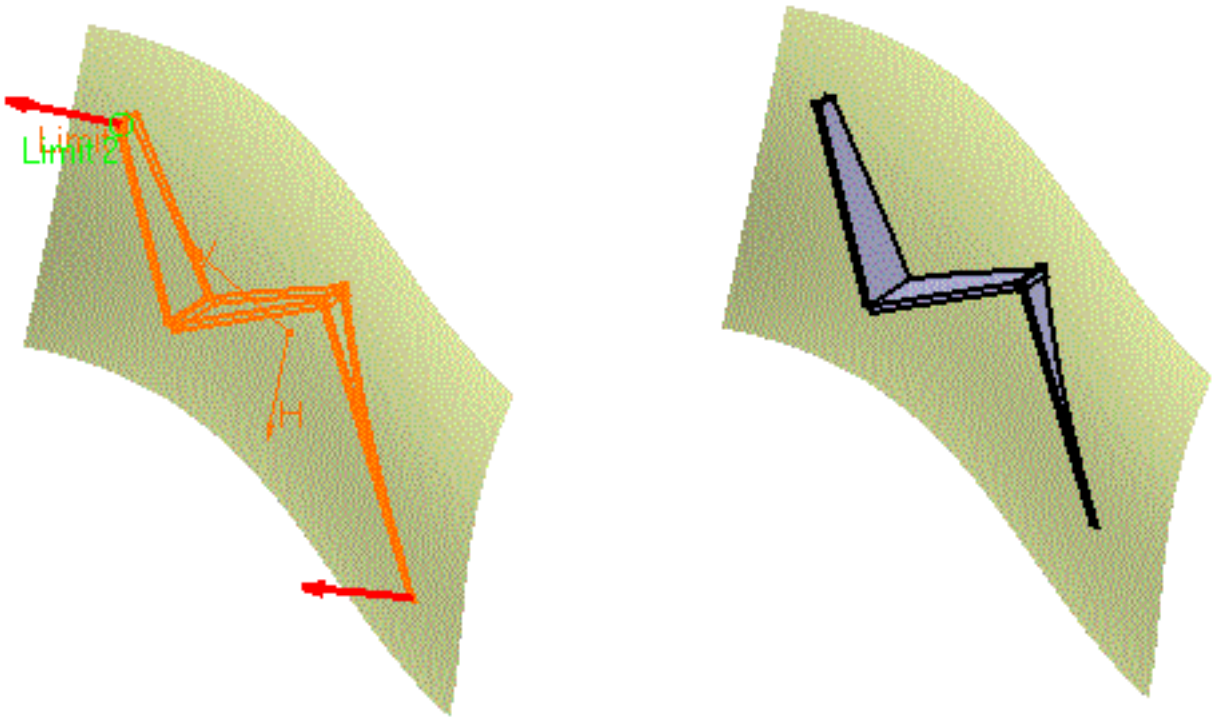


Several types of extrusion are available:

- **Dimension** : the requested input data are a sketch and a dimension,
- **Up to plane** or **Up to surface**: a plane or a surface are input as limit to the extrusion. These functions are used to create walls that are not rectangular.

3. Edit the **Length1** and **Length2** to set both extremities, for option **Dimension**.

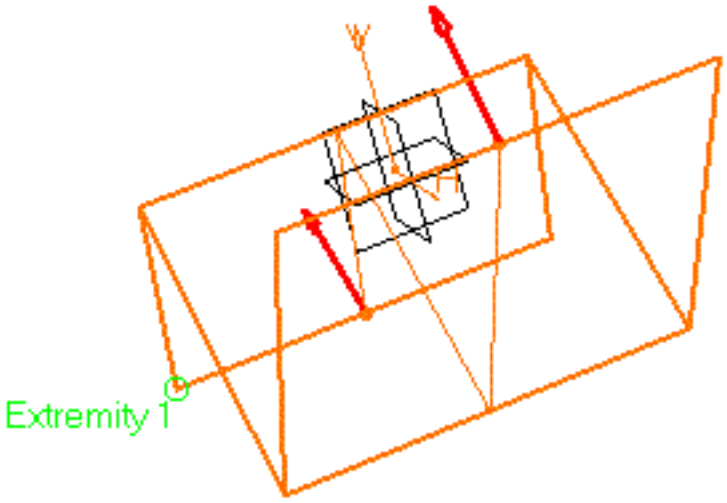
By default, the **Length1** value is positive.



The sketch you selected appears in the **Selection** field
You can now edit it by clicking the **Sketcher** icon  if you wish to modify it.

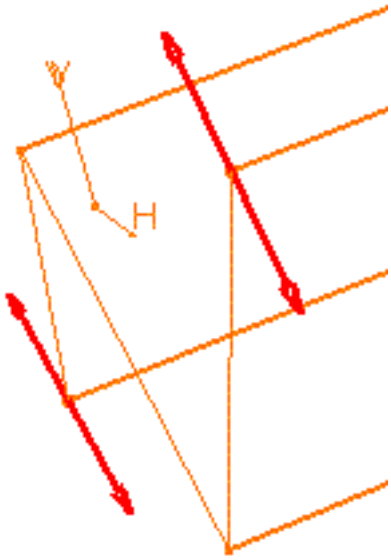
4. Define the options as needed:

- Check the option **Mirrored extent** to extent the material on both sides of the sketch. In that case, only **Length1** can be edited.

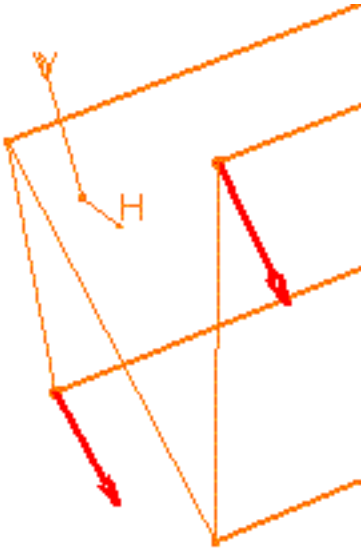


This option is only available if the type is set to **Dimension**.

- Check **Symmetrical thickness** to create thickness inside and outside the sketch.



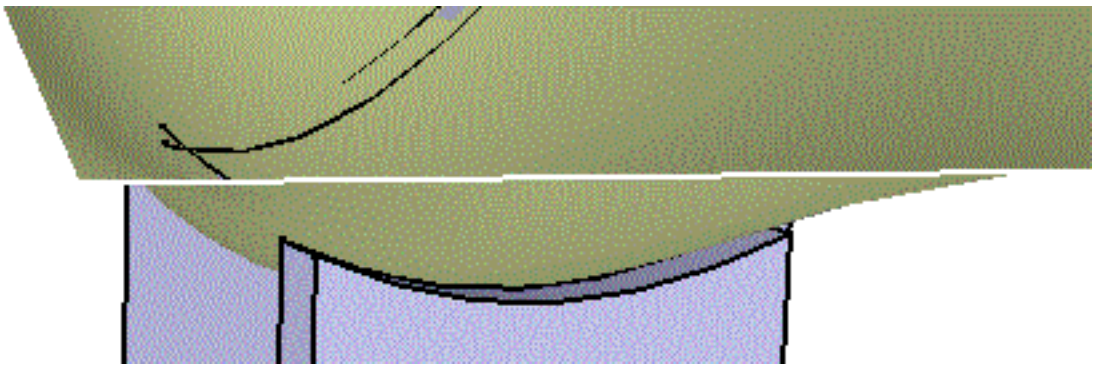
- Push **Invert Material Side** to invert the direction of the creation of the material.



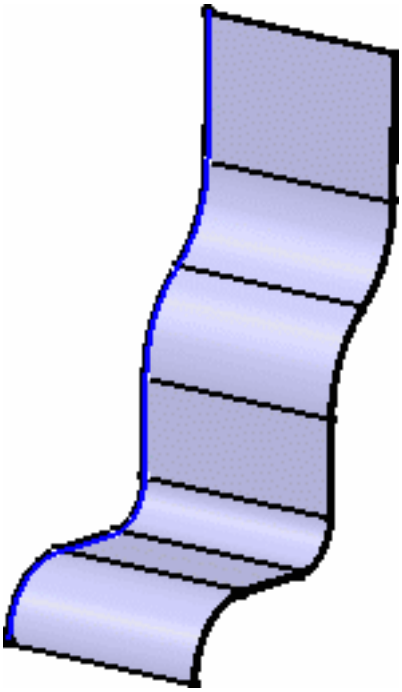
5. Click OK.

   The walls corresponding to the selected sketch are created according to the specified options, and added to the specification tree.

- When the extrusion is the first Sheet Metal feature of the Part, the reference wall is the first wall created based on the first segment of the sketch.
- For option **Up to Surface**, while the wall end that is limited by the surface has the shape of the surface, its thickness does not fit the surface. It is a "rectangular" polygon defined by the first edge that comes into contact with the surface.



- Such an extrusion can also be performed on a sketch made of lines and arcs of circle, provided there are no tangency discontinuities between the different elements.
  However, in this case:
  - the **Up to plane** or **Up to surface** capabilities are not available,
  - you cannot isolate such an extrusion,
  - if the element of the extruded sketch connected to the part is an arc of circle, the extrusion will not display in the unfolded view. To avoid this, prefer to create a User Defined Flange or remove the arc of circle of the extruded sketch and create a bend to connect the extrusion to the part.



- Extrusion walls can be edited and/or isolated.

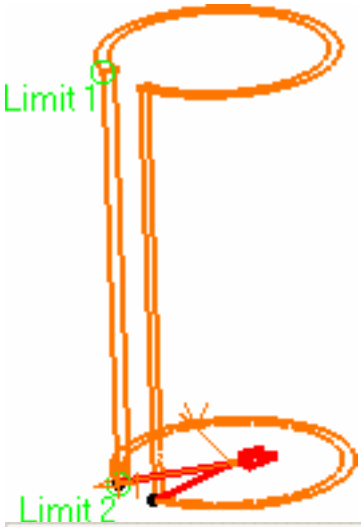
The sketch may not be closed, and must contain at least a line.

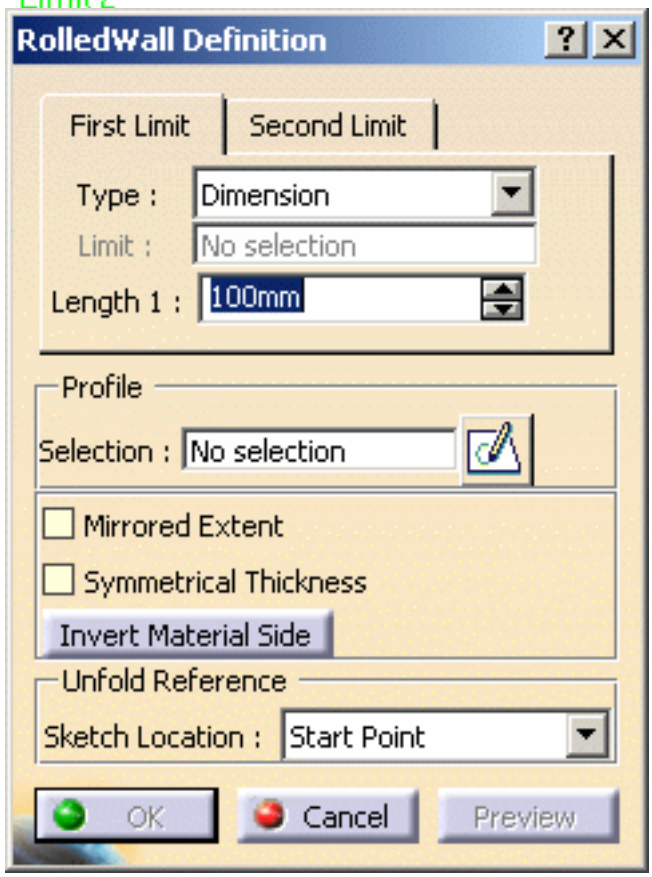
Creating Rolled Walls

This task shows how to create rolled walls (such as pipes, open pipes with flange, etc.).

You must have defined the Sheet Metal parameters, and have a sketch available, in the form of an circular arc.
The part is available in the Rolledwall1.CATPart from the samples directory.

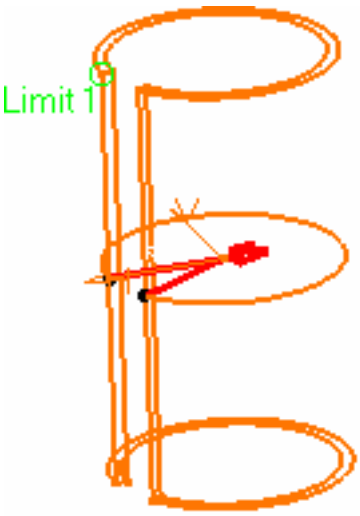
1. Click the **Rolled Walls** icon .
2. Select the circular sketch.



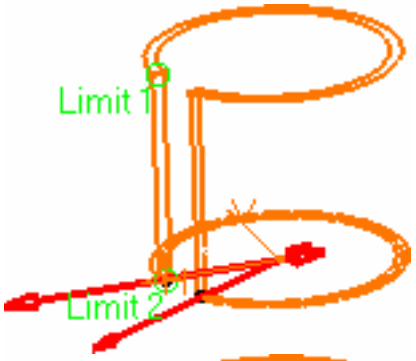


3. Make sure the type is set to **Dimension**.
4. **Length 1** and **Length 2** indicate the location of **Limit 1** and **Limit 2**.
5. Define the options as needed (the length being down to -50 mm):

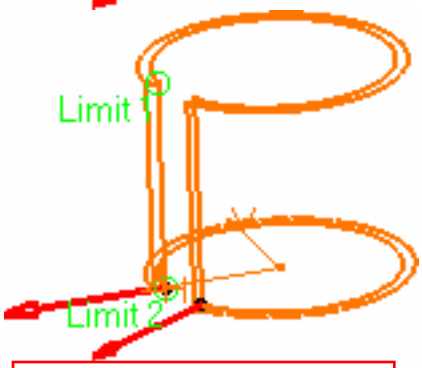
- Check the option **Mirrored extent** to extend the material on both sides of the sketch. In that case, only **Length1** can be edited.



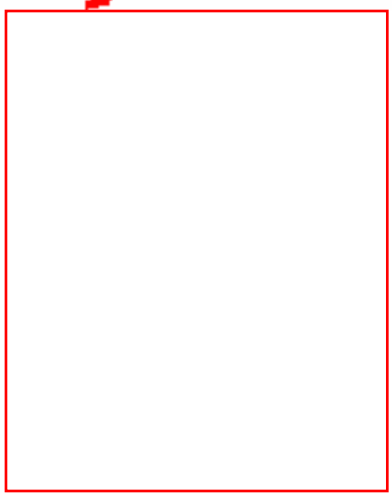
- Check **Symmetrical thickness** to create thickness inside and outside the sketch.



- Push **Invert Material Side** to invert the direction of the creation of the material.



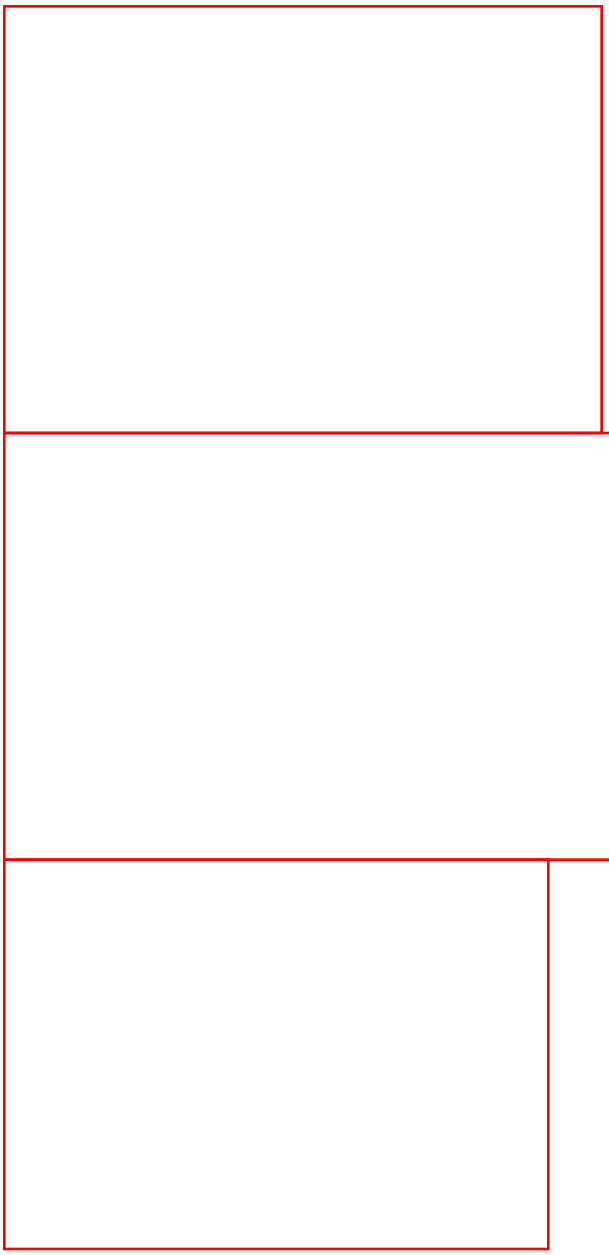
6. Click OK in the dialog box to create the rolled wall.



Unfold Reference

Two symmetrical faces are generated so that the rolled wall can be unfolded. To do so, click the unfold icon .

You can unfold the rolled wall using three different references:



- **Start point**

- **End point**

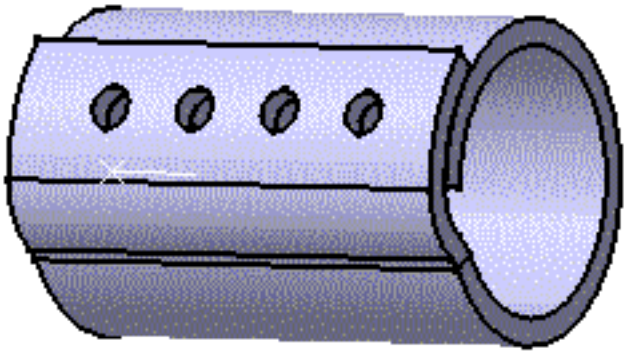
- **Middle point**

The rolled wall is a particular extrusion, indeed the input sketch is either a circular arc or a closed circle.

The sketch may be open. In that case, you can define where the opening should be. The opening angle must be at least 0.5 degree.

The sketch may be closed. In that case, you have no control on the opening location.

The only operations you can combine with a rolled wall in a Sheet Metal model are flanges and cutouts.



No other elements (standard wall, bend,...) are allowed.

Creating Bends on Walls

This section explains and illustrates different methods to create bends on walls. Bends can only be created between walls and not between any other Sheet Metal features, such as stamps for example.


Create bends from wall: select the two walls, set the bend radius value, the bend extremities, and specify the use of corner relief


Create flat bends: select a sketch, set the creation mode and limiting option, set the radius and angle in relation to the selected sketch.

Back Up Next

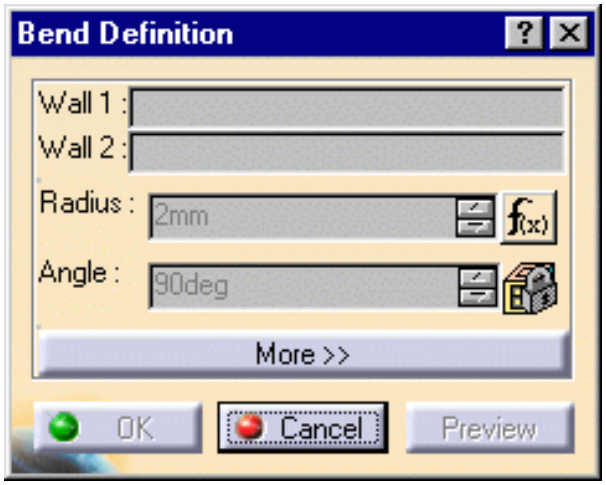
Manually Creating Bends from Walls

This task explains two ways to create bends between walls in the Sheet Metal part. These bends can be created on non-connex walls, and with a constant radius value.

Open the NEWBendExtremities01.CATPart document.

1. Select the **Bend** icon .

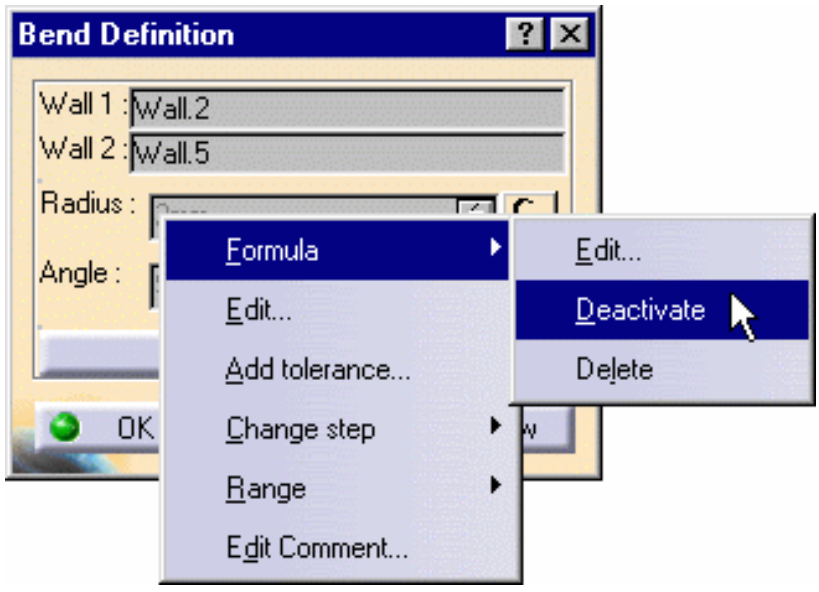
   The Bend Definition dialog box opens.



Note that the Radius field is in gray because it is driven by a formula when editing the sheetmetal parameters: at that time, you cannot modify the value.

2. Select **Wall.2** and **Wall.5** in the specification tree.
   The Bend Definition dialog box is updated.

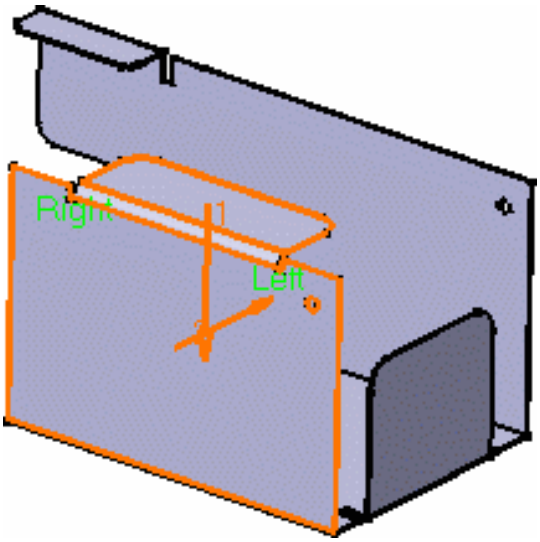
3. Right-click the **Radius** field and select **Formula -> Deactivate** from the contextual menu to change the value.



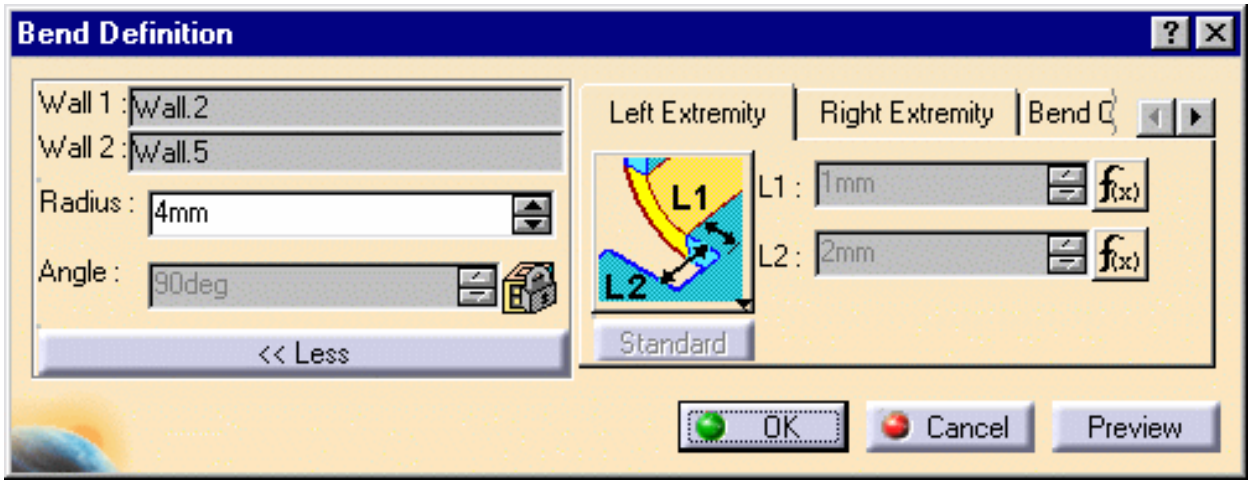
You can set the **Radius** value to 0.

4. Enter 4mm for the Radius and click Preview.

   The bend is previewed, along with its orientation symbolized by arrows. The Left and Right texts further indicate this orientation and are useful to define different bend extremities.



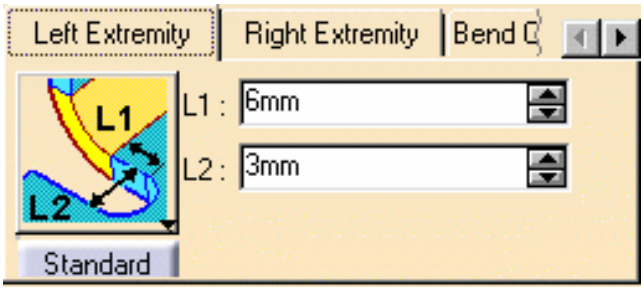
5. Click the **More>>** button to display further options:



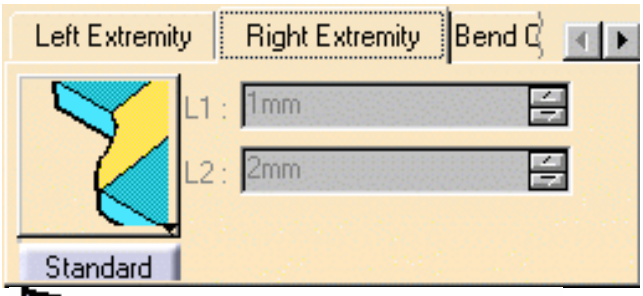
You can define:

- the left and right extremity settings (see also extremities definition settings)
- the corner relief definition settings
- and the bend allowance settings.

6. Within the Left Extremity tab, choose the Mini with round relief bend extremity type, deactivate the L1 and L2 length formulas, and set them to 6mm and 3mm respectively.

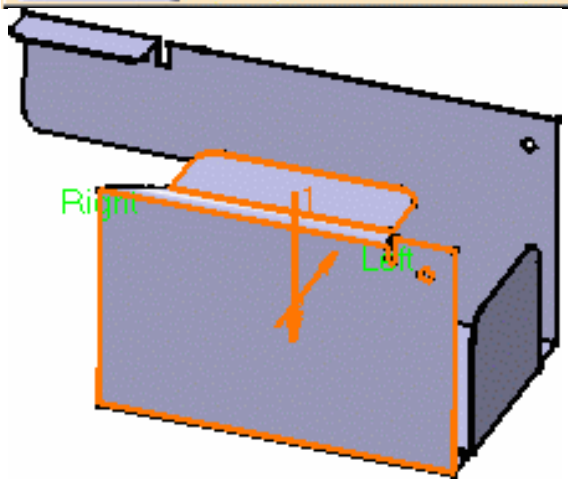


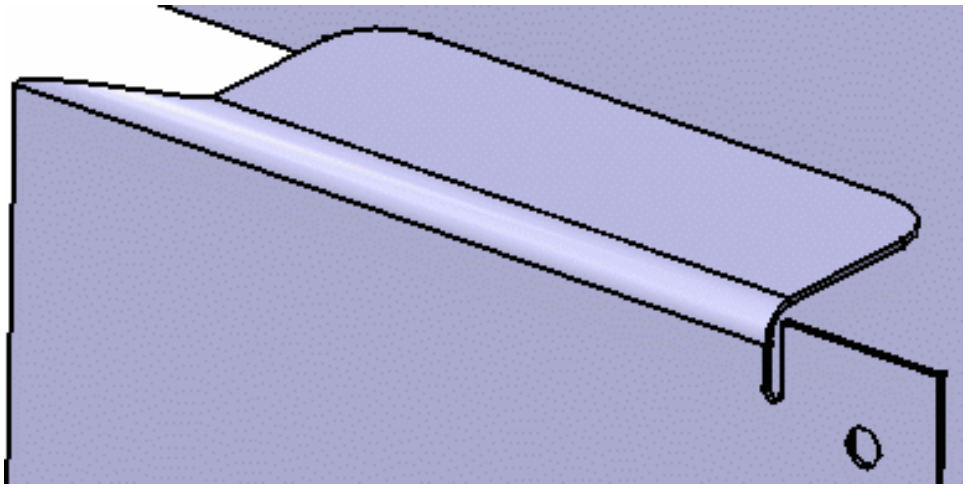
7. Click the Right Extremity tab, and choose the Curved shape type.



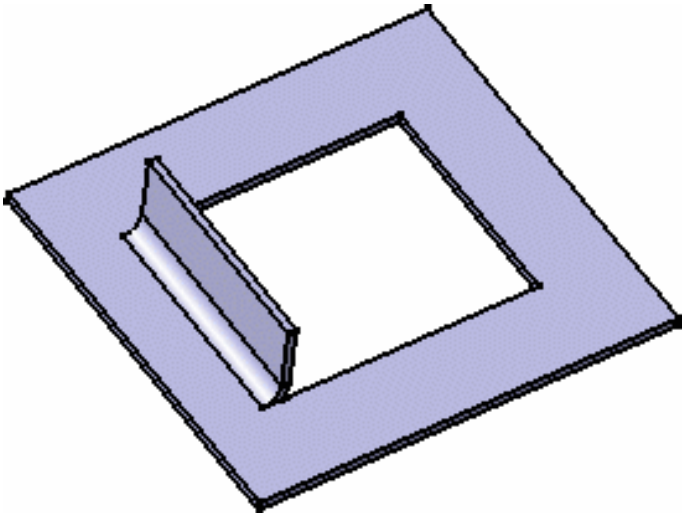
8. Click Preview to visualize the left and right extremities.
9. Click OK in the Bend Definition dialog box.

   The Bend (identified as Cylindrical Bend) is created with the specified extremity types and is added to the specification tree.

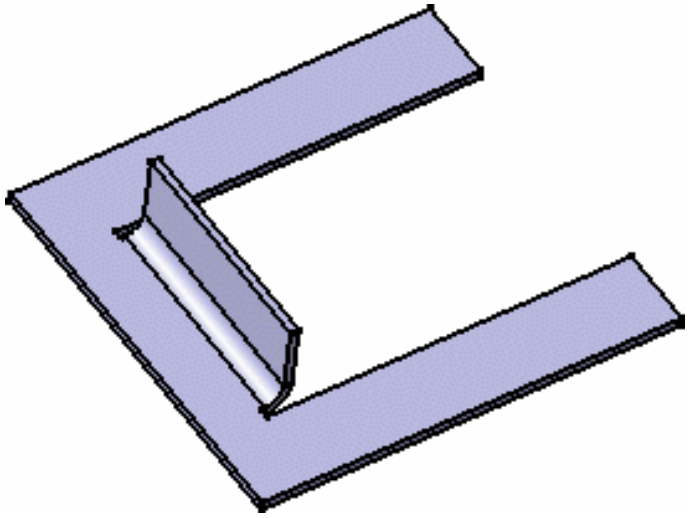




- The extremities and the corner relief defined with the current dialog box will apply locally and prevail over any other global definition.
- Be careful when creating bends with **square** or **round** relief. Depending on the geometry configuration, this can lead to removing more matter than you would expect. Indeed, a corner relief being computed on the whole intersection of the elements involved (bends or bend/wall), in the following configuration, the matter is removed till the end of the wall.

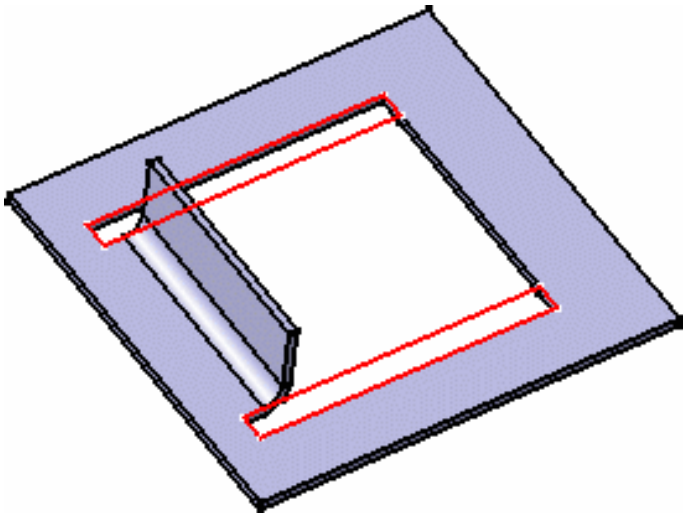


Bend with no relief



Bend with square relief

In this case, it is best to replace the bend corner relieves with cutouts (identified in red in the figure below) that will act as corner relieves removing matter to the part.



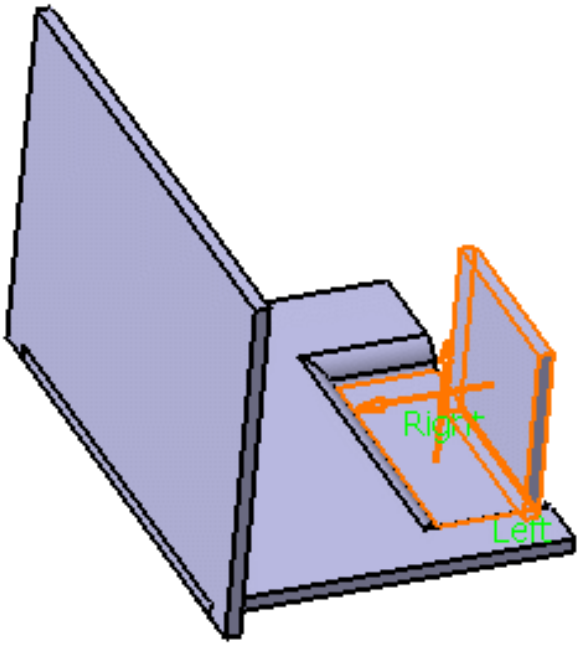
Creating a Bend on a flange or a stamp

Open the NEWBendExtremeties02.CATPart document.

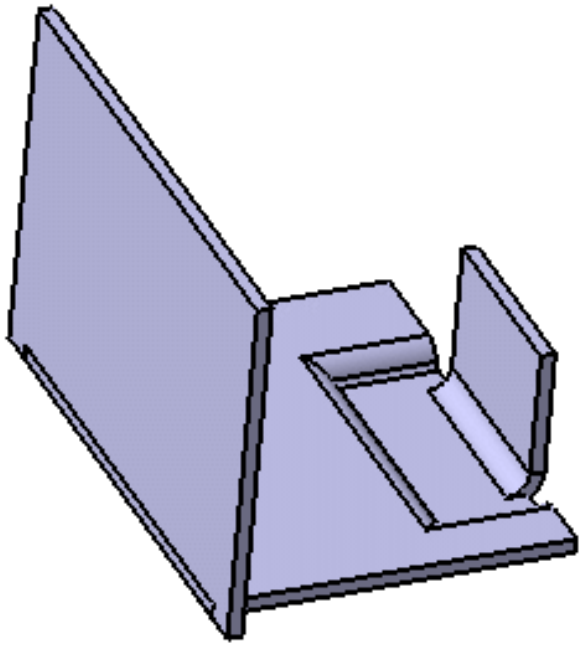
Let's take an example with a surface stamp.

1. Select the **Bend** icon .

   The Bend Definition dialog box opens.
2. Select the surface stamp's face as **Wall 1**.
3. Select Wall.3 in the specification tree as **Wall.2**.



3. Click OK in the Bend Definition dialog box to create the bend on the surface stamp.



To deselect a wall, simply click on another wall available in the part.

The stamp must be open, contain a planar face, and have a radius higher than 0.

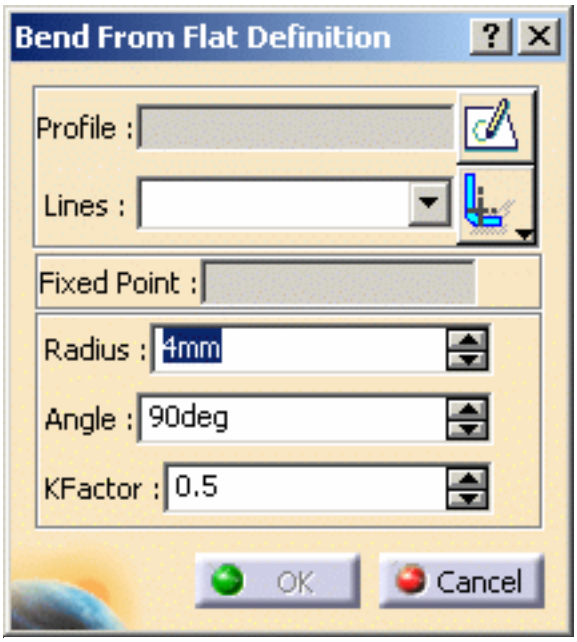
Creating Bends From a Line

This task explains how to generate bends based on a line (also called flat bends). A wall and a bend are created.

Open the NEWFlatBend1.CATPart document.

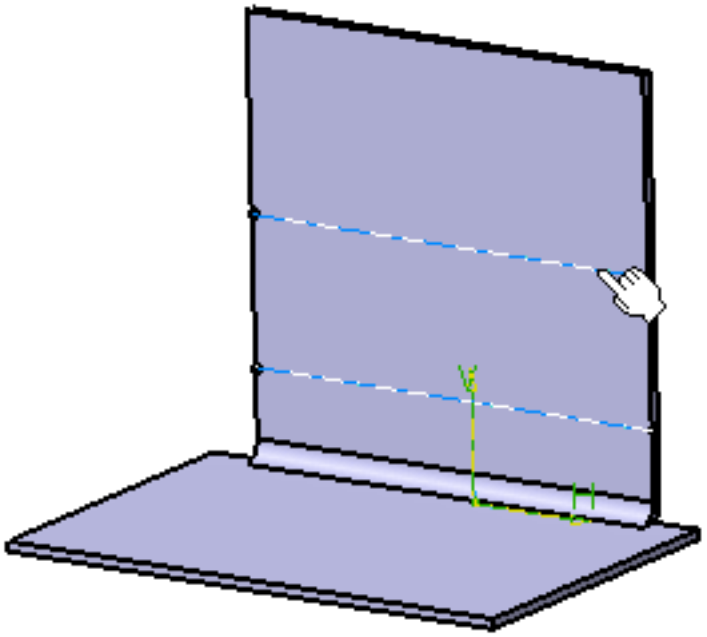
1. Select the reference wall then the **Bend From Flat** icon .

   The Bend From Flat Definition dialog box opens.



2. Select a profile (Sketch.2 here) containing one or several lines. This sketch must necessarily be a line.

   Selected lines appear in the Lines combo list.



3. You can choose the line extrapolation option:

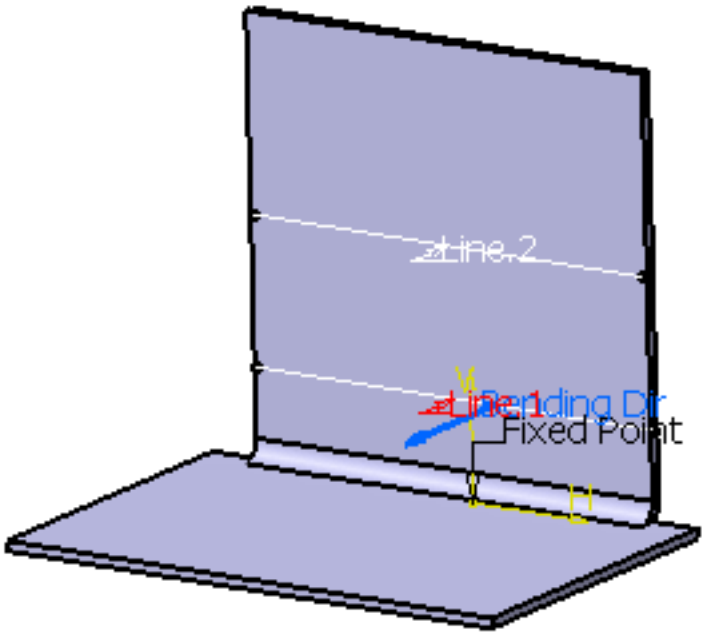
 Axis

 BTL (Bent Tangent Line): line corresponding to the limits of the bend's fillet

 IML (Inner Mold Line): line created by intersecting the internal surfaces of the bend (before filleting) and the wall

 OML (Outer Mold Line): line created by intersecting the bend support and a plane perpendicular to the wall and normal to the OML.

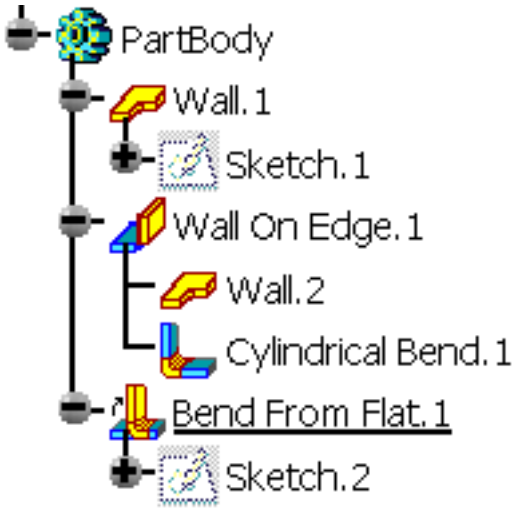
The Radius and the KFactor values are the one defined when editing the sheetmetal parameters:
Right-click the **Radius** or the **KFactor** field and select **Formula -> Deactivate** from the contextual menu to change the value.

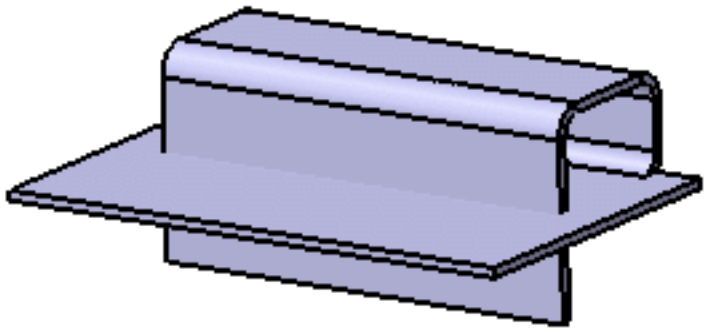


You can set the **Radius** value to 0.

4. Set the angle value between the generated wall and the reference wall.

5. Click OK to create the bend.





- When creating such a bend on a reference wall (first wall), an arrow indicates which part is to be folded. Click this arrow to invert the side that will be bent.
- Bends from line should be performed on end walls, or prior to creating further walls on the bent one.
- Perform the bend before creating the stamping features, as stamps are not retained when the part is folded with the bend.



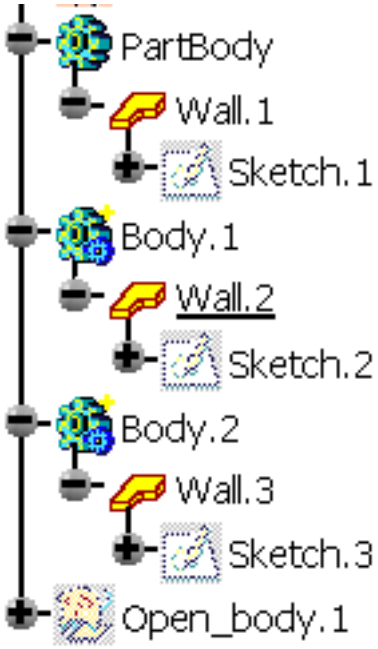
Aggregating Bodies

This task shows you, when creating a bend, to aggregate, under the current body containing a wall, the created bend as well as the body containing the second wall.

Open the NEWBendExtremities03.CATPart document.

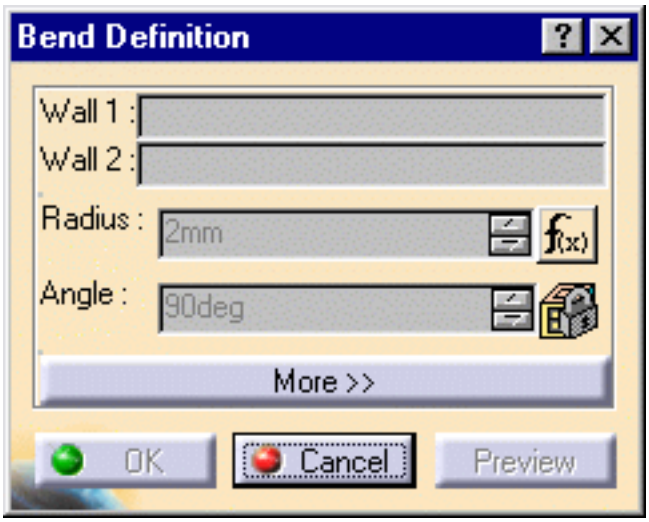
This capability is available with the Bend and Bend From Flat functionalities.
Let's take an example with the Bend.

1. In the specification tree, define **Wall.2** (in Body.1) as current: right-click and select **Define In Work Object** from the contextual menu.

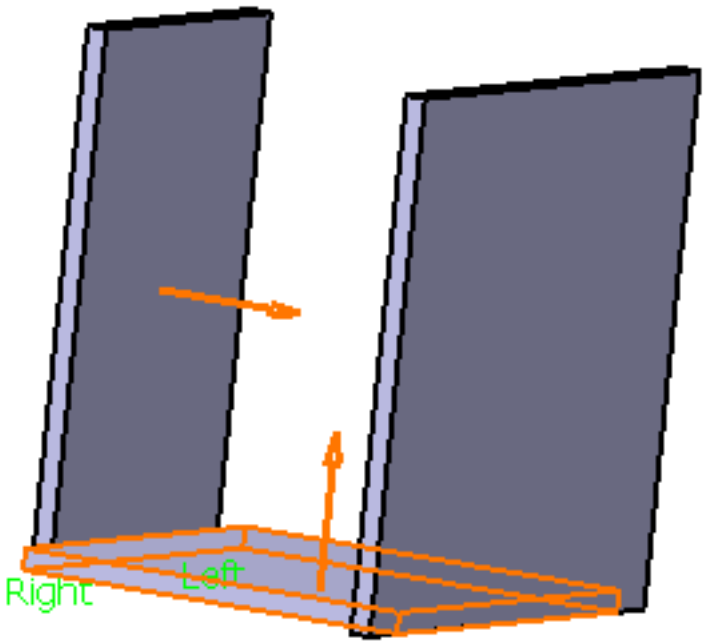


2. Select the **Bend** icon .

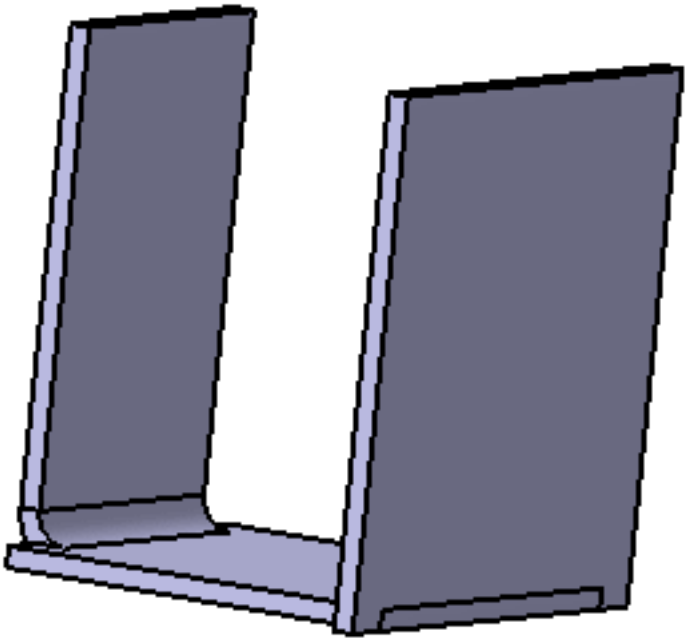
   The Bend Definition dialog box opens.



3. Select **Wall.2** and **Wall.3** in the specification tree.
   The Bend Definition dialog box is updated.

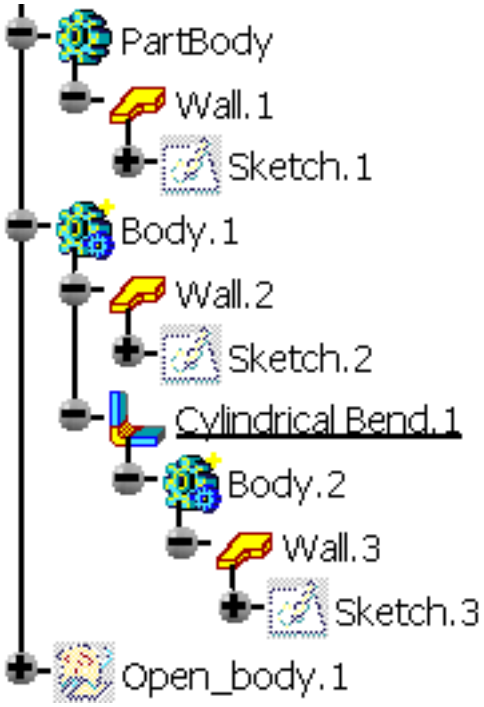


4. Click **Preview** to visualize the bend.
5. Click OK in the Bend Definition dialog box.



The Bend (identified as Cylindrical Bend.x) is created and added to the specification tree.

You can see that it appears under the Body.1 node and aggregates Body.2 containing Wall.3.



You can perform the same operation by defining PartBody as the current object and selecting Wall.1 and Wall.2 as inputs. PartBody contains the cylindrical bend, that aggregates Body.1 (itself containing Wall.2).

Creating Swept Walls

This section explains and illustrates how to create and use various kinds of swept walls, i.e. walls based on a given contour that is swept along a spine.


Create a flange: select a spine, and set the radius, length, and angle values.


Create a hem: select a spine, and set the radius, and length values.


Create a tear drop: select a spine, and set the radius, and length values.


Create a swept flange: select a spine, and a user-defined profile

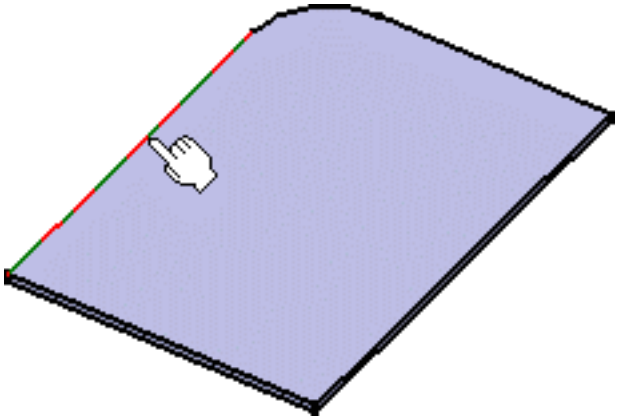
Redefine swept walls limits: choose the **Relimited** type, and select a point lying on the spine or a plane normal to the spine and intersecting it as limits


Display swept walls axes in drafting sheets: open a CATDrawing document, click the **Unfolded View** icon, and select a plane

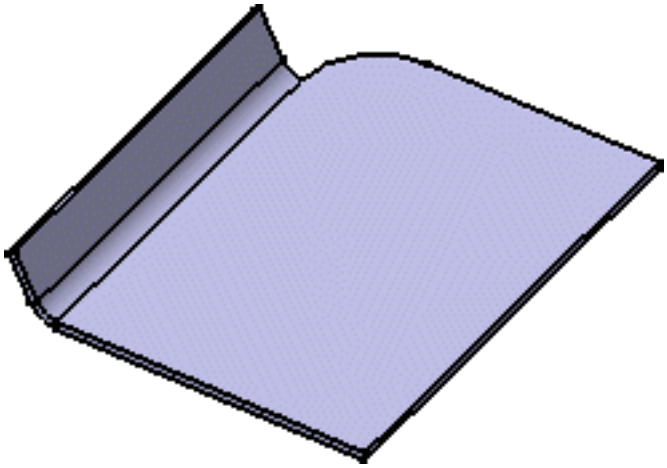
Selecting the Spine

Whatever the type of the swept wall you wish to create, you first need to select one or more contiguous edges to make up the spine along which the contour, either pre- or user-defined, is to be swept. You can:

- manually select one, or more, edge(s)

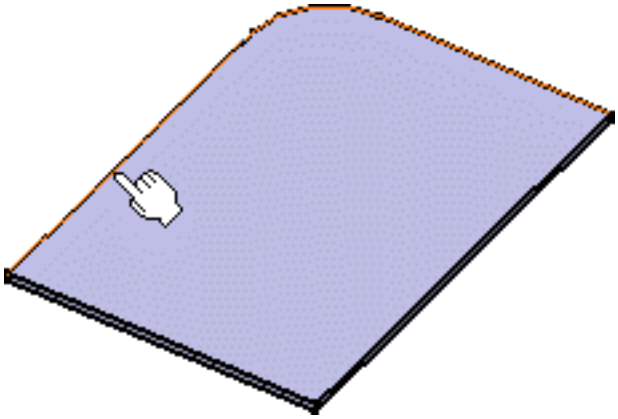


Selection without propagation

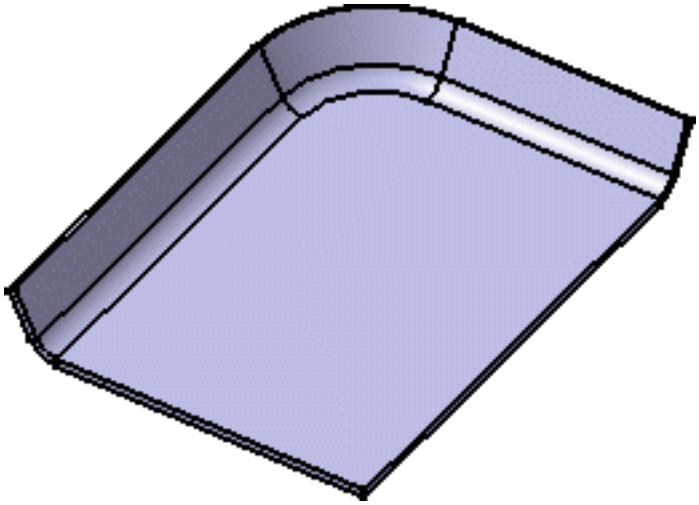


Resulting flange without propagation

- select one edge and click the **Tangency Propagation** button: all contiguous and tangent edges are selected. In this case, would you need to remove one edge, you need to manually select it. Remember that only extremity edges can be removed without breaking the continuity between edges.



Selection with propagation



Resulting flange with propagation

Back Up Next

Creating a Flange

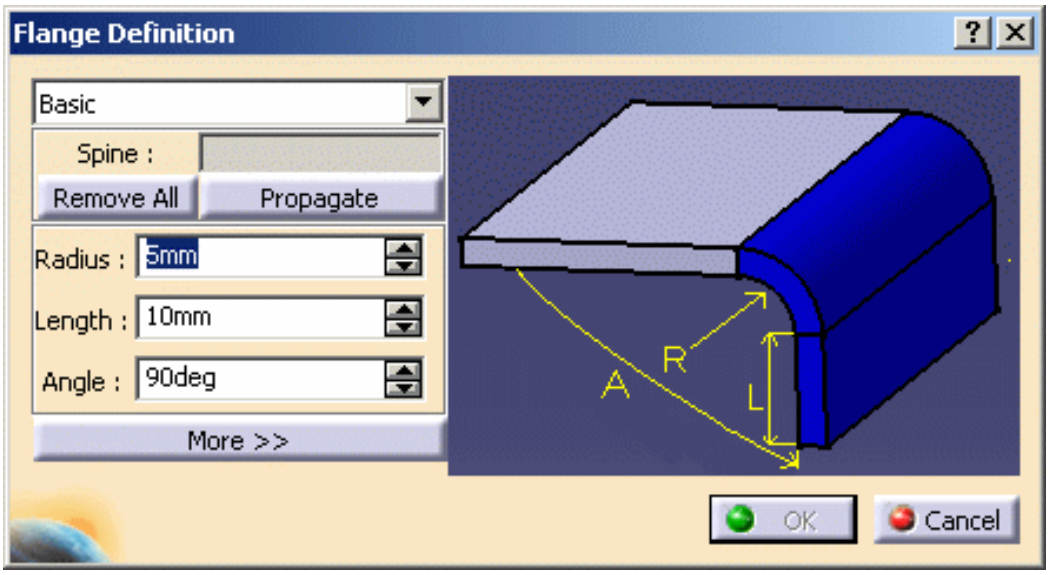
This task explains how to generate a flange from a spine and a profile.

Open the NEWSweptWall01.CATPart document from the samples directory.

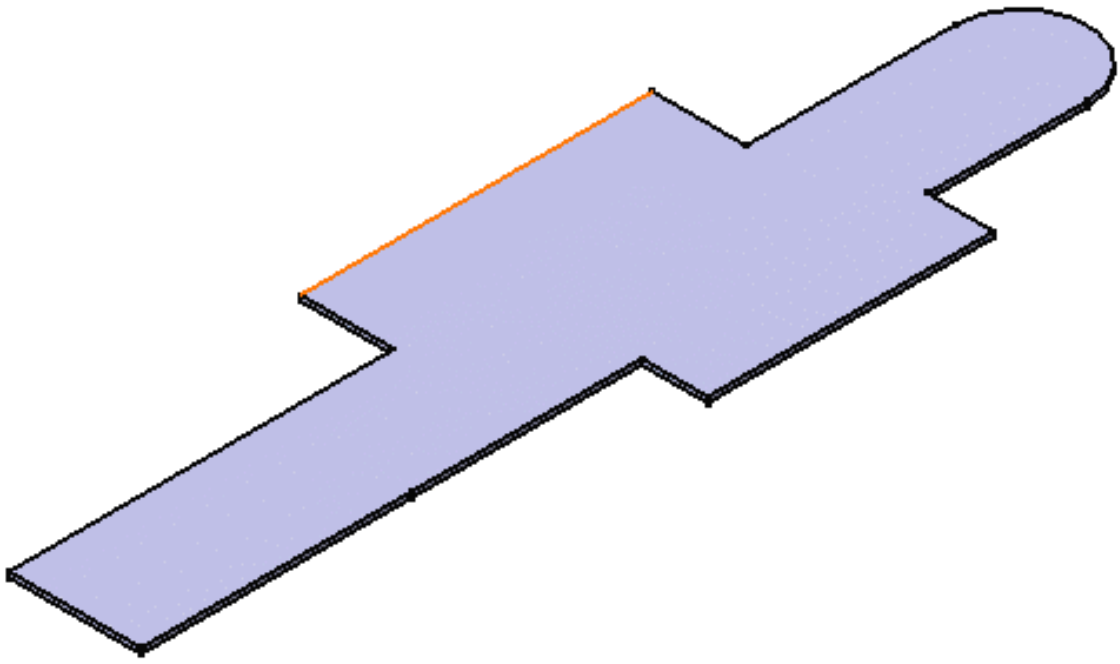
1. Select the **Flange** icon .

   The Flange Definition dialog box opens.

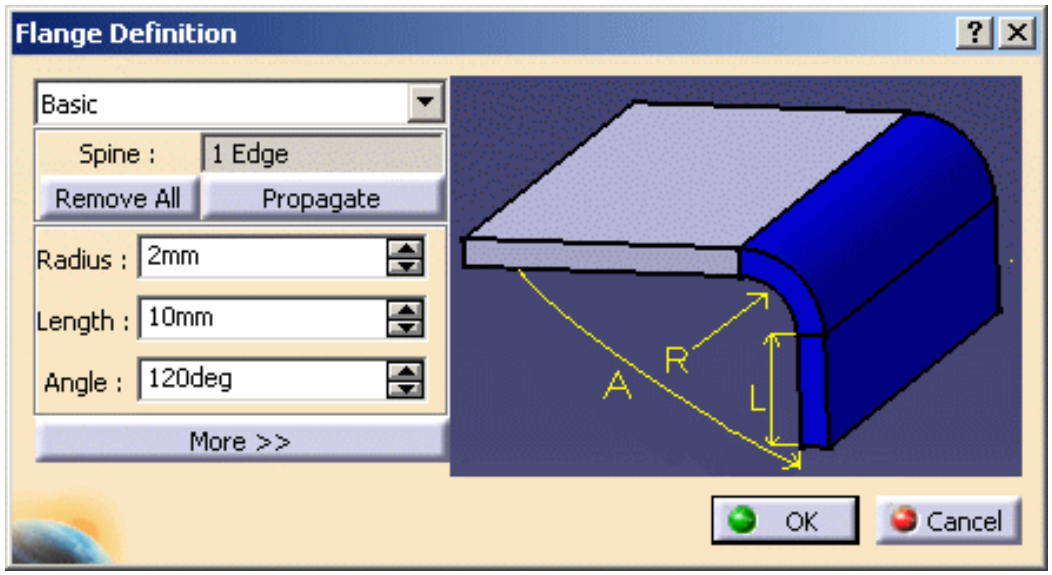




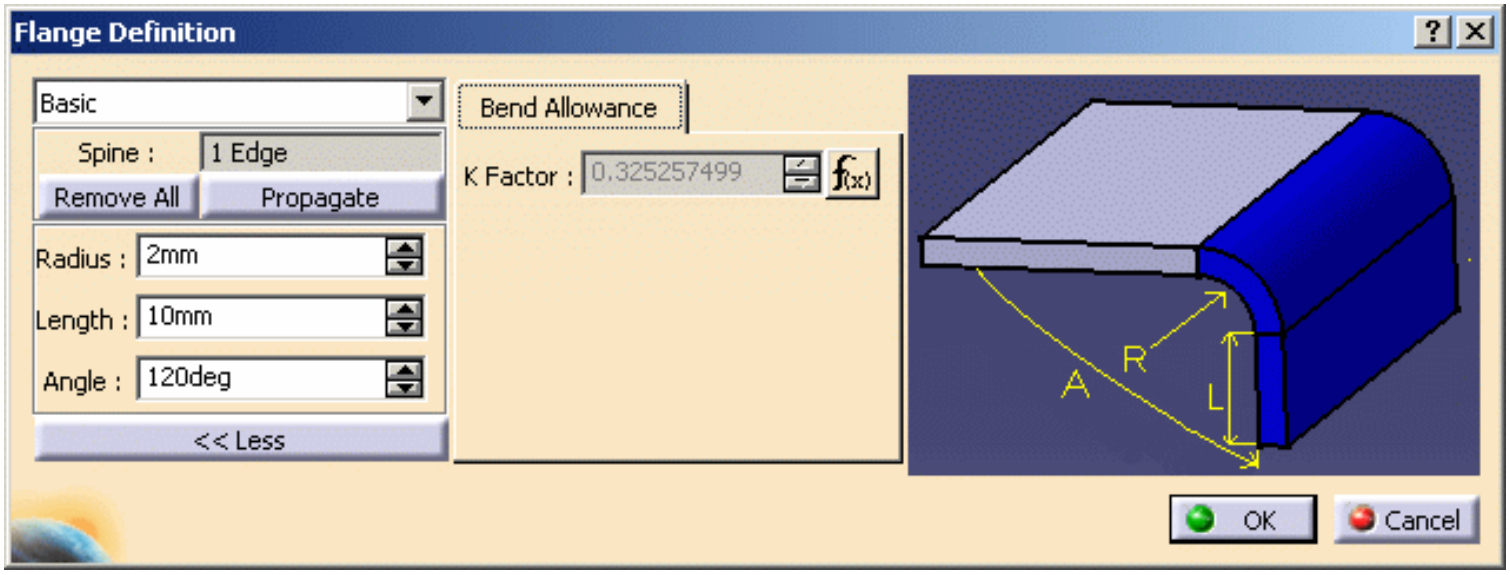
2. Select the edge as shown in red.



3. Enter 2mm in the Radius field, 10mm in the Length field and 120° for the Angle.

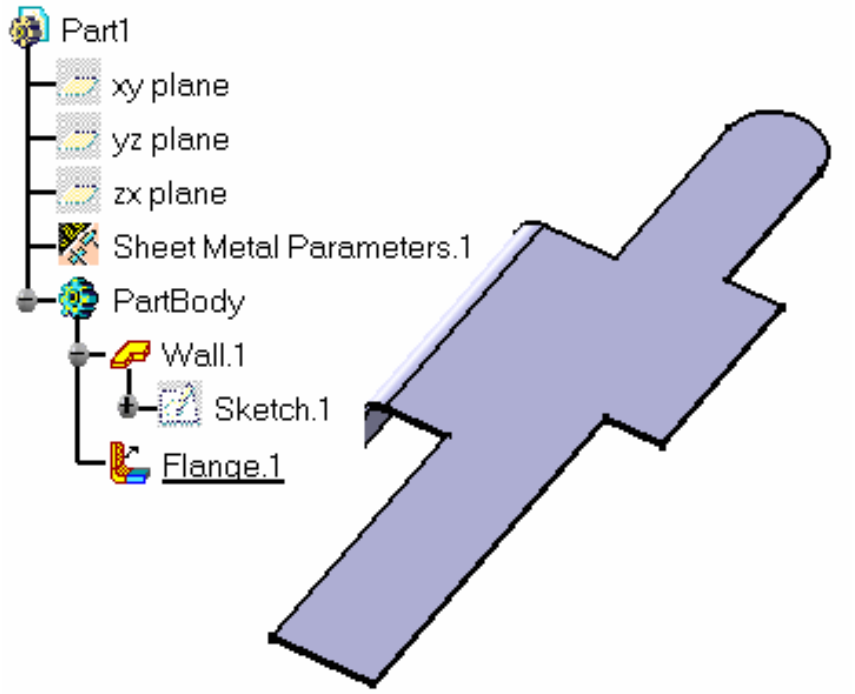


4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
   You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



   In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click OK to create the flange.



The feature is added to the specification tree.

- Use the **Remove All** button to remove the selected edge(s).
- Use the **Propagate** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the flange limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

Creating a Hem

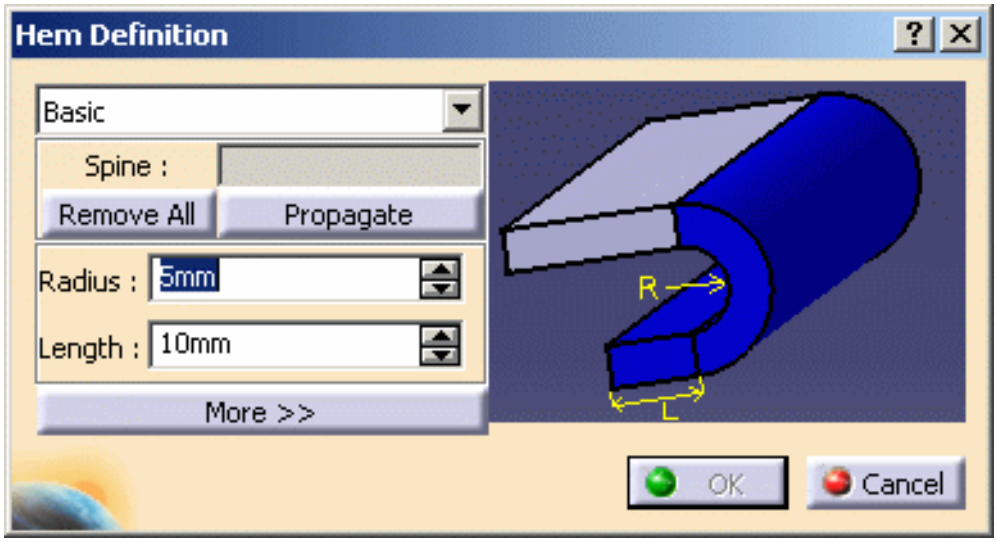
This task explains how to generate a hem from a spine and a profile.

The NEWSweptWall01.CATPart document is still open from the previous task.
If not, open the NEWSweptWall02.CATPart document from the samples directory.

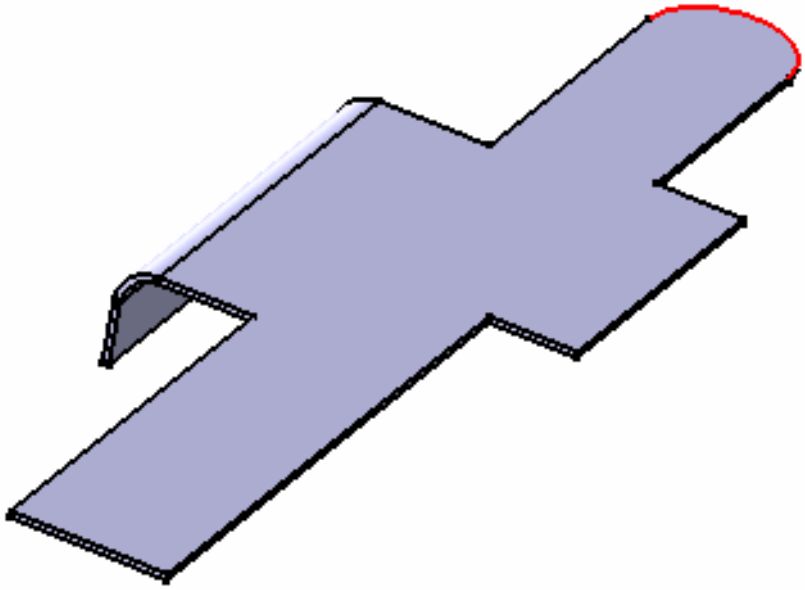
1. Select the **Hem** icon  in the **Swept Walls** sub-toolbar.



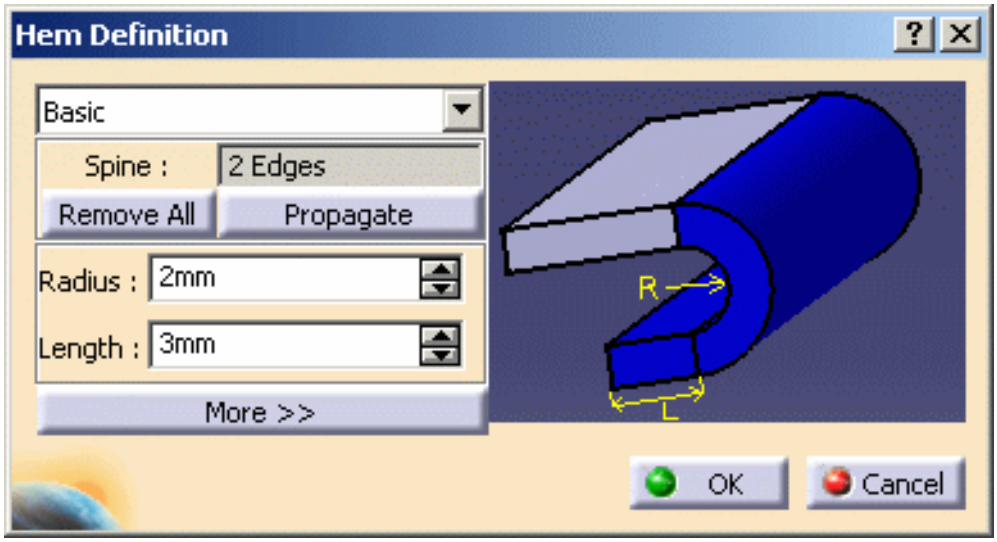
The Hem Definition dialog box opens.



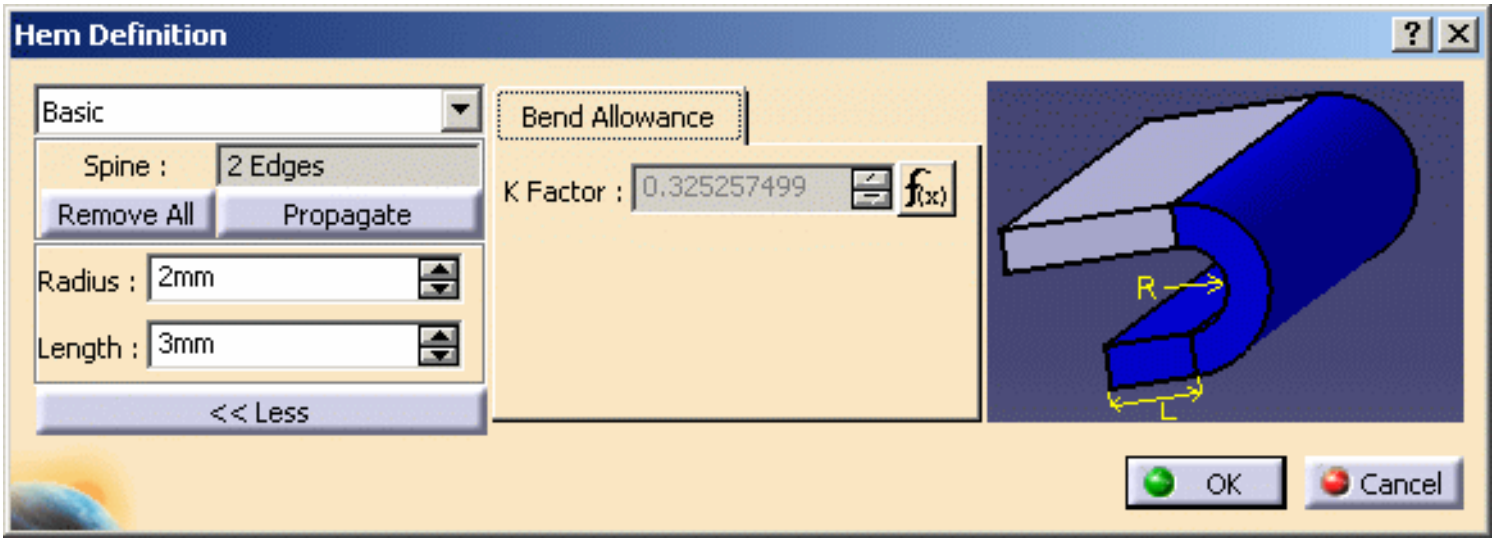
2. Select the edges as shown in red.



3. Enter 2mm in the **Radius** field, and 3mm in the **Length** field.

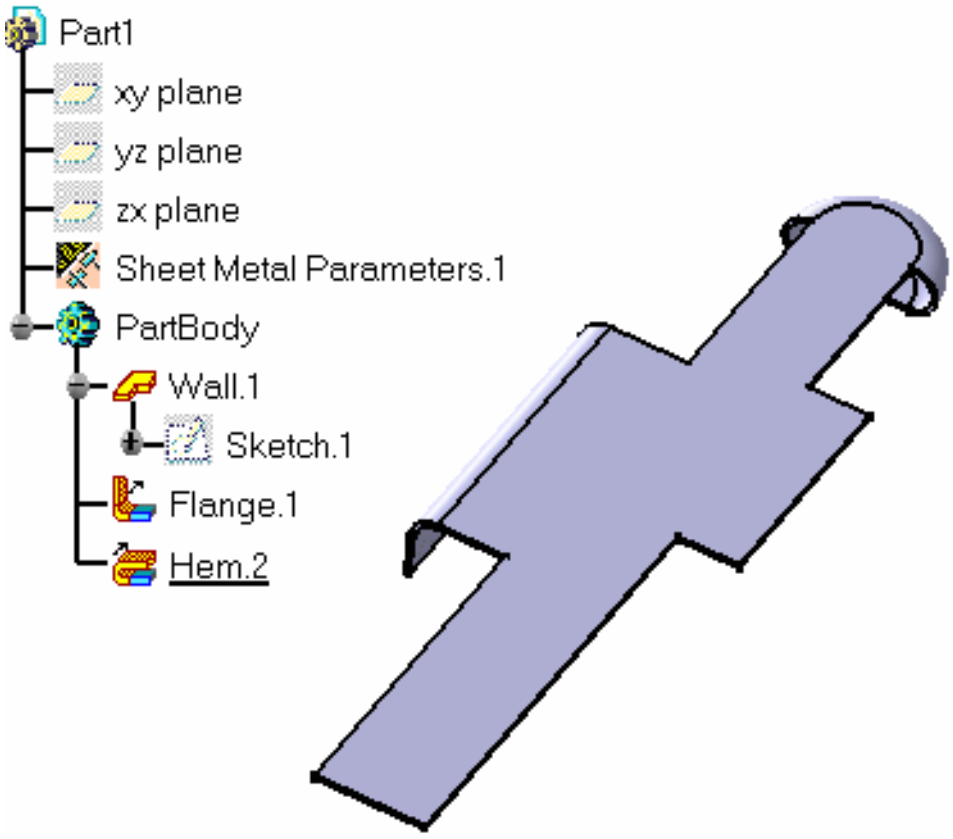


4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click **OK** to create the hem.



The feature is added to the specification tree.

- Use the **Remove All** button to remove the selected edge(s).
- Use the **Propagate** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the flange limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

Creating a Tear Drop

This task explains how to generate a tear drop from a spine and a profile.

The NEWSweptWall01.CATPart document is still open from the previous task.
If not, open the NEWSweptWall03.CATPart document from the samples directory.

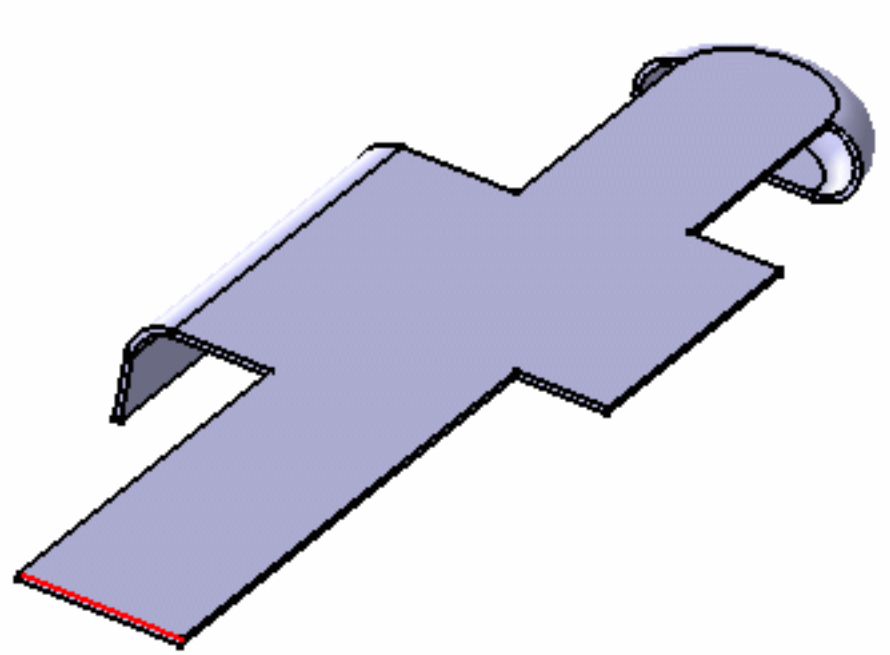
1. Select the **Tear Drop** icon in the **Swept Walls** sub-toolbar.





The Tear Drop Definition dialog box opens.

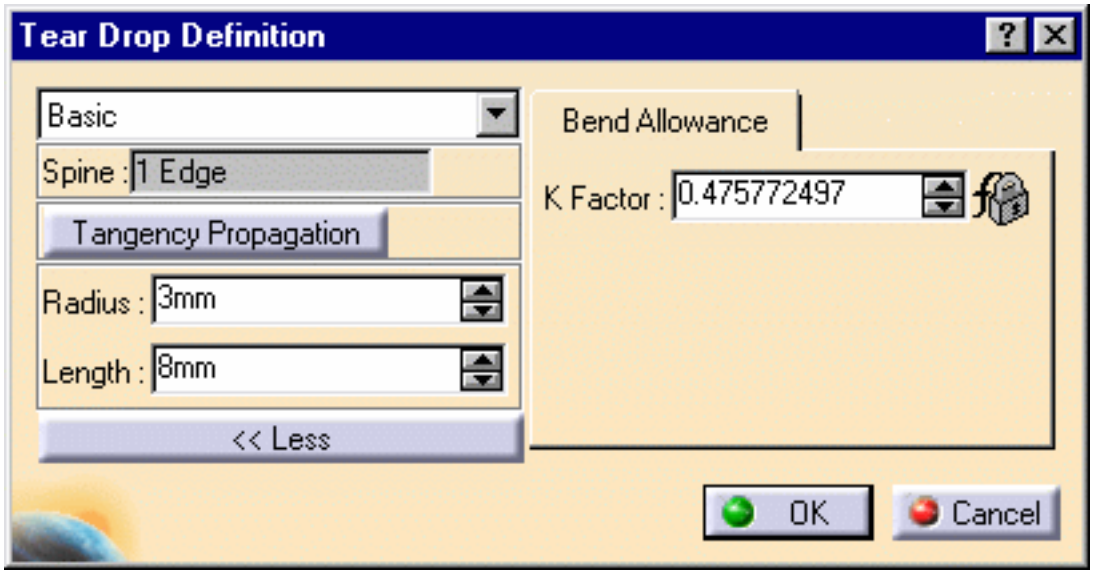
2. Select the edge as shown in red.



3. Enter 3mm in the **Radius** field, and 8mm in the **Length** field.

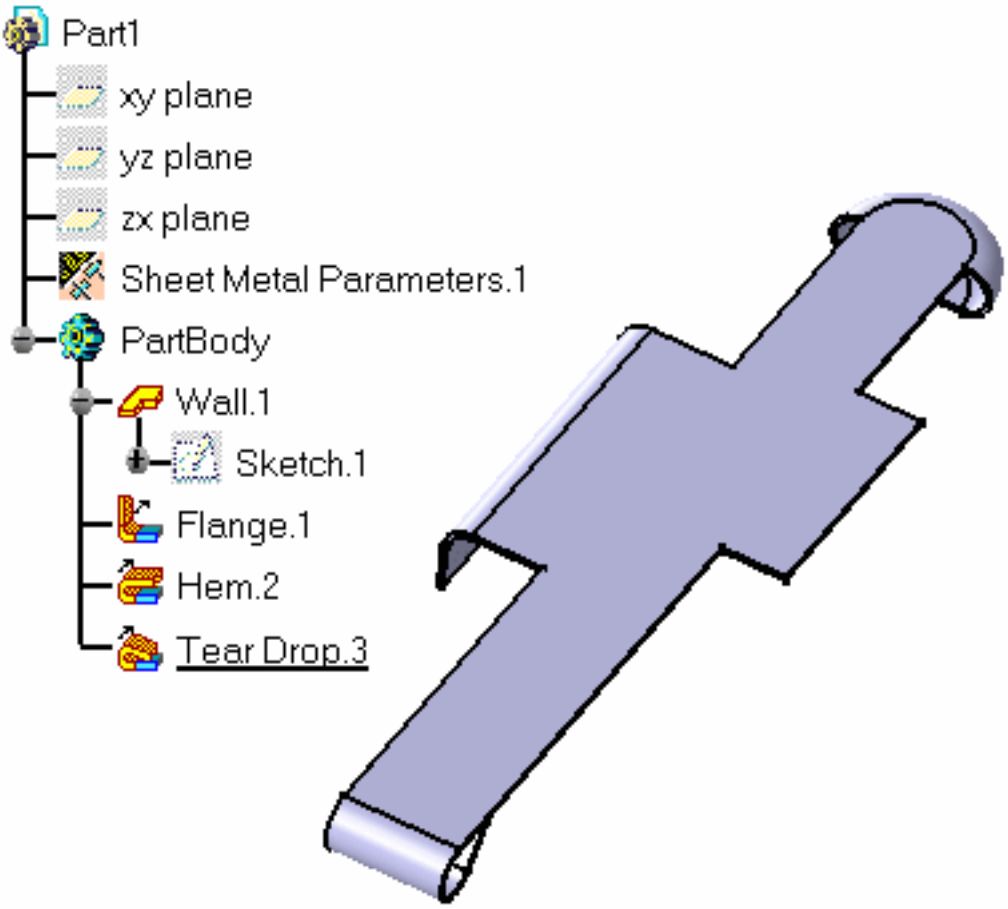


4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
   You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



   In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click **OK** to create the tear drop.



The feature is added to the specification tree.

- Use the **Tangency Propagation** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the tear drop limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).

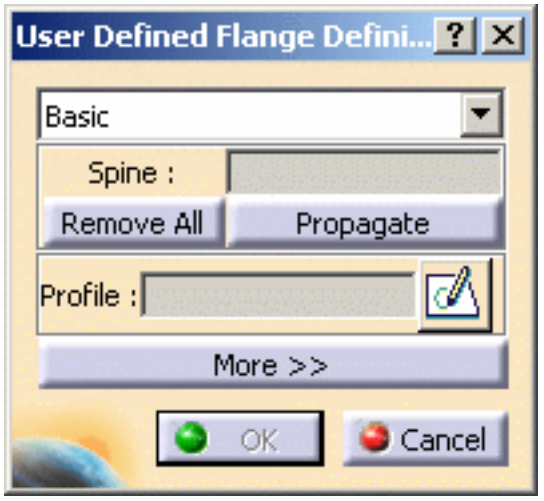
Creating a Swept Flange

This task explains how to generate a swept flange from a spine and a user-defined profile.

The NEWSweptWall01.CATPart document is still open from the previous task.
If not, open the NEWSweptWall04.CATPart document from the samples directory.

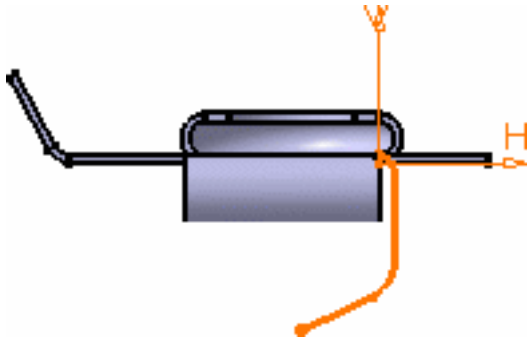
1. Select the **Swept Flange** icon in the **Swept Walls** sub-toolbar.





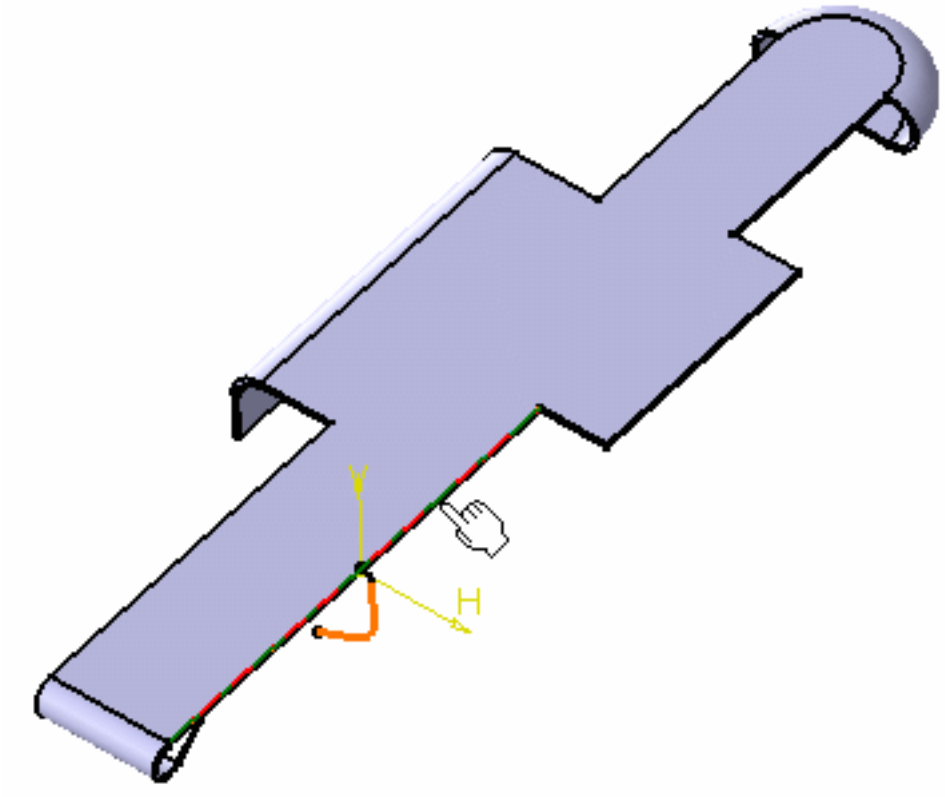
The **User Defined Flange** Definition dialog box opens.

2. Using the **Sketcher** icon , define a profile in the yz plane as shown below:

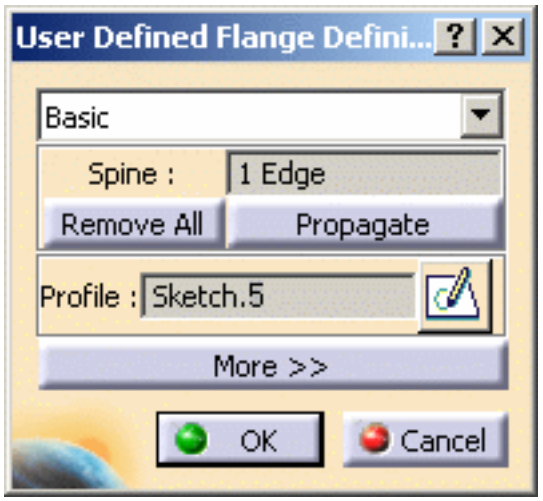


Then quit the Sketcher, using the **Exit** icon .

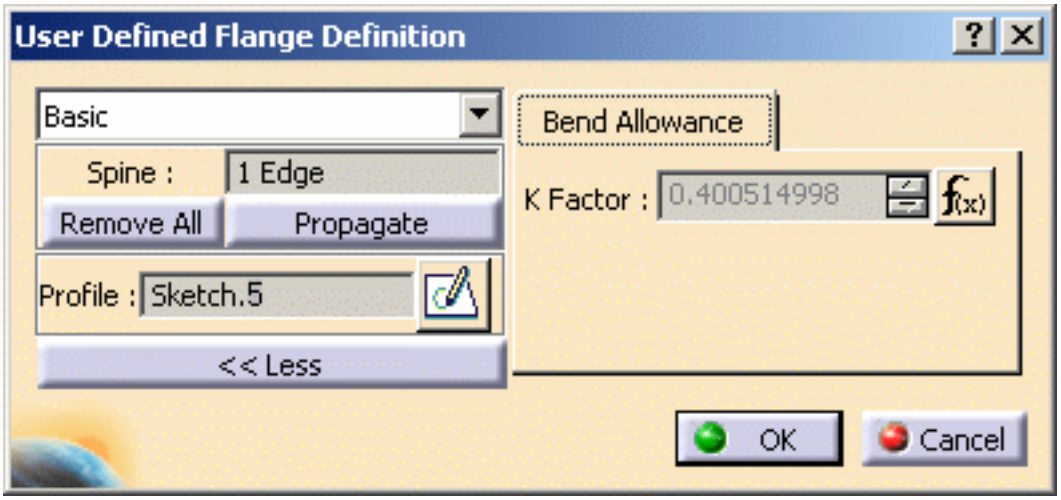
3. Select the edge and the profile, as shown in red.



The dialog box looks like this:

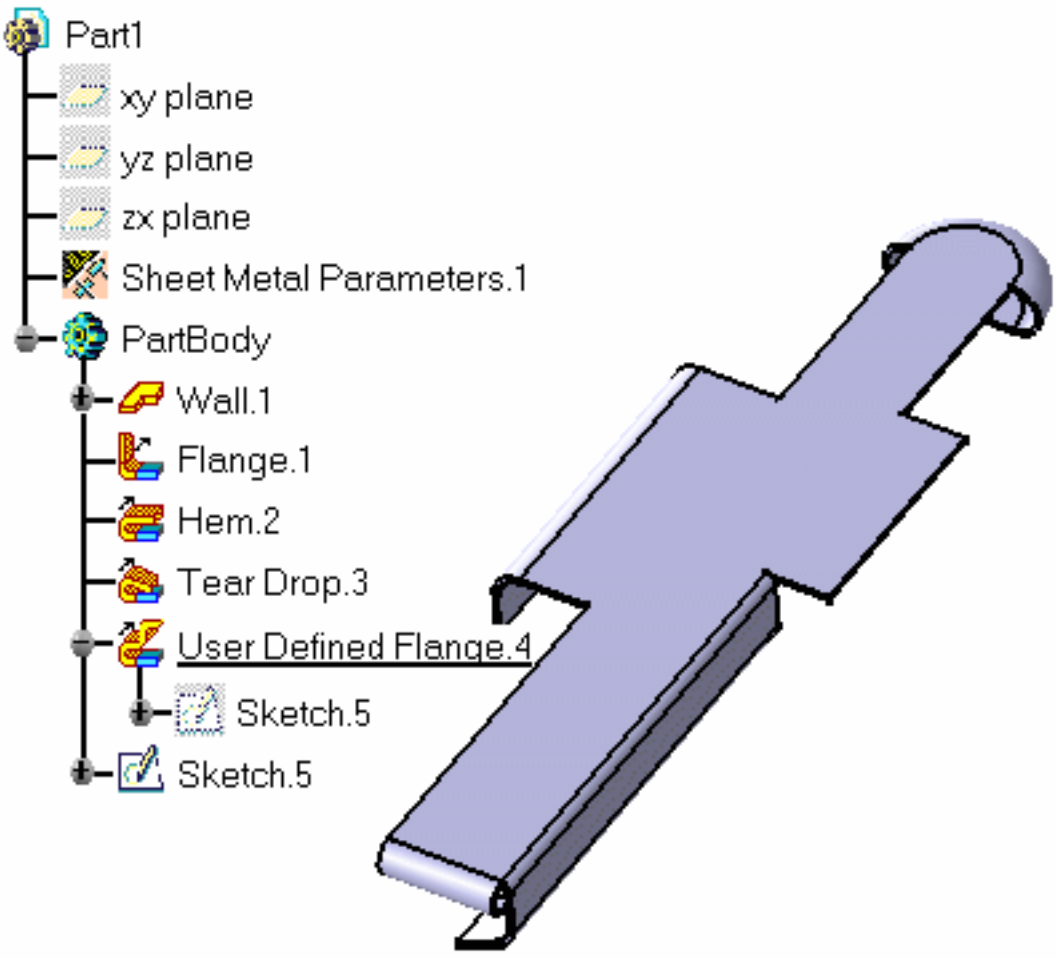


4. Click the **More** button to display the **Bend Allowance** tab allowing you to locally redefine the bend allowance settings.
   You may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.



In this case, the new K Factor value overrides the value set in the Sheet Metal Parameters.

5. Click **OK** to create the swept flange.



The feature is added in the specification tree.

- Use the **Remove All** button to remove the selected edge(s).
- Use the **Propagate** button to select all tangentially contiguous edges forming the spine (see Selecting the Spine).
- You can redefine the flange limits by choosing the **Relimited** option (see Redefining Swept Walls Limits).



Redefining Swept Wall Limits

This task explains how to redefine the spine's limits when creating any type of swept walls, using existing geometric elements: points lying on the spine or intersecting planes..

Open the SweptWall01.CATPart document.

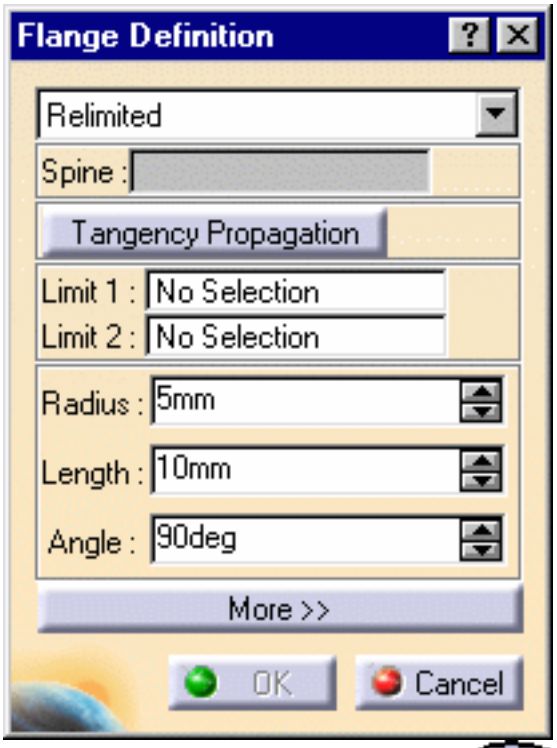
Let's take en example with the Flange (SheetMetal Design). This task can also be performed in the New SheetMetal Design as well as

1. Select the **Flange** icon .

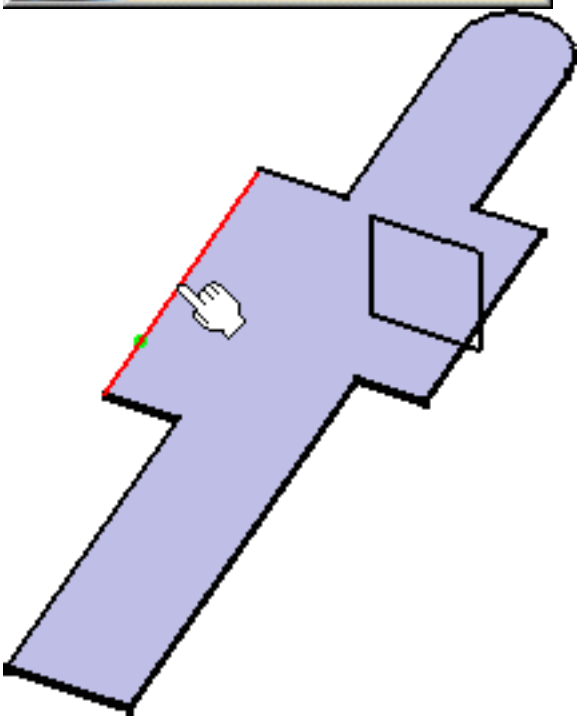
   The Flange Definition dialog box opens.

2. Using the combo list, choose the Relimited type.

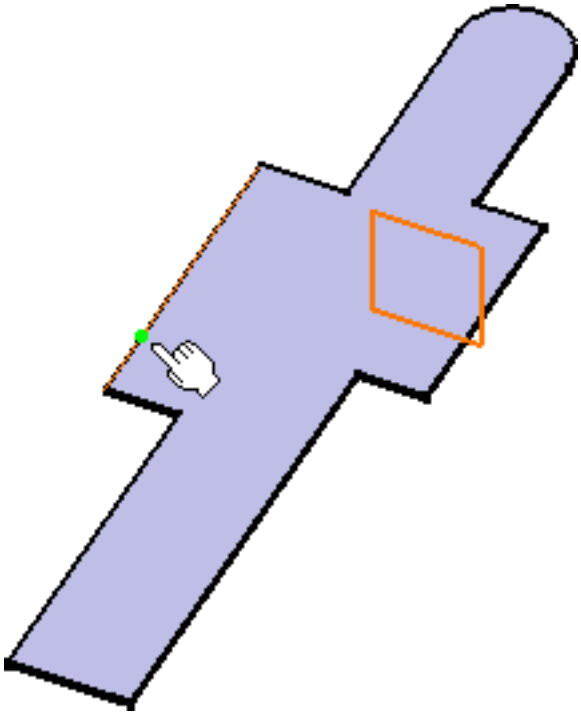
   The Flange Definition dialog box is updated and now displays two **Limit** fields.



3. Select the spine.
   Here we select a single edge. See also Selecting the Spine.



4. Successively select the two limiting elements.
   Here we select a point lying on the spine as the first selecting element, and a plane intersecting the spine as the second limiting element.

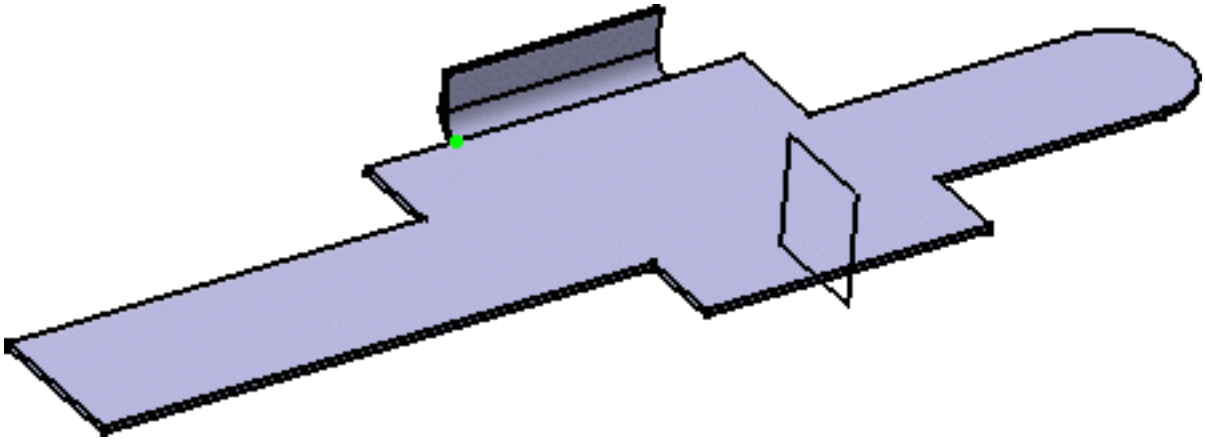


Make sure intersecting elements are normal to the spine, and they intersect it only once.

5. Specify the swept wall values.
   In the example of the flange you set the Radius, Length and Angle values. You may also click **More >>** to display further options. See Creating a Flange.

6. Click OK.

The swept wall is created within the limits on the spine.



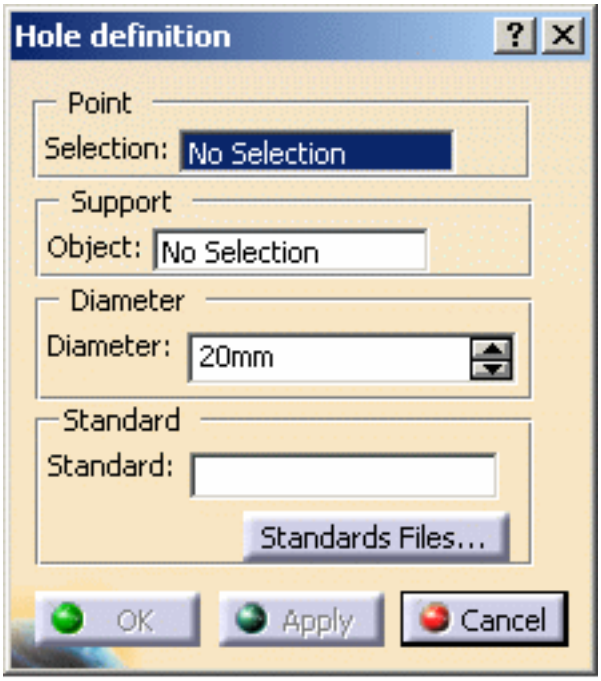
Creating a Hole

This task shows you how to create a hole, that consists in removing material from a body.

Open the Hole1.CATPart document.

1. Click the **Hole** icon .

   The Hole definition dialog box opens.



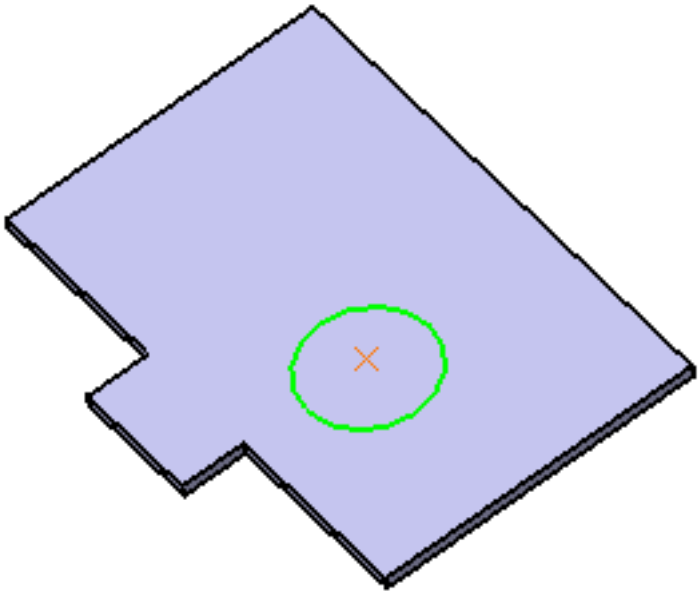
2. Select the **Point** that will be the center of the hole.
   It can be either a sketch containing one or more points, or a point, or several points. The points must be on the same support.

- The point can be selected anywhere in the geometry, not necessarily on a surface. In that case, an orthogonal projection will be performed.
- You can also directly click the surface: a point will be created under the pointer.
- To deselect a point, click it in the specification tree.

3. Select the **Support** object where the hole will be positioned.

- The support can be different from the support where the point lies. In that case, an orthogonal projection will be performed.

The hole is previewed with default parameters.

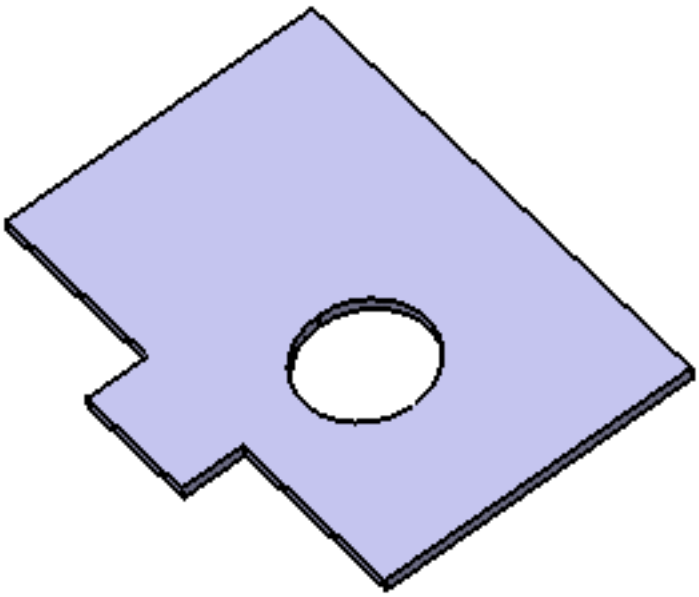


4. Define the value for the diameter of the hole in the **Diameter** field.

If you change the **Diameter** value using the spinners, the preview of the hole automatically updates. However, if you enter a value directly in the field, you need to click the **Apply** button to update the preview.

5. Click OK to validate.

The hole (identified as Hole.xxx) is created and the specification tree is updated accordingly.



Holes can be created on the flattened part and on bends.

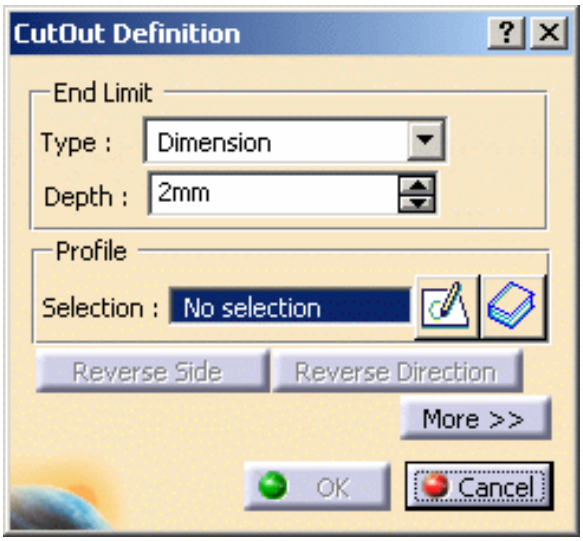
To have further information on **Standard Files...**, please refer to the Customizing section.

Creating a Cutout

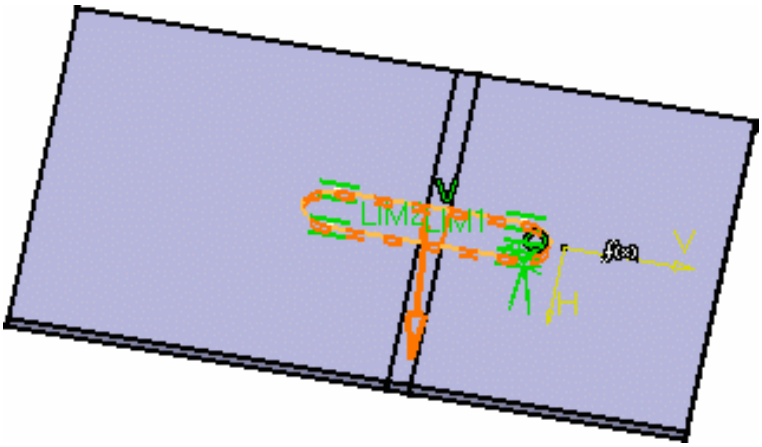
This task explains how to create a cutout in a wall.
Creating a cutout consists in extruding a profile and removing the material resulting from the extrusion.

Open the NEWCutout1.CATPart document.

1. Click the **Cutout** icon .
2. Select a profile.



The Pocket Definition dialog box is displayed and CATIA previews a cutout with default parameters.



Once the sketch is selected, you can modify it by clicking the Sketcher icon .

3. Select the type.

Several limit types are available:

- **Dimension**: the cutout depth is defined by the specified value
- **Up to next**: the limit is the first face the application detects while extruding the profile. This face must stops the whole extrusion, not only a portion of it, and the hole goes through material.
- **Up to last**: the last face encountered by the extrusion is going to limit the cutout
- **Up to plane**: the cutout is limited by the selected plane
- **Up to surface**: the cutout is limited by the selected surface

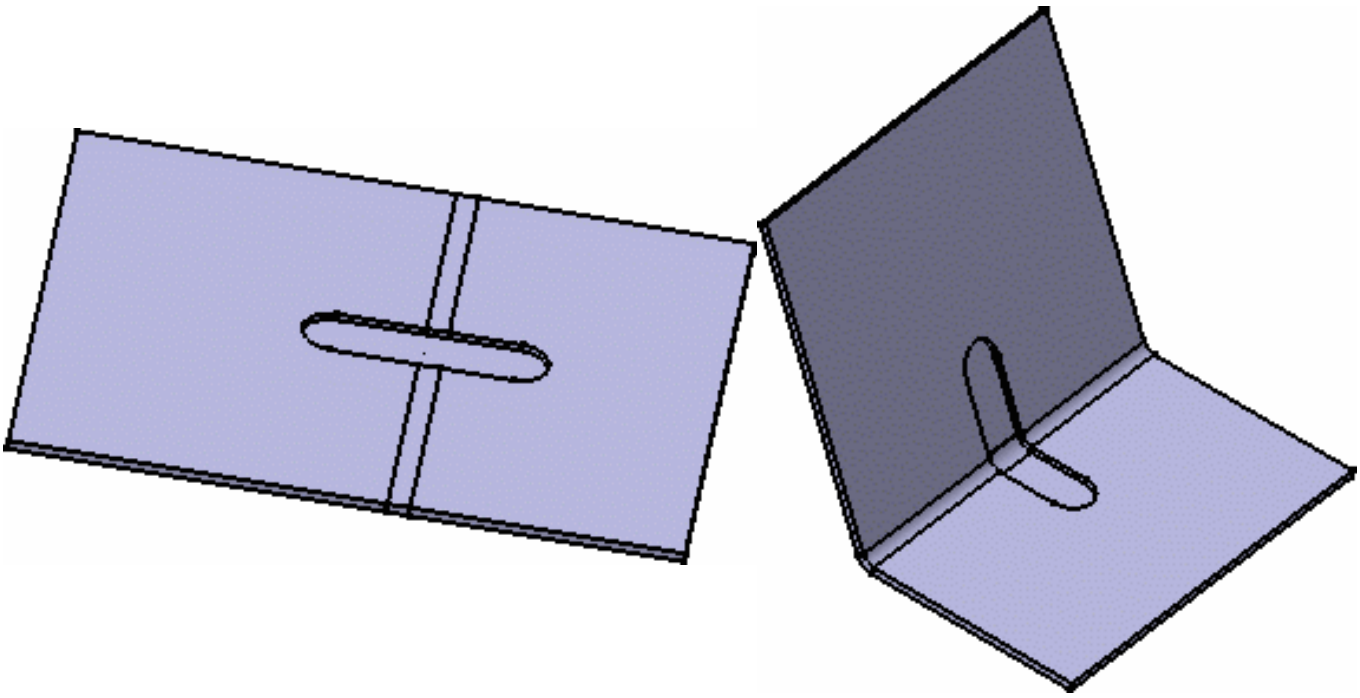
When an **Offset** value is required, it is the distance between the limiting element and the top face of the cutout, if the latter does not result in a complete hole through the material.

The LIM1 and LIM2 texts in the geometry area indicate the top and bottom limits of the cutout.

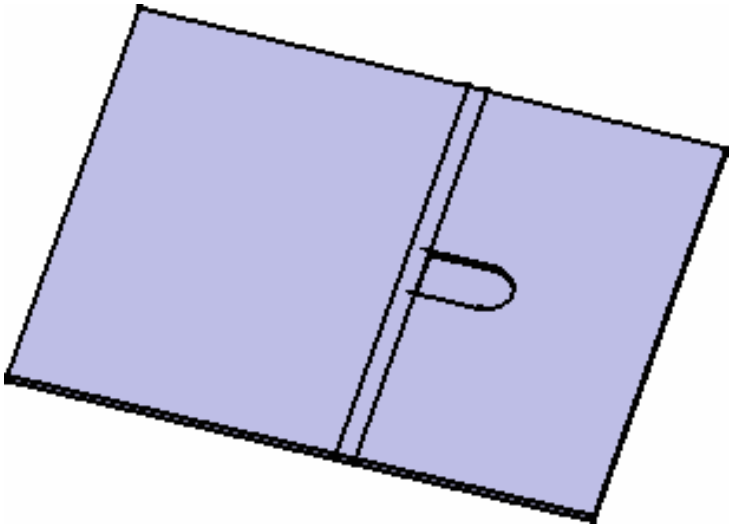
- The **Reverse Side** option lets you choose between removing the material defined within the profile, which is the application's default behavior, or the material surrounding the profile.
- The **Reverse Direction** option allows you to invert the direction of the extrusion pointed by the arrow.

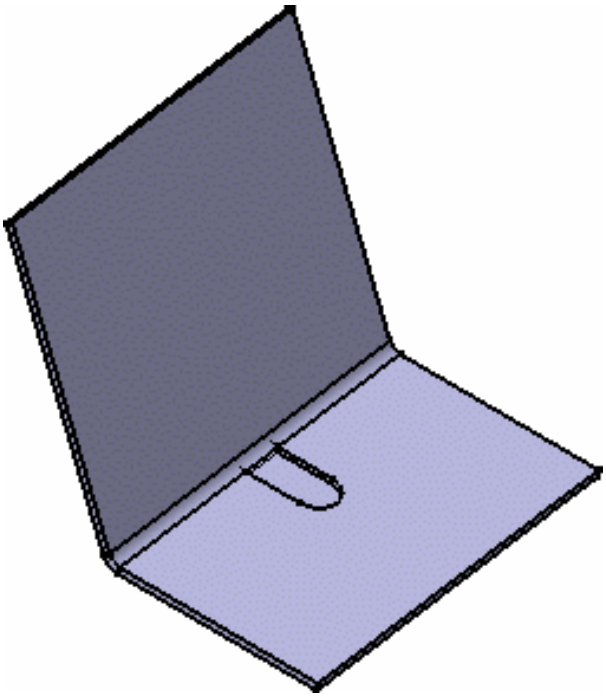
4. Click OK in the CutOut Definition dialog box.

   The cutout is created.

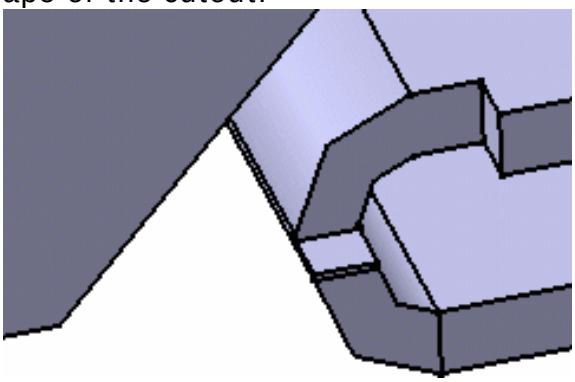


You may want to fold or unfold the part prior to creating the cutout, depending on the selected profile and the expected results. Selecting the same profile, the resulting cutout when created in the unfolded view is seen above, while below, the cutout was created in folded view.

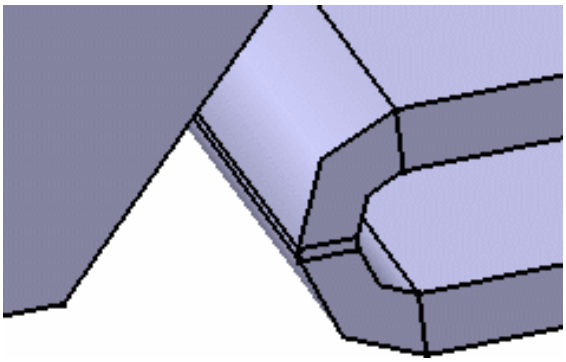




However, you also have to be careful when designing the Part, not to create the cutout in the unfolded view as this may lead, when folding the part, to completely or partially removing another section of the Part. It is best, whenever possible, to create the wall based on a sketch integrating the shape of the cutout.



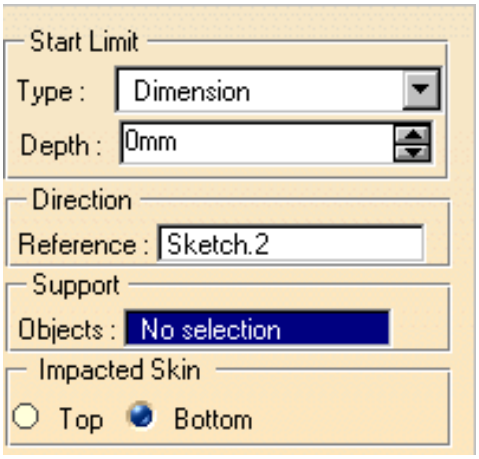
Cutout removing material



Cutout not removing material based on wall's sketch modification

4. Click **More>>** to display the maximum information.

The Direction is already selected (Sketch.3). If not, it must be set as normal to the profile.



5. Select the **Support** (here we chose the wall)

May you want to create a cutout on an overlapping element or a bend with radius=0, either choose the top skin of the element (as shown in the picture above), or unfold the part to create the cutout.

You can use the Catalog icon  to open the Catalog Browser.

- Refer to the *Component Catalog Editor* documentation to have further information on how to use catalogs.
- Refer to the Create a Pocket task in the *Part Design User's Guide* for further details on how to create cutouts.

Stamping

This section explains and illustrates how to create and use various kinds of stamps.

- Stamps must be created on walls, or walls on edge, except for the stiffness rib that is to be created on a bend.
- If a stamp is created over the limit between several supports, such as walls, bends, and so forth, this stamp is not visible on the unfolded view.
- When unfolding a part, only the largest imprint of the stamp is retained on the stamped wall.
- Stamps cannot be created on an unfolded part.

Creating Standard Stamping Features
Creating User-Defined Stamping Features

Back Up Next

Creating Standard Stamping Features

This section explains and illustrates how to create and use various kinds of stamps.
The table below lists the information you will find.

 Create a flanged hole: select a point on a face, and set the stamping parameters.

 Create a bead: select a profile, and set the stamping parameters.

 Create a circular stamp: select a point on a face, and set the stamping parameters.

 Create a surface stamp: select a sketch, and set the stamping parameters.

 Create a bridge: select a point on a face, set the stamping parameters, and select an edge to give the bridge orientation.

 Create a flanged cutout: select a profile, and set the stamping parameters.

 Create a stiffening rib: select the external surface of a bend, and set the stamping parameters.

 Create a curve stamp: select a sketch, and set the stamping parameters.

 Create a louver: select a sketch, an opening line and set the stamping parameters.

Up

Creating a Flanged Hole

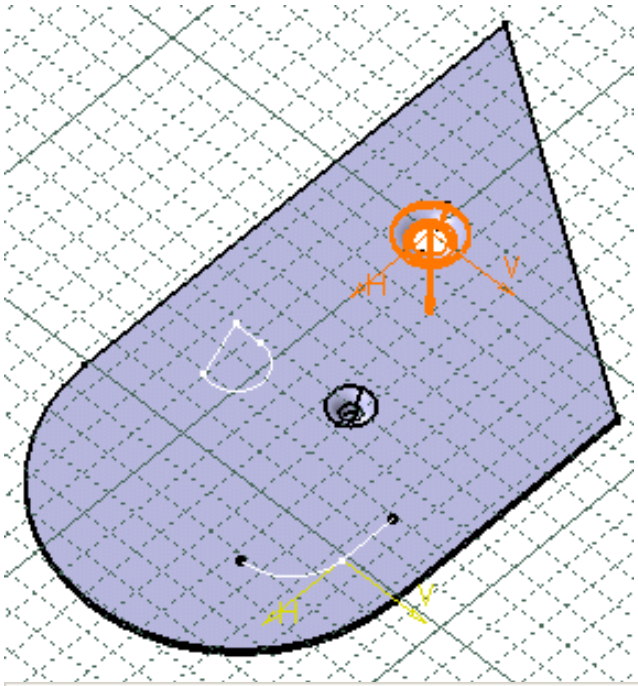
This task shows you how to create a flanged hole by specifying the punch geometrical parameters.

Open the NEWStamping.CATPart document.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping.CATPart document.

1. Click the **Flanged Hole** icon .

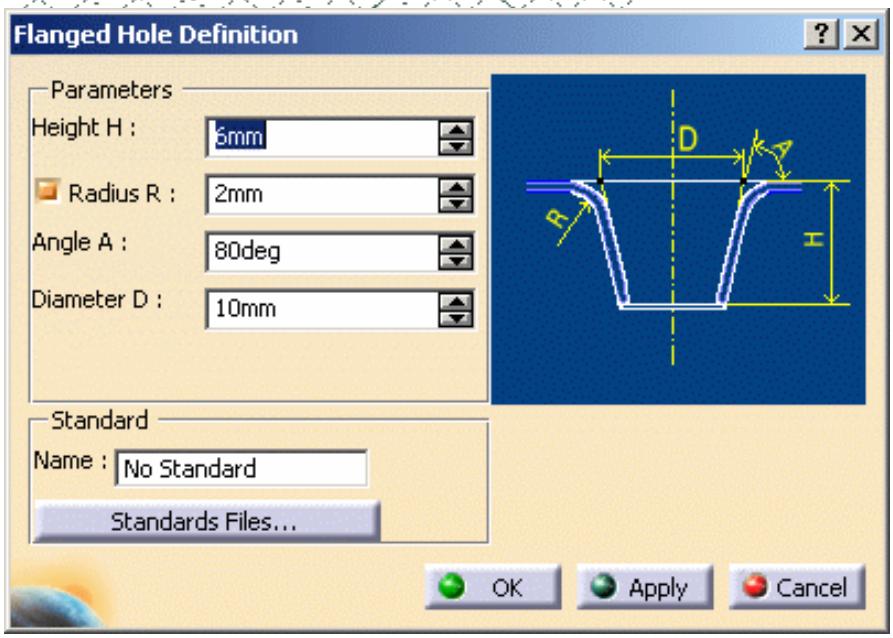
2. Select the surface where you want to place the hole.

   A grid is displayed to help you position the flanged hole and the Flanged Hole Definition dialog box opens, providing default values.



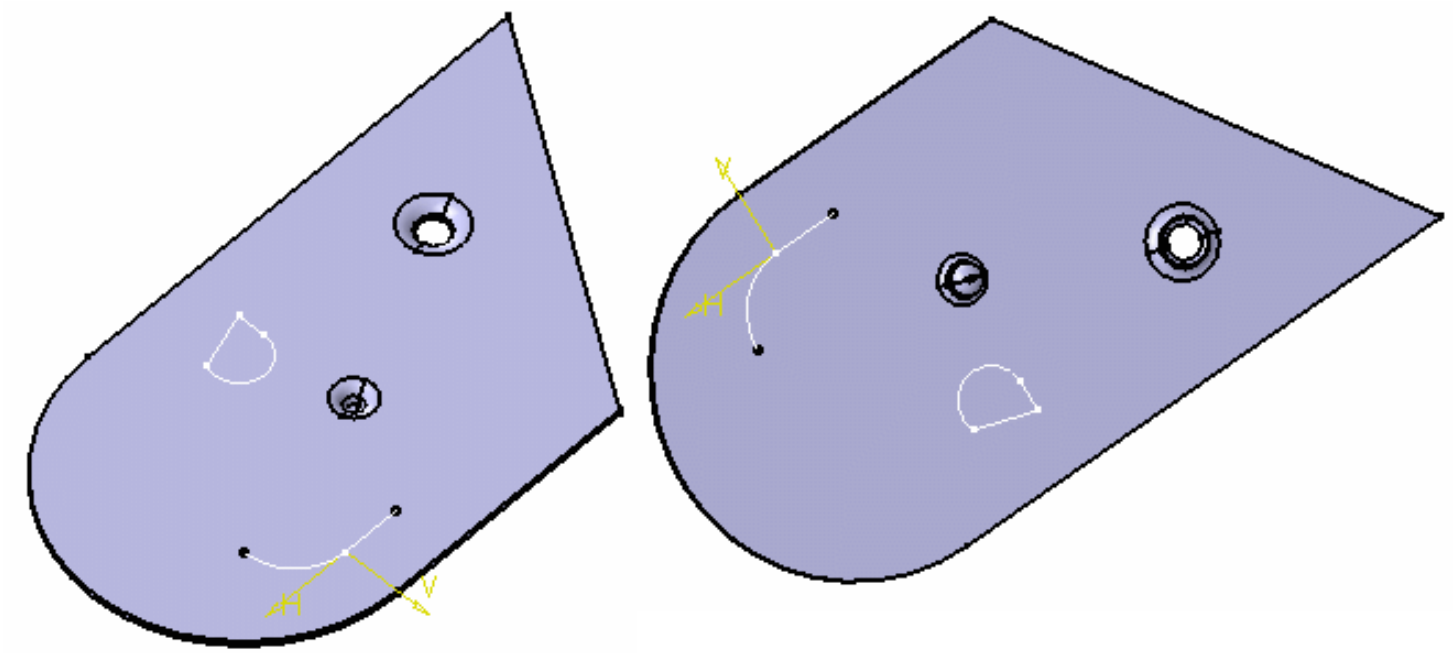
3. Change the value in the different fields, if needed:

   - Height H
   - Radius R
   - Angle A
   - Diameter D



4. Click **Apply** to preview the flanged hole.
5. Click **OK** to validate.

The flanged hole (identified as Flanged Hole.xxx) is created and the specification tree is updated accordingly.



You can use 0 as the Radius value to deactivate the Radius R value, and to create the flanged hole without a fillet.



Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Bead

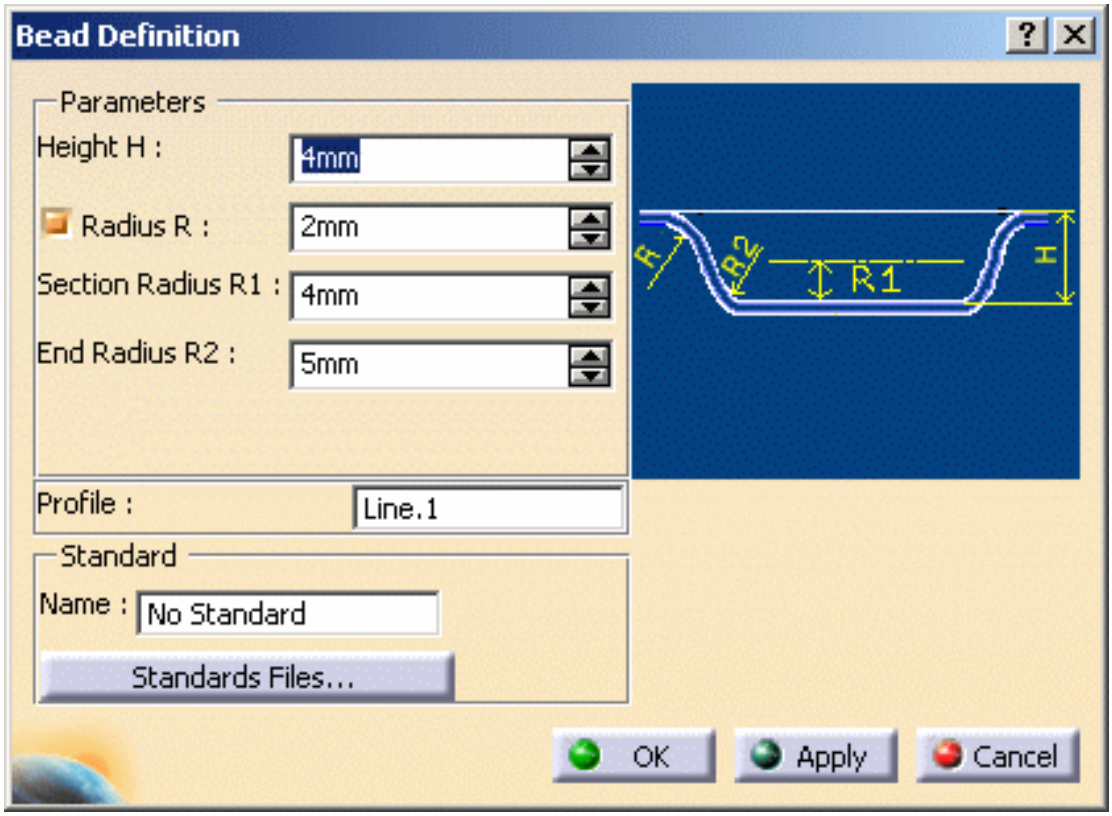
This task shows you how to create a bead, that is a local deformation in the web.

Open the NEWStamping6.CATPart document.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping6.CATPart document.

1. Click the **Bead** icon .

2. Select the spine profile where you want to place the bead.

   The Bead definition dialog box is displayed, providing default values.

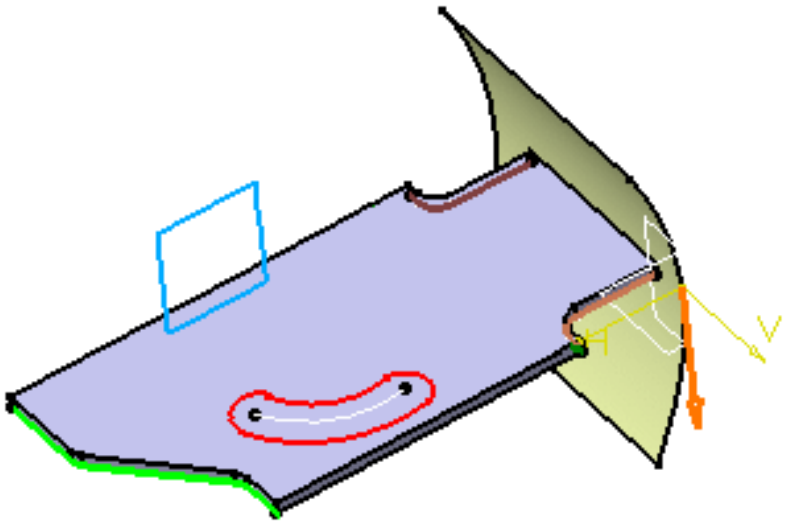


3. Change the value in the different fields, if needed:

   - Height H
   - Radius R
   - Section Radius R1 (corresponding to the cross section value)
   - End Radius R2

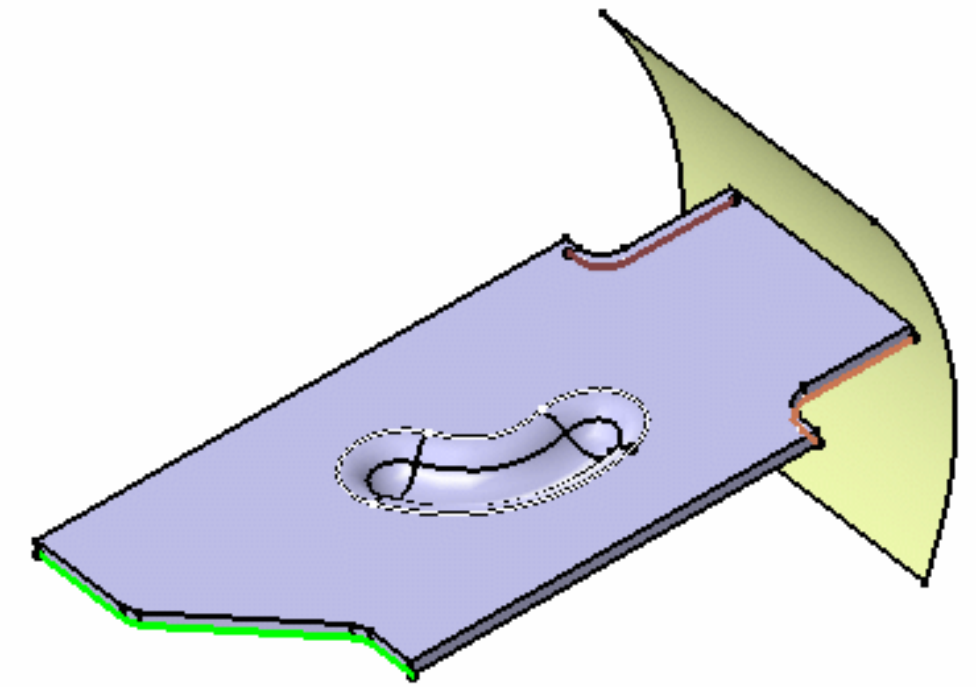
The **Sketch** is automatically set to the sketch you chose.

The vector for the direction of the bead is shown in the model and a preview of the bead appears and a vector shows its direction.



4. Click **Apply** to preview the bead.
5. Click **OK** to validate.

The bead (identified as Bead.xxx) is created and the specification tree is updated accordingly.



The vector cannot be reverted until the bead spine is defined.

You can use 0 as the Radius value to deactivate the Radius R value, and to create the bead without a fillet.

Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Circular Stamp

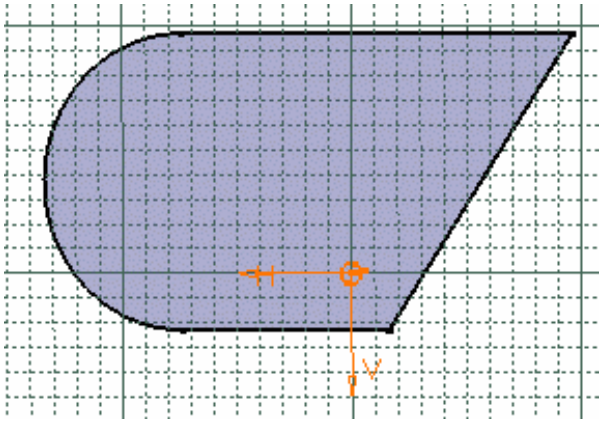
This task shows you how to create a point stamp by specifying the punch geometrical parameters.

Open the NEWStamping.CATPart document from the samples directory.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping.CATPart document.

1. Click the **Circular Stamp** icon .

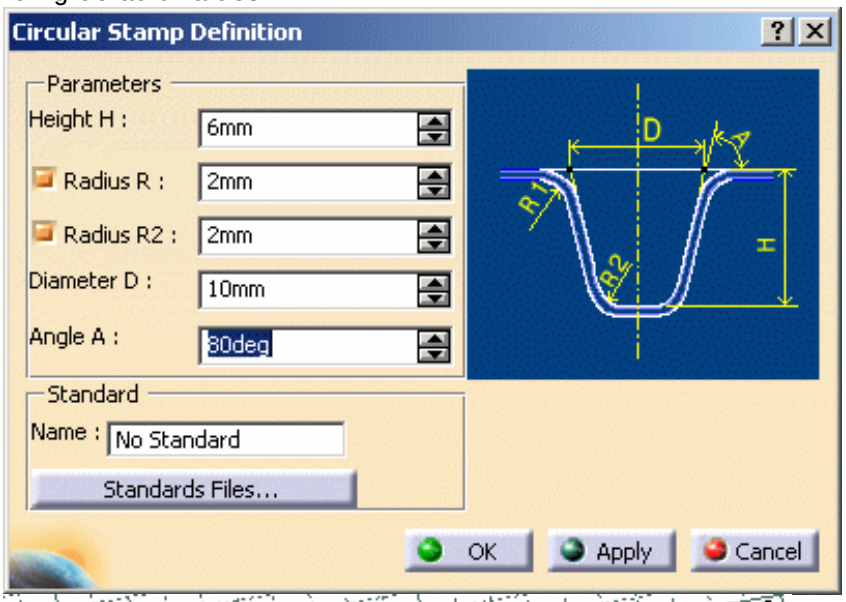
2. Select a point on the top face.

   A grid is displayed to help you position the circular stamp.

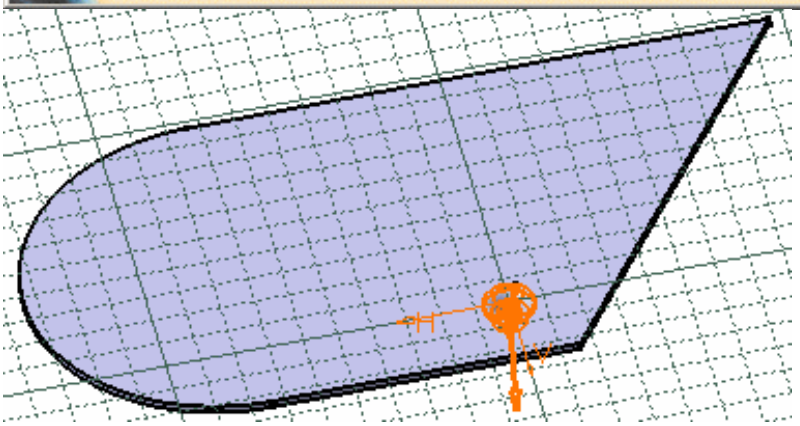


The Circular Stamp Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:
   - Height H
   - Radius R1
   - Radius R2
   - Angle A
   - Diameter D

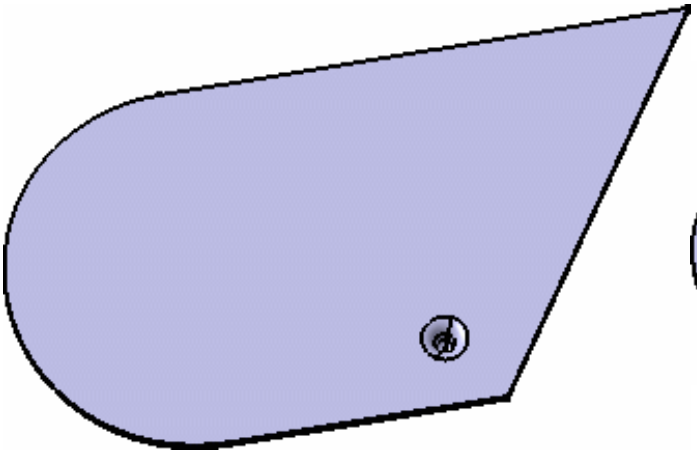


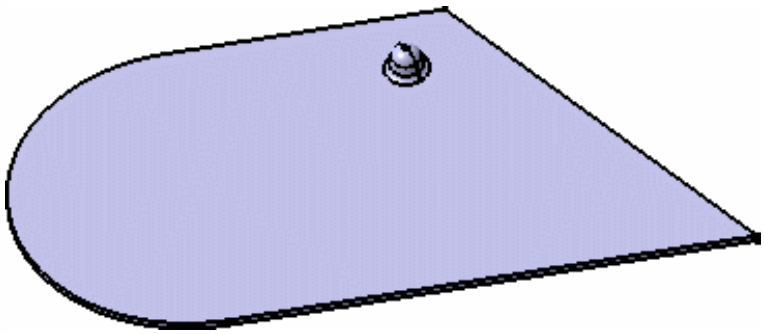
4. Click **Apply** to preview the circular stamp.



5. Click **OK** to validate.

   The circular stamp (identified as Circular Stamp.xxx) is created and the specification tree is updated accordingly.





You can use 0 as the Radius value to deactivate the Radius R and Radius R2 values, and to create the point stamp without a fillet.



Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Surface Stamp

This task shows you how to create a surface stamp by specifying the punch geometrical parameters.

Open the NEWStamping4.CATPart document from the samples directory.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping4.CATPart document.

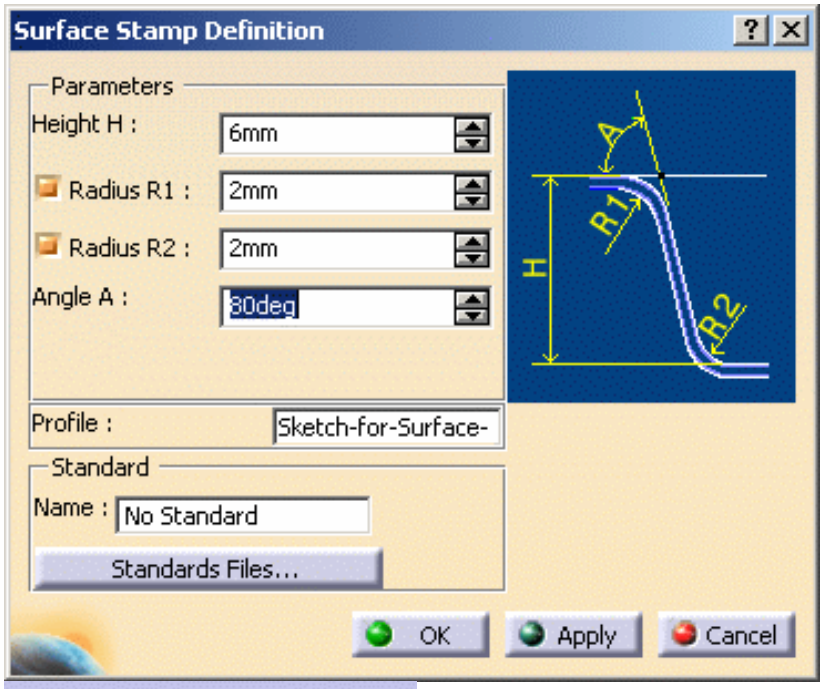
1. Click the **Surface Stamp** icon .

2. Select **Sketch-for-Surface-Stamp**, the profile previously defined.

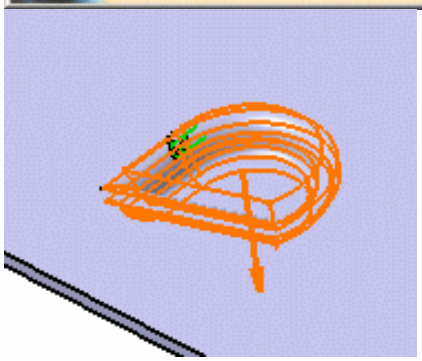
   The Surface Stamp Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

   - Height H
   - Radius R1
   - Radius R2
   - Angle A

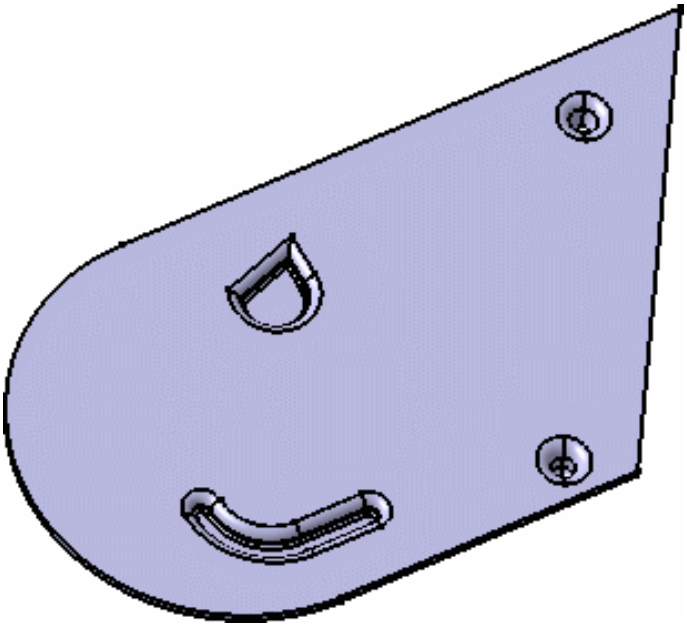


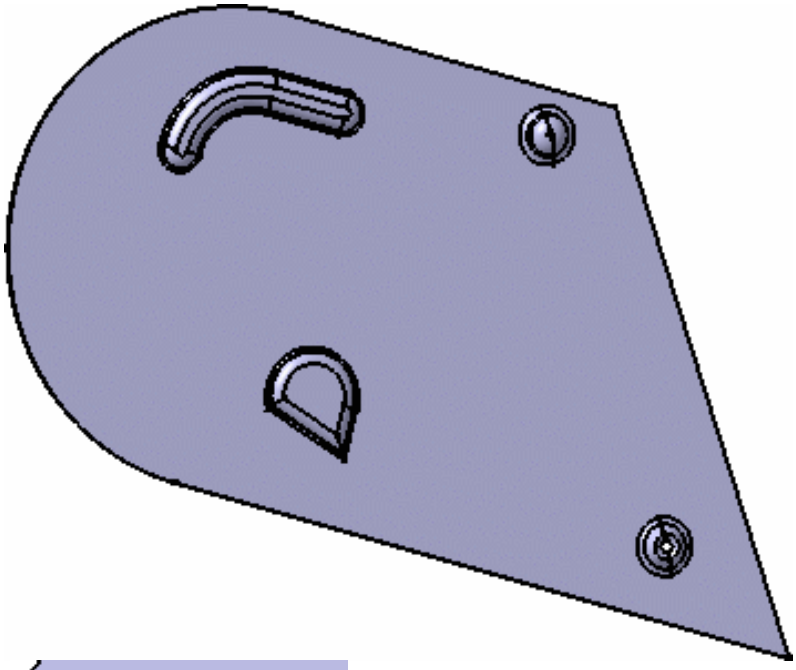
4. Click **Apply** to preview the surface stamp.



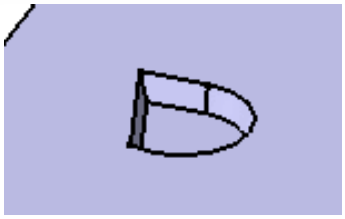
5. Click **OK** to validate.

   The surface stamp (identified as Surface Stamp.xxx) is created and the specification tree is updated accordingly.





You can use 0 as the Radius value to deactivate the Radius R1 value, and to create the surface stamp without a fillet.



Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Bridge

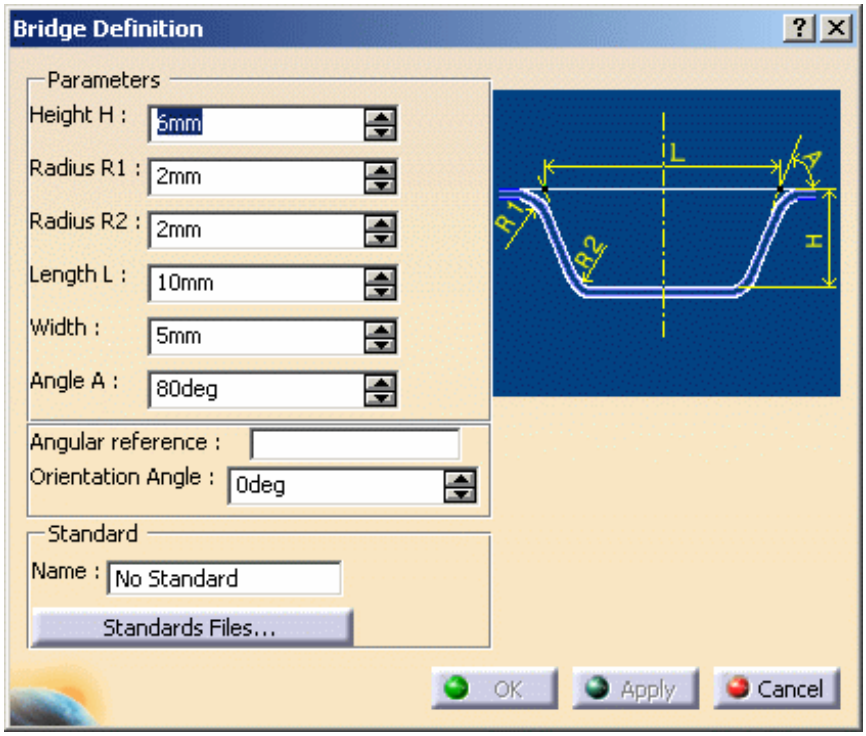
This task shows you how to create a bridge by specifying the punch geometrical parameters.

Open the NEWStamping5.CATPart document from the samples directory.

1. Click the **Bridge** icon .

2. Select a point on the top face where you want to place the bridge.

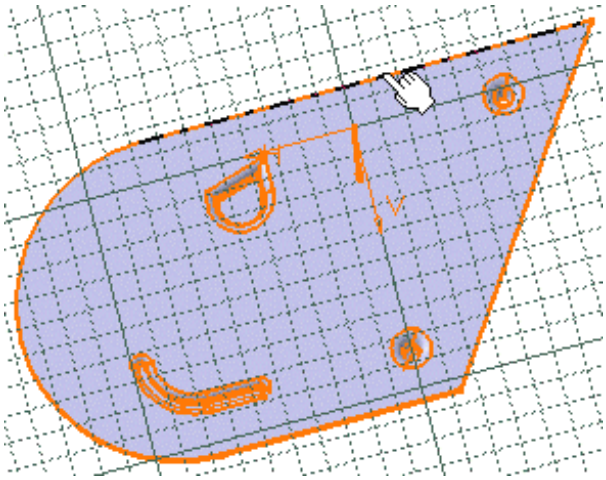
   The Bridge Definition dialog box opens, providing default values.



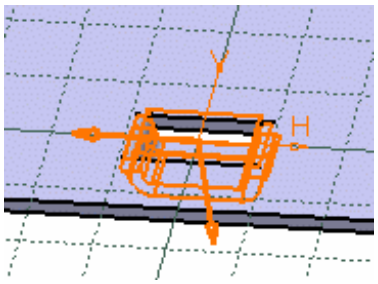
3. Change the value in the different fields, if needed:

   - Height H
   - Radius R1
   - Radius R2
   - Angle A
   - Length L1
   - Length L2

4. Select an edge to give the direction of the bridge.

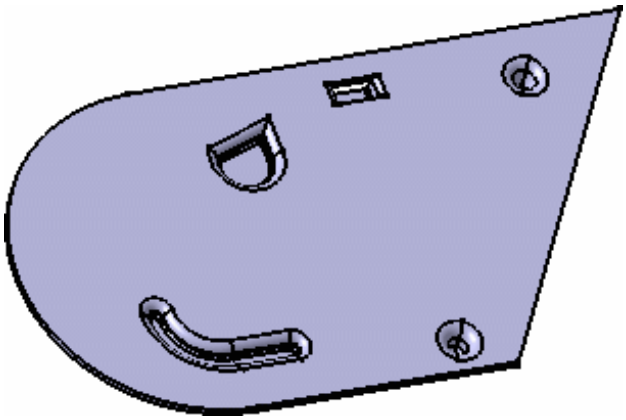


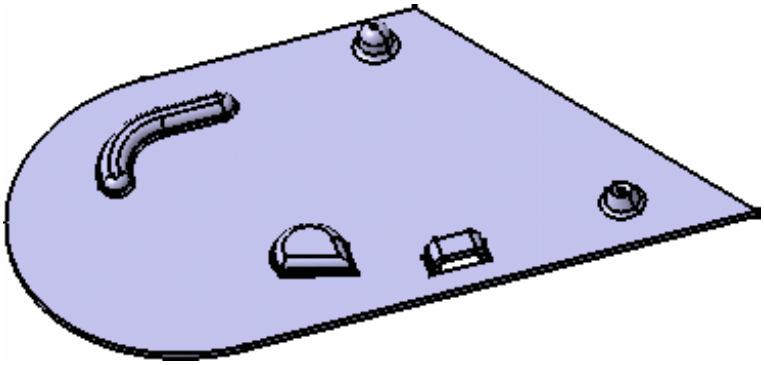
5. Click **Apply** to preview the bridge.



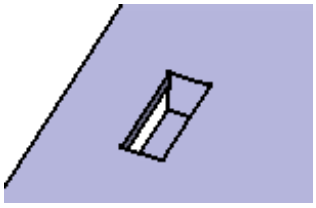
6. Click **OK** to validate.

The bridge (identified as Bridge.xxx) is created and the specification tree is updated accordingly.





You can use 0 as the Radius value to deactivate the Radius R value, and to create the bridge without a fillet.



Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Flanged Cutout

This task shows you how to create a flanged cutout by specifying the punch geometrical parameters.

Open the NEWStamping8.CATPart document.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping8.CATPart document.

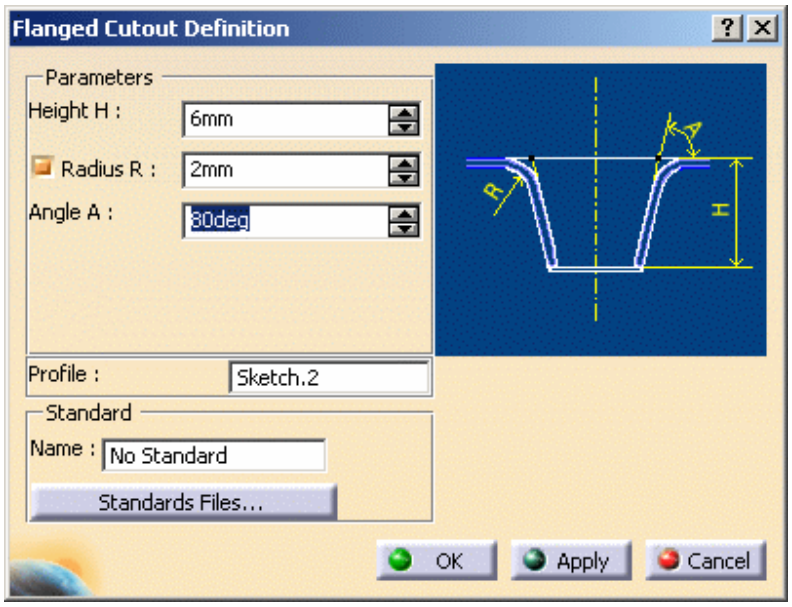
1. Click the **Flanged Cutout** icon .

2. Select a profile.

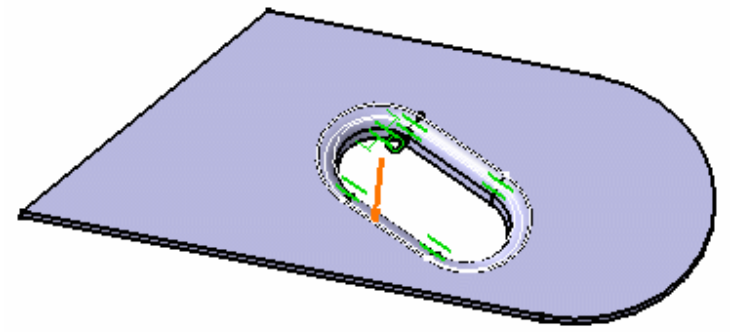
   The Flanged Cutout Definition dialog box is displayed, providing default values.

3. Change the value in the different fields, if needed:

   - Height H
   - Radius R
   - Angle A

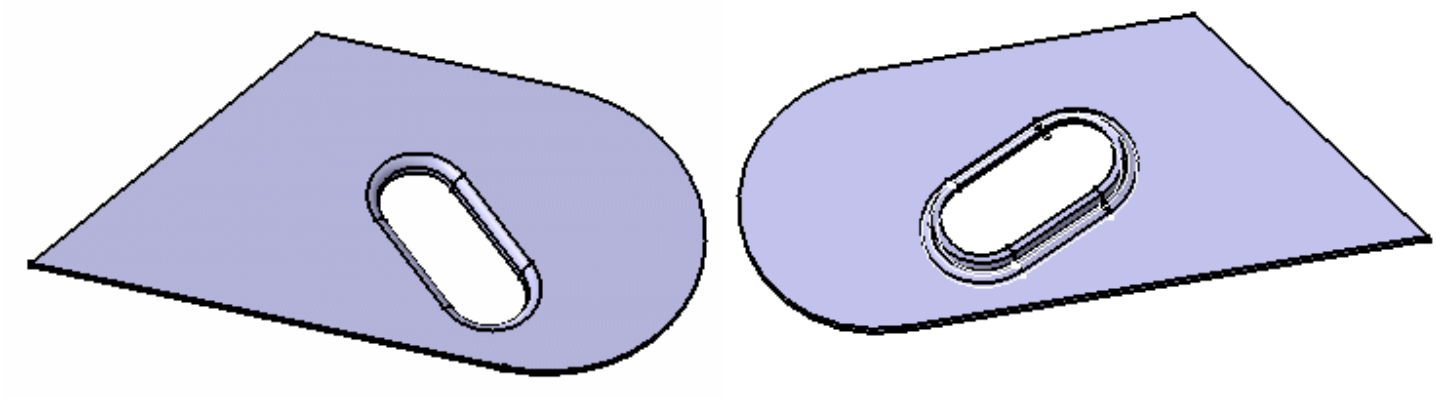


4. Click **Apply** to preview the flanged cutout.

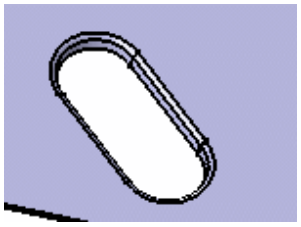


5. Click **OK** to validate.

The flanged cutout (identified as Flanged Cutout.xxx) is created and the specification tree is updated accordingly.



You can use 0 as the Radius value to deactivate the Radius R value, and to create the flanged cutout without a fillet.



Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Stiffening Rib

This task shows you how to create a stiffness rib by specifying the punch geometrical parameters.

Open the NEWStamping7.CATPart document from the samples directory.

If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping7.CATPart document.

1. Click the **Stiffness Rib** icon .

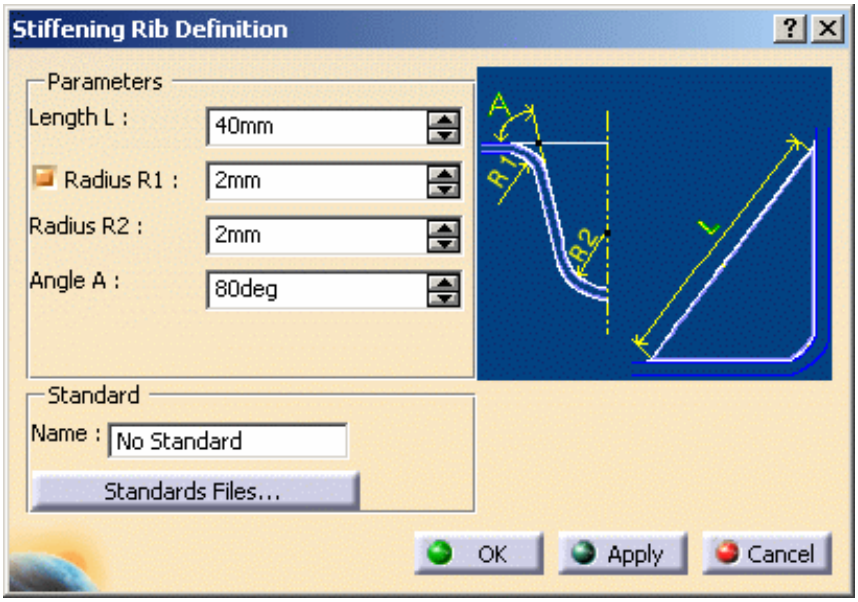
2. Select the external surface of Bend.1, where you want to place a stiffener.
   Note that the stiffener will always be centered on the bend radius, wherever the point may be along the curve.

   A grid is displayed.

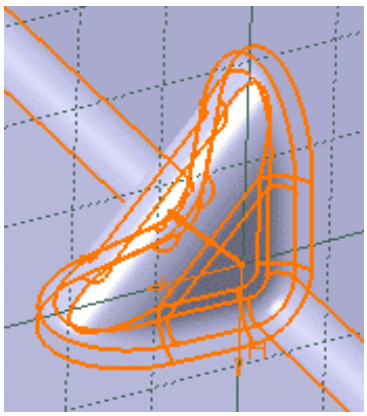
   The Stiffening Rib Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

   - Length L
   - Radius R1
   - Radius R2
   - Angle A

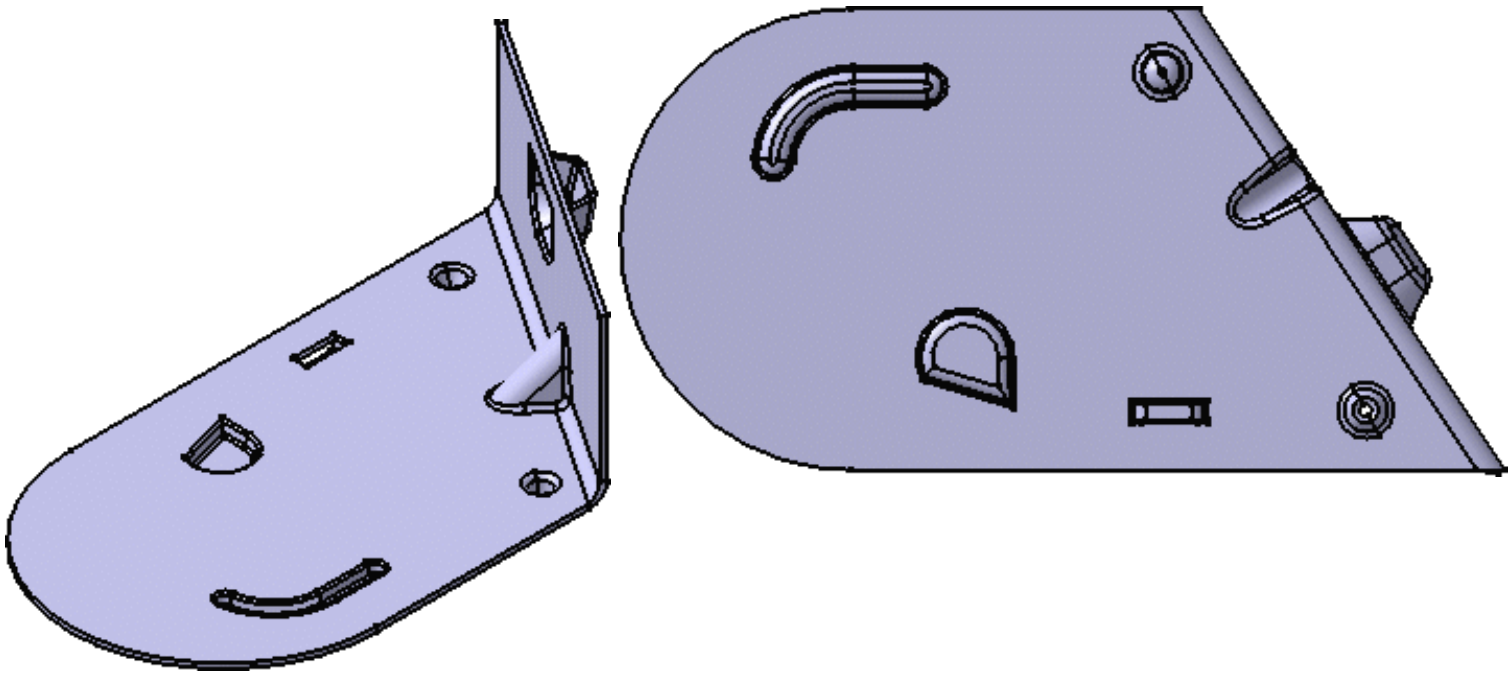


4. Click **Apply** to preview the stiffness rib.

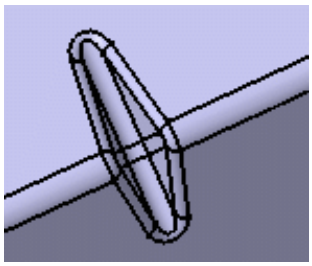


5. Click **OK** to validate.

   The stiffening rib (identified as Stiffnening Rib.xxx) is created and the specification tree is updated accordingly.



You can use 0 as the Radius value to deactivate the Radius R1 value, and to create the stiffening rib without a fillet.



Please refer to the Customizing Standard Files chapter to define the Standards Files.

Creating a Curve Stamp

This task shows you how to create a curve stamp by specifying the punch geometrical parameters.

Open the NEWStamping3.CATPart document.
If you use the Aerospace SheetMetal Design workbench, open the Aero_Stamping3.CATPart document.

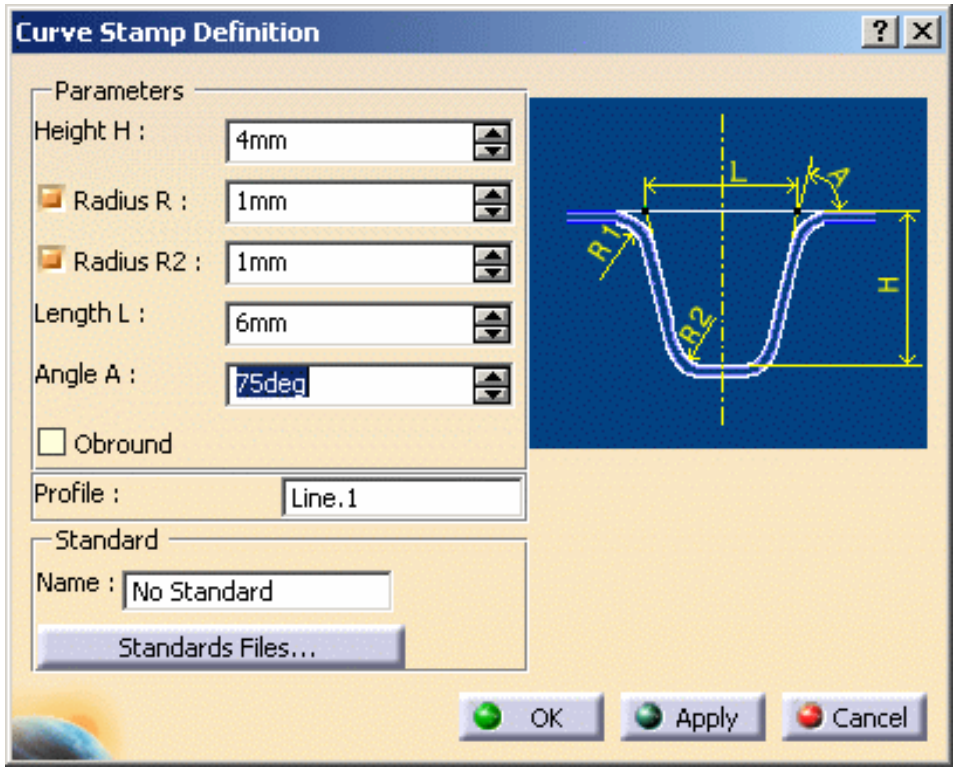
1. Click the **Curve Stamp** icon .

2. Select **Sketch-for-Curve-Stamp**, the curve previously defined.

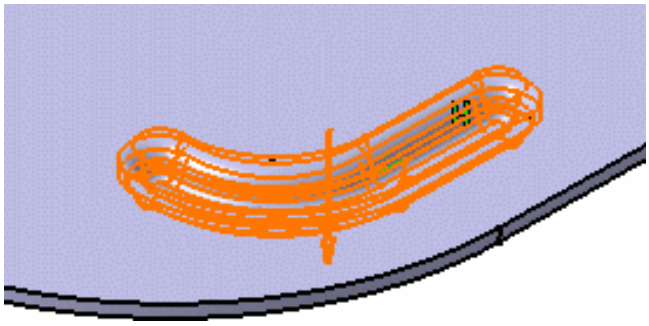
   The Curve Stamp Definition dialog box opens, providing default values.

3. Change the value in the different fields, if needed:

   - Height H: the total height
   - Radius R1: the outer bend radius
   - Radius R2: the inner bend radius
   - Angle A: the stamping draft angle
   - Length L: the stamps' maximum width

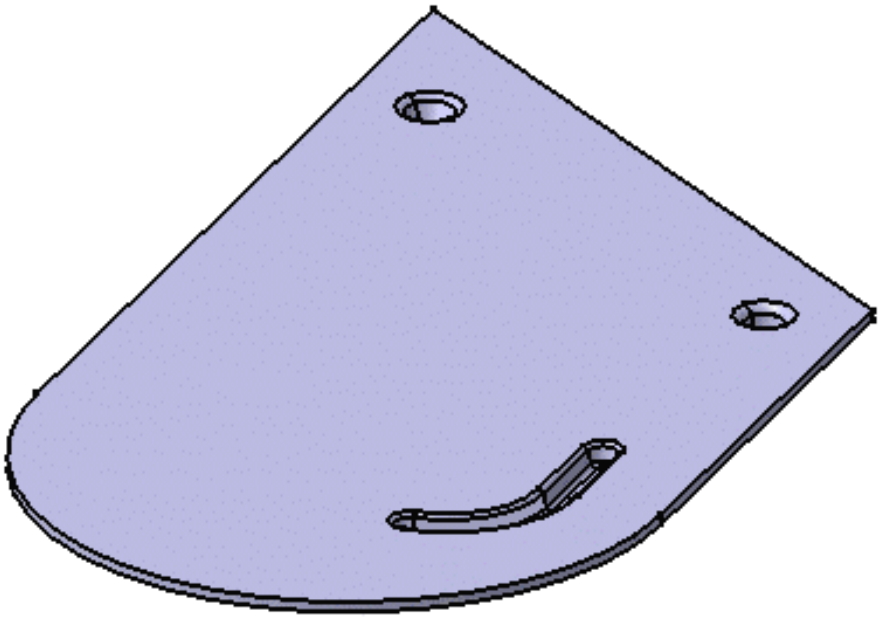


4. Click **Apply** to preview the curve stamp.

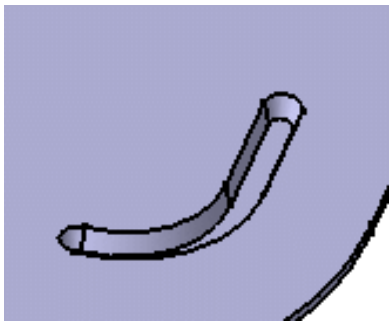


5. Click **OK** to validate.

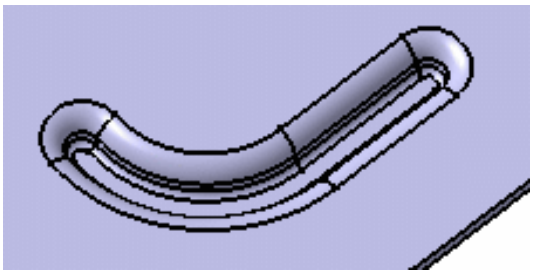
The curve stamp (identified as Curve Stamp.xxx) is created and the specification tree is updated accordingly.



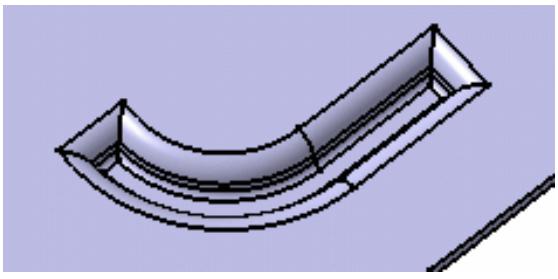
- You can use 0 as the Radius value to deactivate the Radius R and Radius R2 values, and to create the curve stamp without a fillet.



- Check the **Obround** option to round off the edges of the curve stamp.



Obround option checked



Obround option unchecked

Please refer to the Customizing Standard Files chapter to define the Standards Files.



Creating a Louver

This task shows you how to create a louver by specifying the punch geometrical parameters.

Open the NEWStamping11.CATPart document.

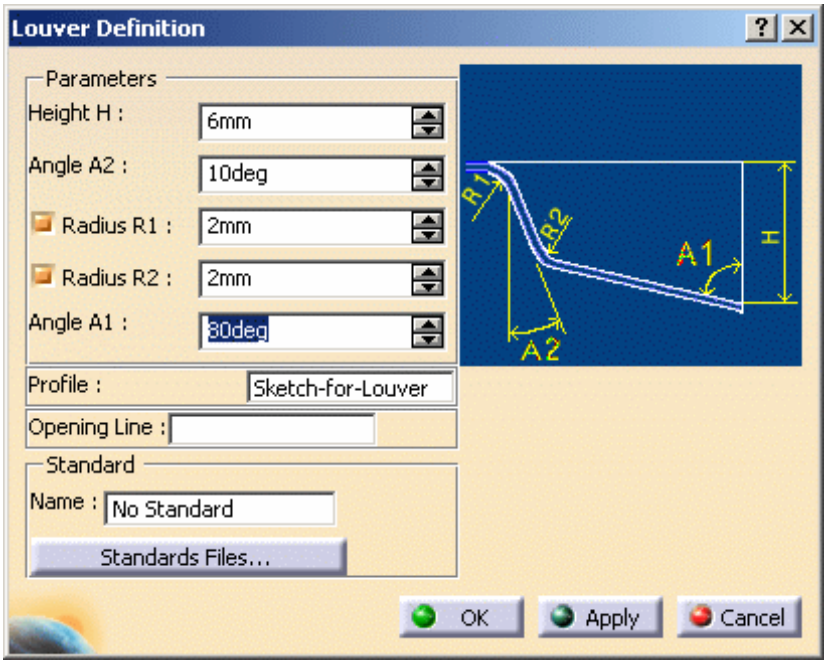
1. Click the **Louver** icon .

2. Select **Sketch-for-Louver**, a profile previously defined on **Wall.2**. The **Louver Definition** dialog box opens, providing default values.

The louver opening face is represented in the sketch by the element that does not present any tangency continuity with the other lines/curve segments of the sketch. In case there are several non-continuous elements, the first one is used as the opening face.

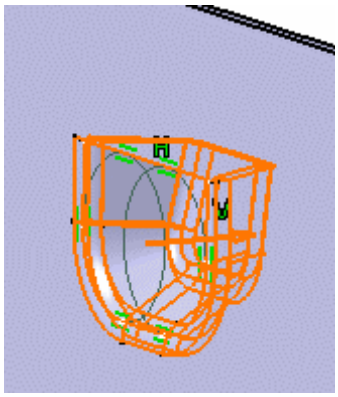
3. Change the value in the different fields, if needed:

   - Height H
   - Radius R1
   - Radius R2
   - Angle A1
   - Angle A2



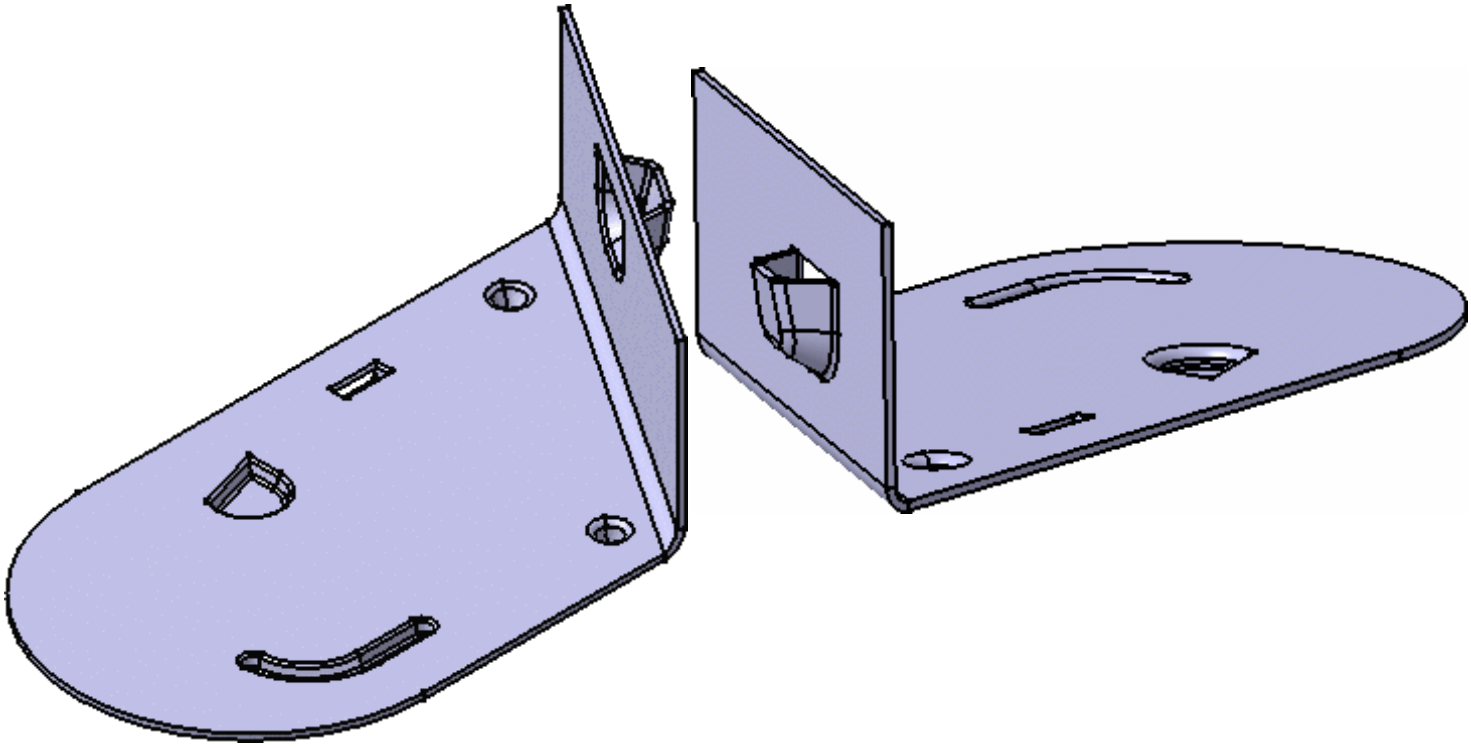
4. Select an edge of the sketch as the **Opening Line**: it indicates the position of the opening face of the louver.

5. Click **Apply** to preview the louver.



6. Click **OK** to validate.

   The louver (identified as Louver.xxx) is created and the specification tree is updated accordingly.



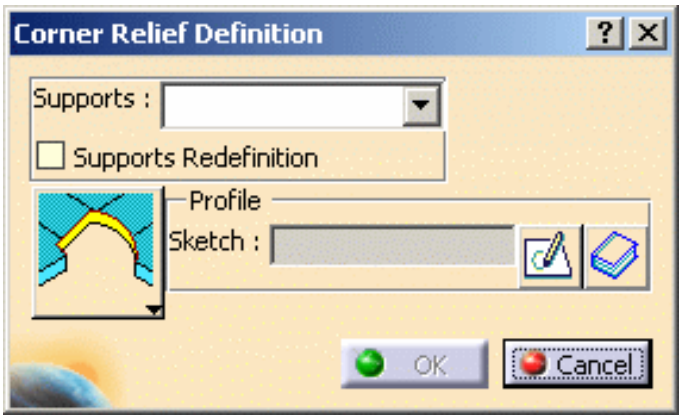
Creating a Local Corner Relief

This task explains how to define a corner relief locally on a set of supports.

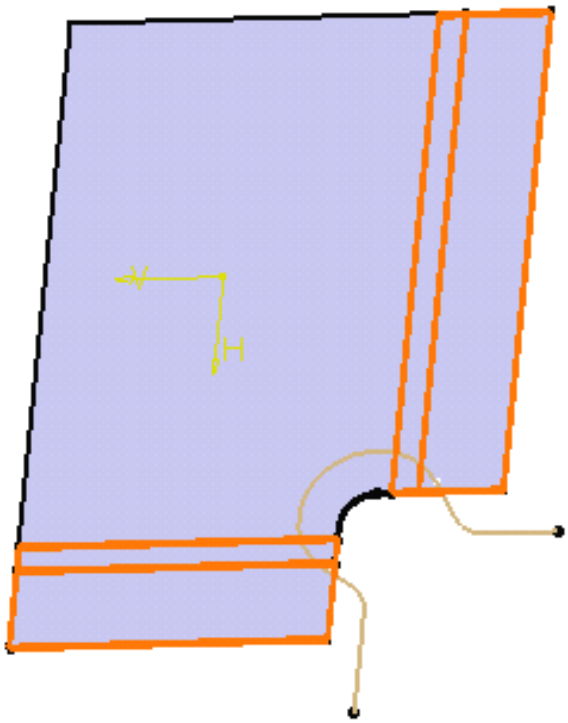
Open the NEWCornerRelief01.CATPart document from the samples directory.
The part needs to be unfolded prior to creating the corner relief.

1. Click the **Corner Relief** icon .

   The Corner Relief Definition dialog box is displayed.



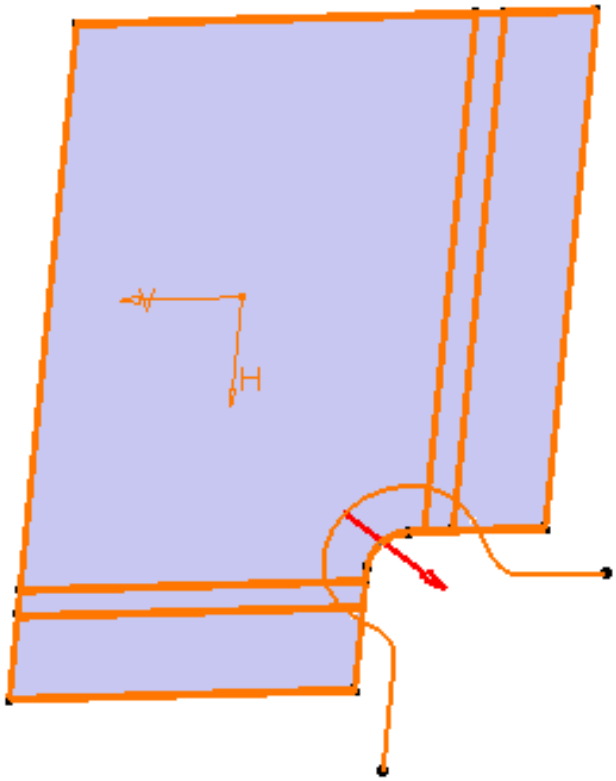
2. Select the supports on which a corner relief should be created (here we chose Flange.1 and Flange.2)



A notch was defined on the web profile between the two fillets' flanges; so that flanges do not intersect. This operation enables to prepare the web as to create the flanges that will be later used to define the corner relief.

- By default the **User Profile** is active in the Corner Relief Definition dialog box.

1. Select the sketch, directly in the 3D geometry.

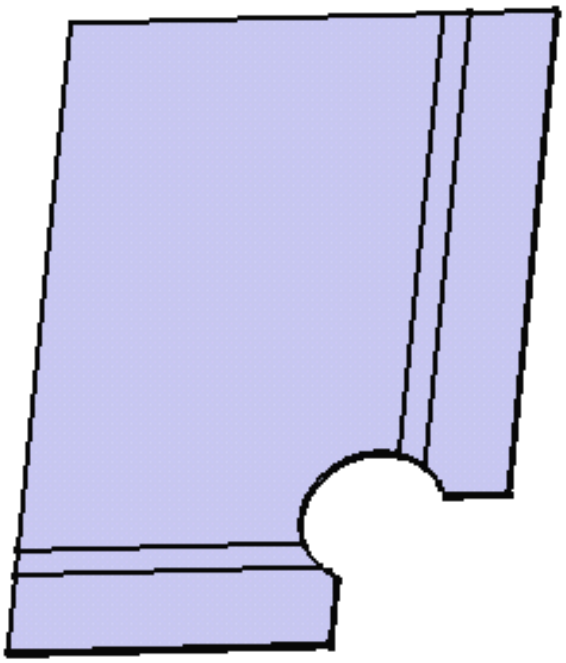


As soon as the sketch has been selected, the **Sketcher** icon is displayed in the dialog box allowing you to edit the selected sketch, if needed.

The red arrow lets you choose the direction of matter to remove. Click it to reverse the direction.

The sketch must be continuous in tangency.

2. Click OK in the Corner Relief Definition dialog box.



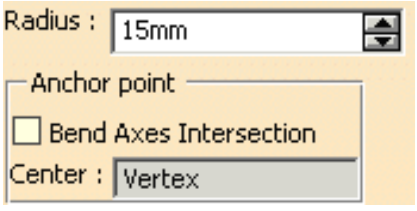
You can use the **Catalog** icon to open the Catalog Browser.

For more information on catalogs, please refer to the Using Catalogs chapter in the *CATIA Infrastructure User Guide*.

- Select the **Circular Profile**



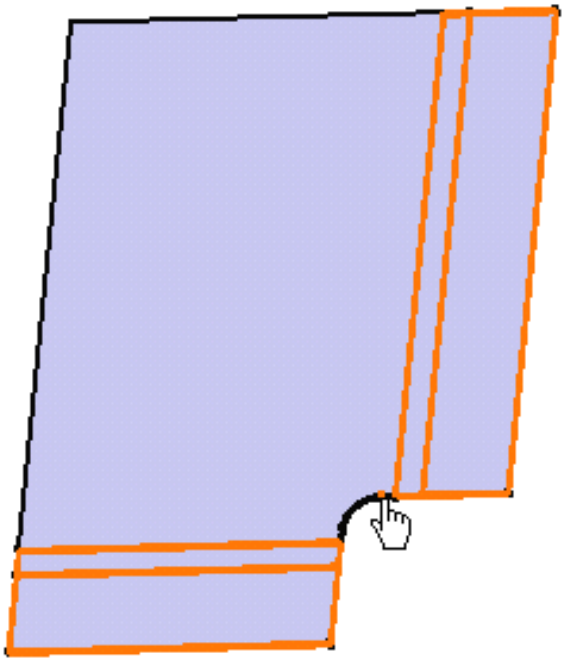
using the down arrow.



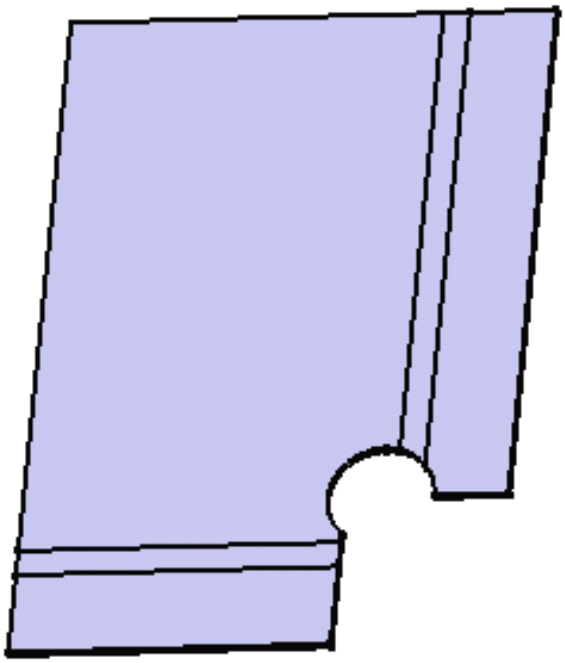
1. Define the default radius: it is equal to the bend radius + the thickness.
   In our example, we defined a radius of 15 mm.

By default the corner relief center is located at the intersection of the bend axes. You can select a point as the circle's center.

2. Select the vertex between the two flanges: it will be the center of the corner relief.

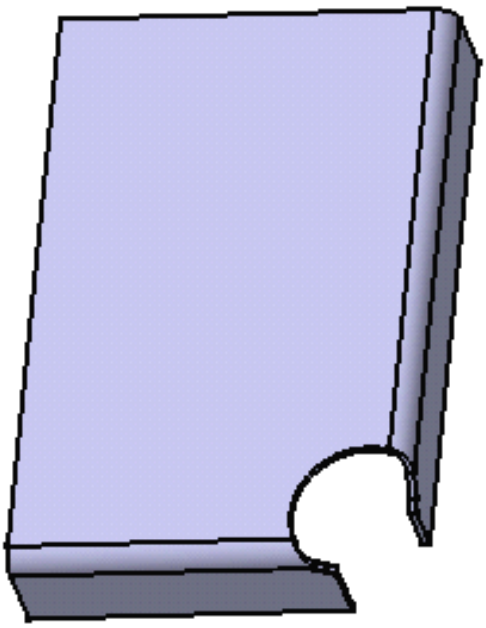


3. Click OK in the Corner Relief Definition dialog box.

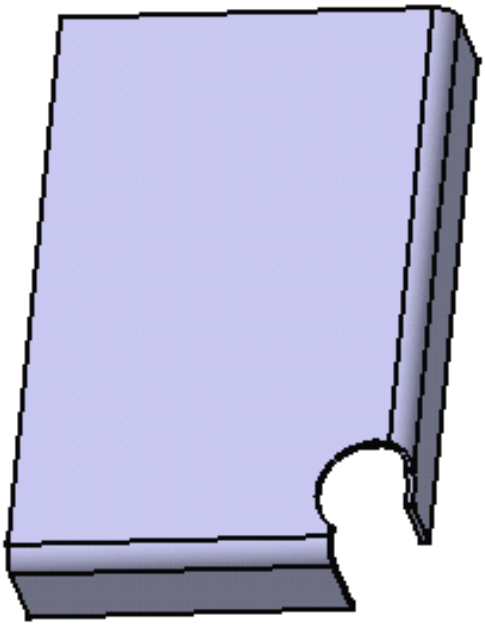


The created element (identified as Corner Relief.xxx) is added to the specification tree.

3. Fold the part to check the corner relief in 3D.

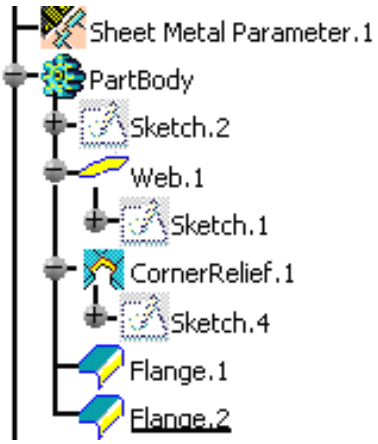


Folded user corner relief

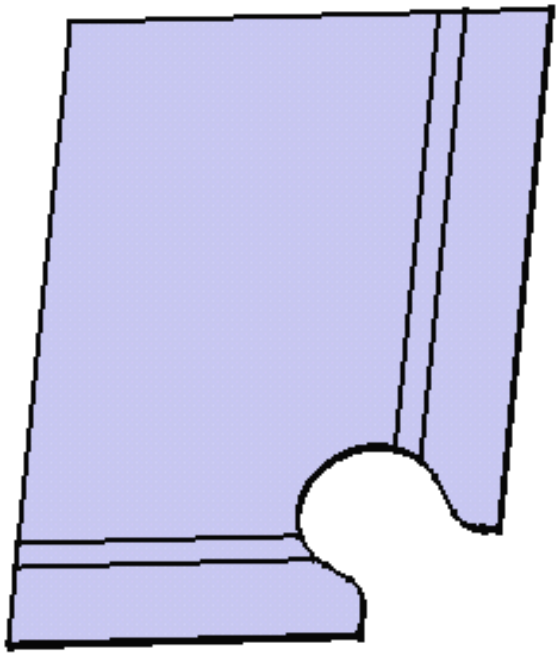


Folded circular corner relief

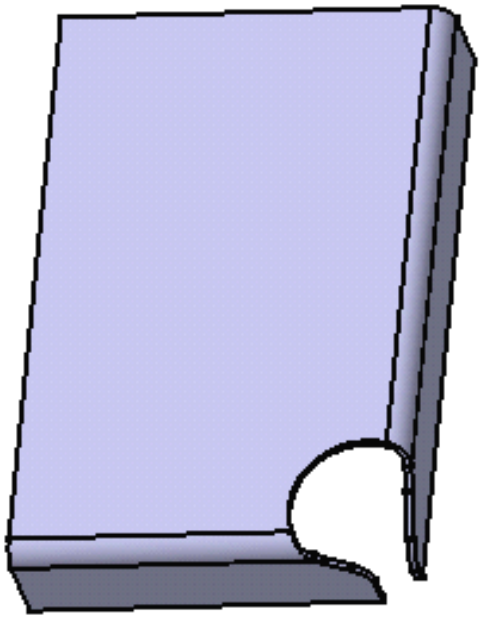
The **Supports Redefinition** button enables to redefine the supports' sides thus adding matter to these supports.
In that case, the created element (identified as Corner Relief.xxx) appears before the supports in the specification tree.



Please note that checking this button means that the corner relief replaces the flange's side. This side must therefore exists: when creating the flange, do not define the side as **None**.



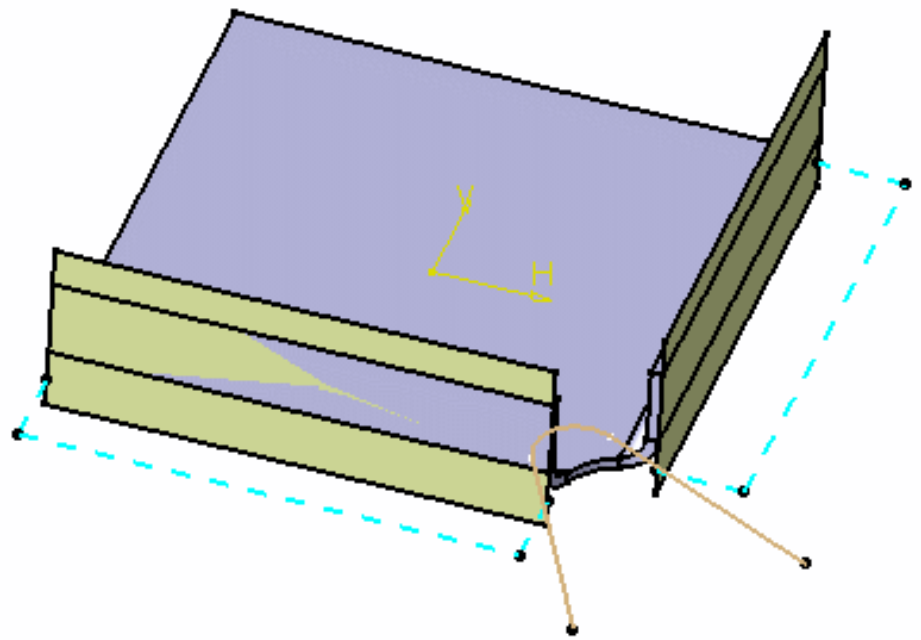
Unfolded user corner relief with redefined supports



Folded user corner relief with redefined supports

The image besides shows two flanges creating with Angle as support type.
The two blue dotted lines represent the limits of the unfolded flanges.

The creation of a corner relief with supports redefined will not be created as it is not be located within the limits of the unfolded flanges.

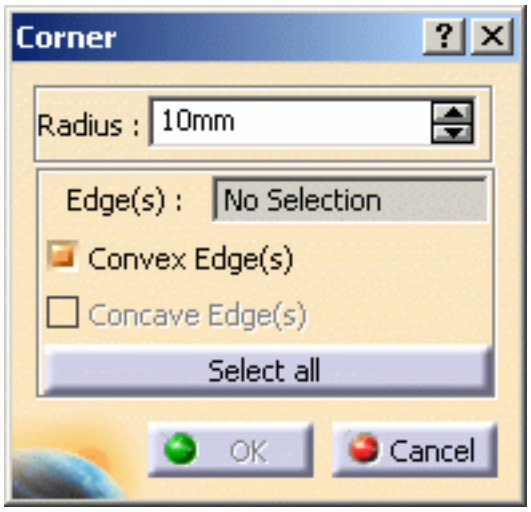


Creating Corners

This task shows how to create one or more corner(s) on a Sheet Metal part, that is to round off sharp edges, much like a fillet between two faces of a Part Design Body. This corner creation operation can be performed indifferently on the folded or unfolded view, and only one support (i.e. the corner when previewed should not lie over two supports).

Open the Corner1.CATPart document.

If you use Aerospace SheetMetal Design, open the Corner_Aero1.CATPart document.



1. Click the **Corner** icon .

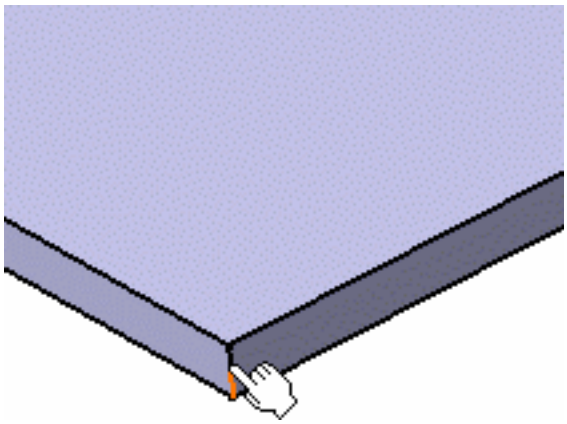
   The Corner Definition dialog box is displayed.

2. Set the radius value.

3. Choose the type of edge you wish to round off:

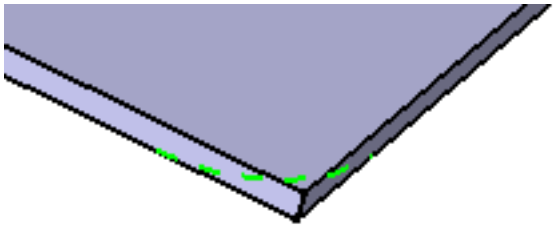
- using the **Select All** button: all convex edges, or all edges of both types
- any edge manually selected

Once you have selected an edge, you can no longer modify this option, unless you cancel the selection.



4. Select a sharp edge on a part.

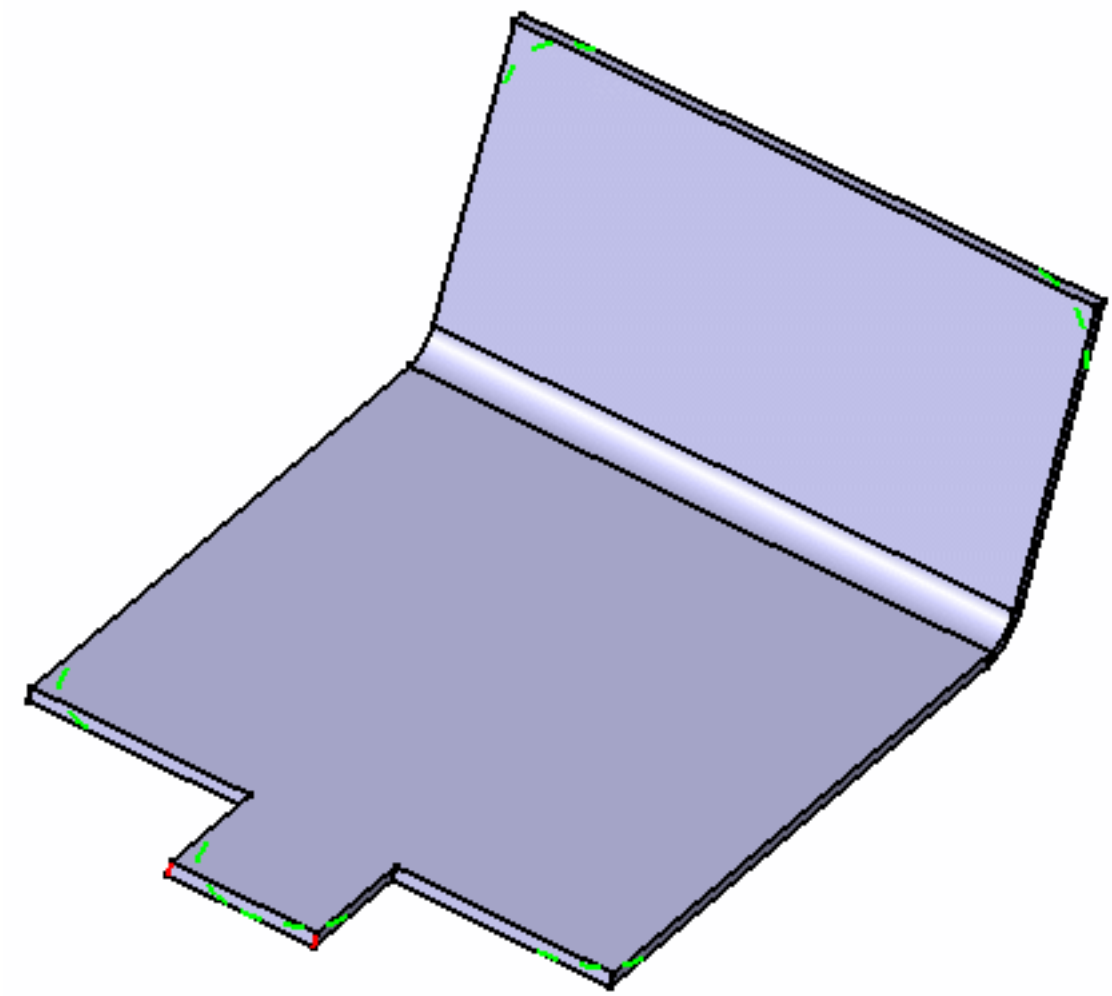
   As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.



The corner is previewed on the edge, with the current radius value.

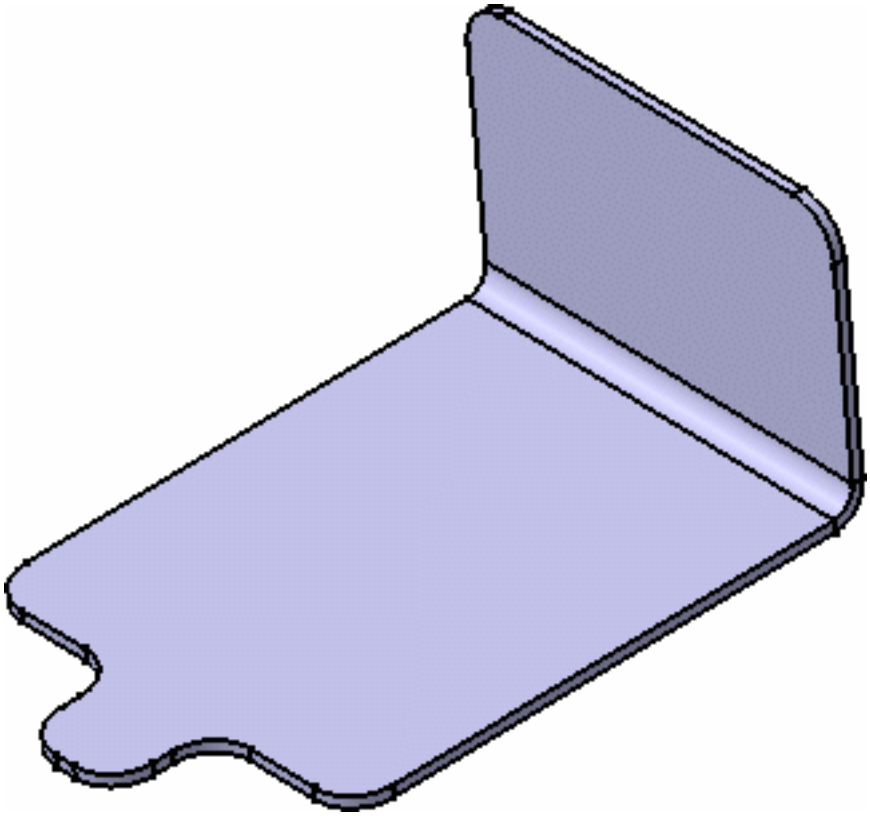
5. Click **Cancel Selection** then click the **Select All** button.

   All sharp edges of the part are selected and the corners previewed.



6. Click OK in the dialog box.

All sharp edges of the part are rounded off to create smooth corners.



To deselect an edge, simply click it again. For quick selection in a complex part, you can select all edges with the **Select All** check button, then deselect one or two edges.

- When you select an edge that is not sharp, such as the edge between a wall and a bend for example, a warning is issued.
- As you select more edges, the **Edge(s)** field of the dialog box is updated.
- When using the **Select All** button, you select all edges present at the time. If when modifying the part, new edges are created, these will not be automatically rounded off.

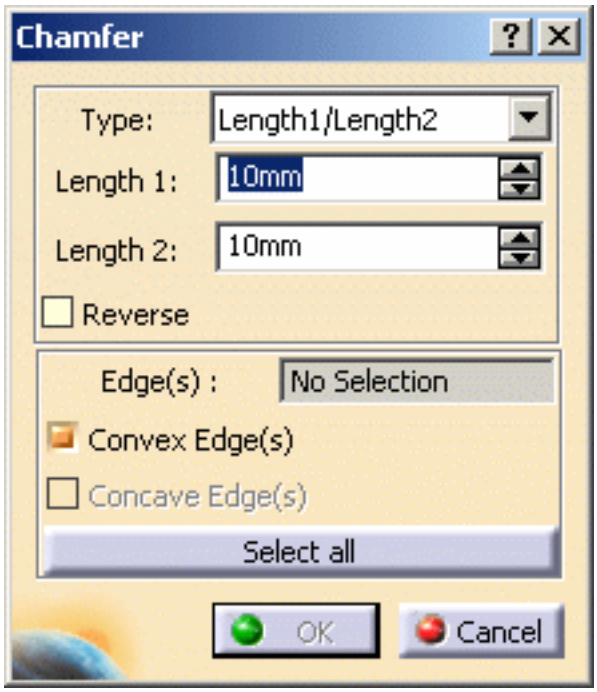
Creating Chamfers

This task shows how to create one or more chamfer(s) on a Sheet Metal part, that is to cut off, or fill in sharp edges of Sheet Metal parts.
This chamfer creation operation can be performed indifferently on the folded or unfolded view, and only one support (i.e. the chamfer when previewed should not lie over two supports).

Open the Corner1.CATPart document.
If you use Aerospace SheetMetal Design, open the Corner_Aero1.CATPart document.

1. Click the **Chamfer** icon .

   The Chamfer Definition dialog box is displayed.



You can choose the type of edge you wish to chamfer:

- using the **Select All** button: all edges
- any edge manually selected

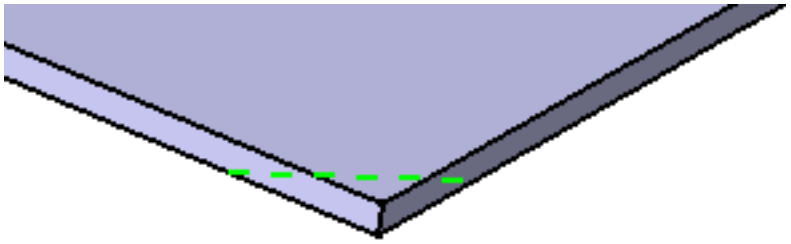
3. Select a sharp edge on a part.

   The chamfer is previewed on the edge.

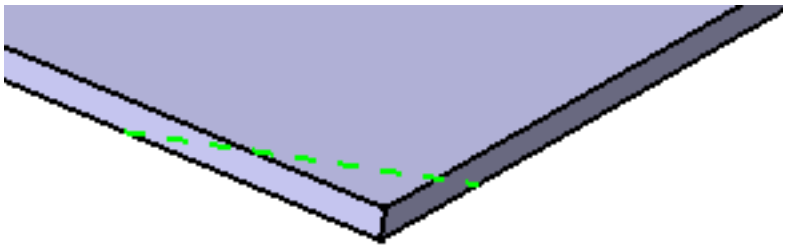
As soon as you selected one edge, the dialog box is updated and the **Select All** button changes to **Cancel Selection**.

4. Choose a chamfer **Mode**. You can either enter:

- two lengths: these lengths are computed from the selected edge on both sides.
  Here, we chose two lengths of 10mm.



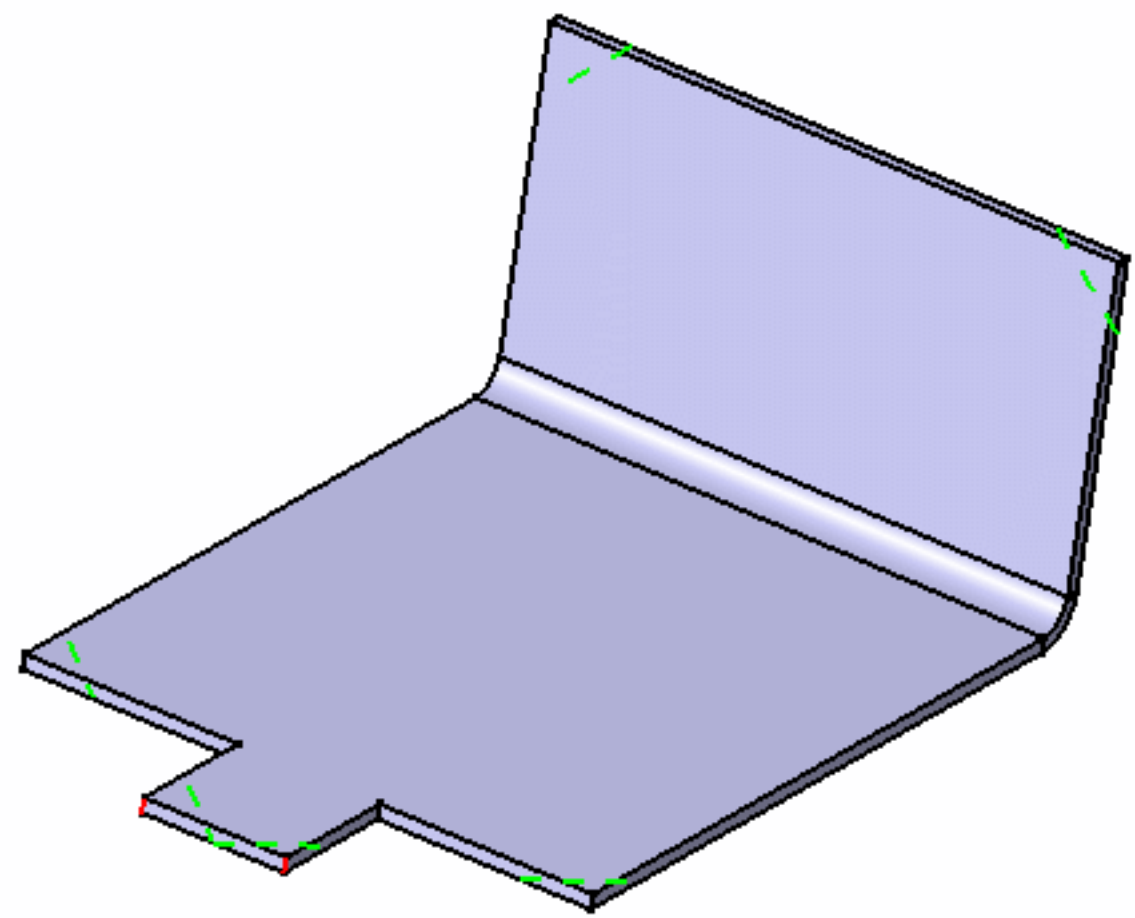
- a length value and an angle: the length is computed on one side of the edge and the angle from the chamfer's limit on the same side.
  Here, we chose a length of 10mm and an angle of 60deg.



You can use the **Reverse** button to inverse all edges' side, on which the values are taken into account.

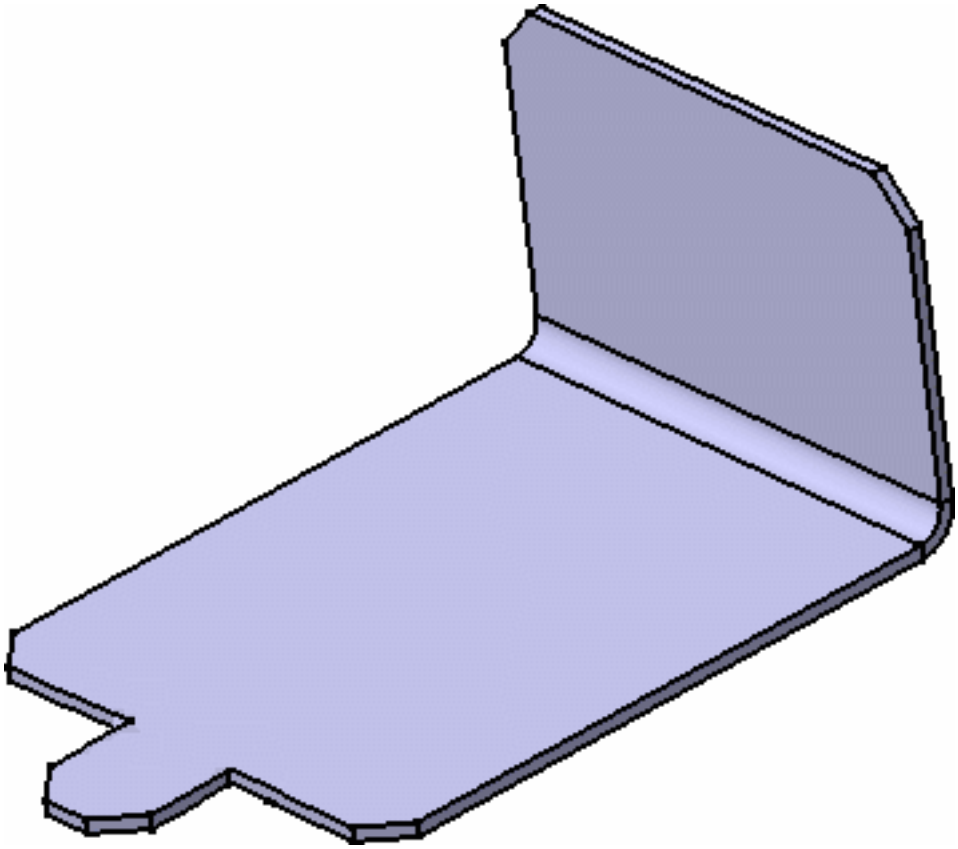
5. Click **Cancel Selection** then click the **Select All** button.

   All sharp edges of the part are selected, the **Select All** button taking into account the chosen type and the chamfers previewed.



6. Click OK in the dialog box.

   All sharp edges of the part are cut off or filled in.



To deselect an edge, simply click it again. For quick selection in a complex part, you can select all edges with the **Select All** button, then deselect one or two edges.

- When you select an edge that is not sharp, such as the edge between a wall and a bend for example, a warning is issued.
- As you select more edges, the **Edge(s)** field of the dialog box is updated.
- When using the **Select All** button, you select all edges present at the time. If when modifying the part, new edges are created, these will not be automatically chamfered.

Mapping Elements

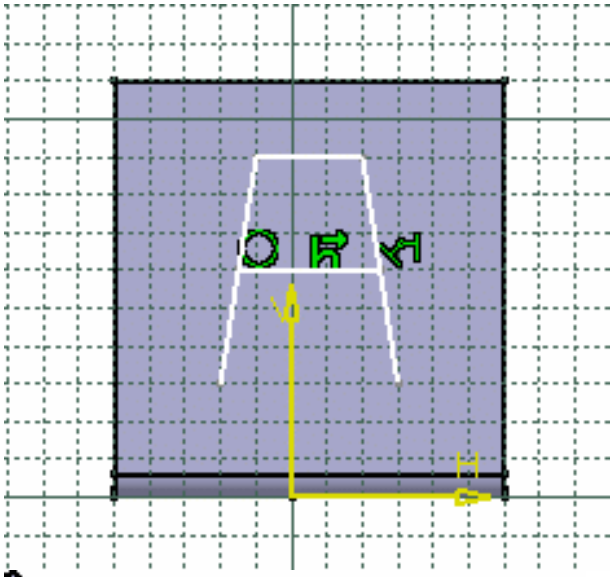


This task shows how to create curves or points from a sketch (as designed using the Sketcher) or from existing curves or points, onto a Sheet Metal part; and to fold/unfold it, just as other Sheet Metal elements.
This is especially useful when:

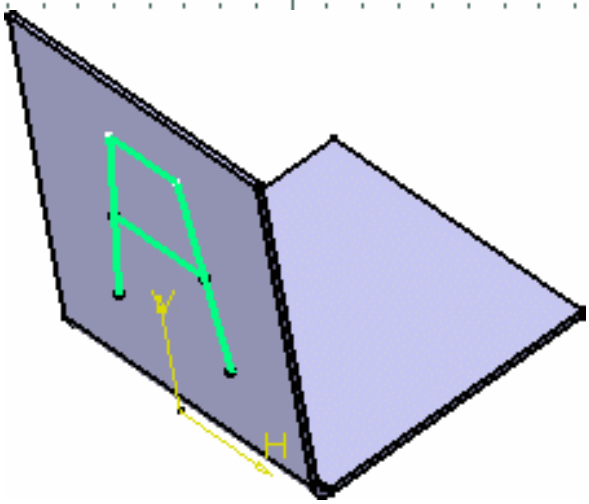
- you want to generate a logotype
- you want to define an area for chemical milling
- you want to create a cutout (pocket) to solve the overlapping of walls for example (the overlapping can be checked with the Sheet Metal Production product).

Open the Mapping1.CATPart document.
If you use the New SheetMetal Design or the Aerospace SheetMetal Design, open the NEWMapping1.CATPart document.

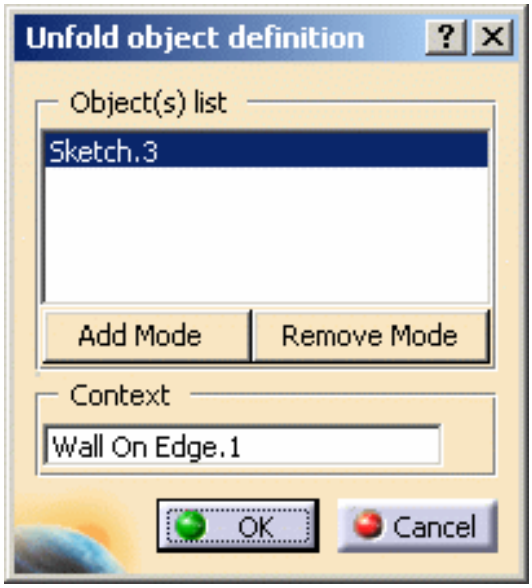
1. Click the **Sketcher** icon, select the wall onto which the curve should lie, and draw the sketch you wish.
   This is the sketch that will be mapped onto the part.
2. Exit the Sketcher.



The 3D part looks like this:



3. Make sure the sketch is selected, and click the **Fold / Unfold Points or Curves** icon .



The Elements To Map definition dialog box is displayed.
It indicates which elements have been selected for mapping.

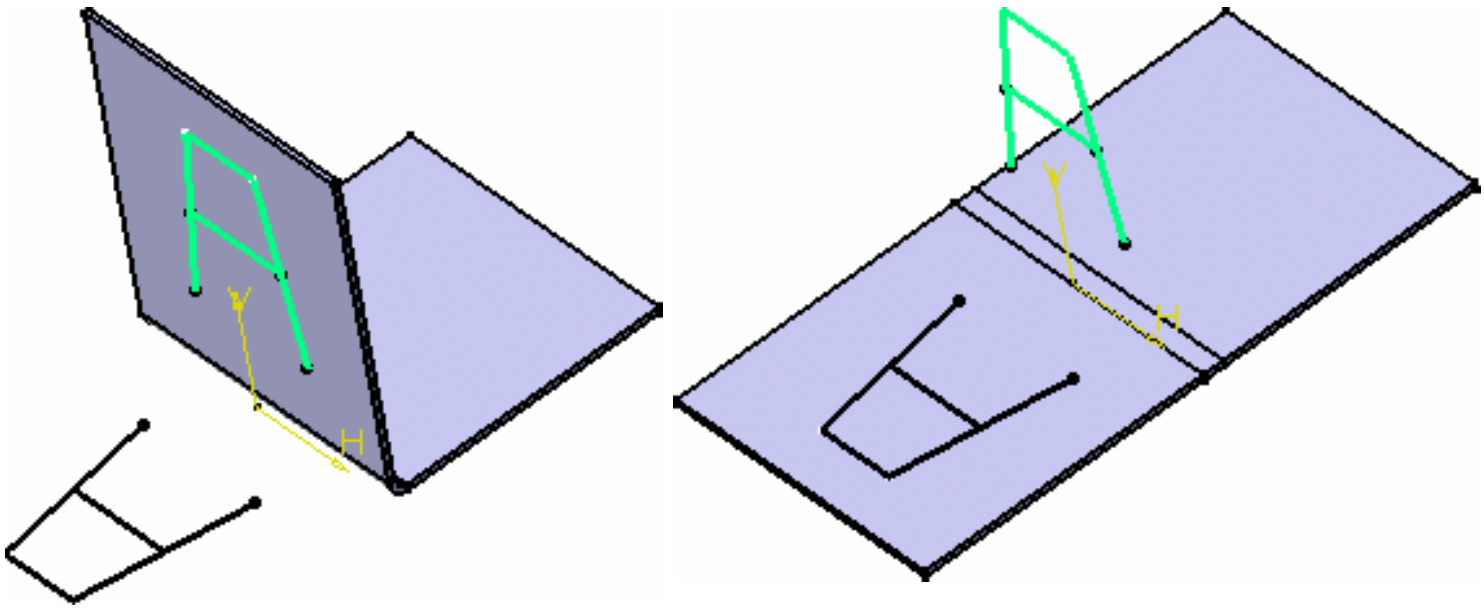
You can manage the list of elements:

- to remove an element, select it from the list and use the **Clear selection** contextual menu
- to add an element, select it directly in the geometry.
  Order in the list does not matter.

4. Select the **Mapping Context**, that is the element of the part on which the curve should be generated when folding or unfolding.

The **Mapping Context** is not necessarily the support element on which the element to be mapped has been drawn. Indeed, by default, the **Mapping Context** is the last Sheet Metal feature that has been created or modified, that is the current feature in the specification tree.

5. Click OK.
   A curve is created and added in the specification tree.



Folded view of the curve mapping *Unfolded view of the curve mapping*

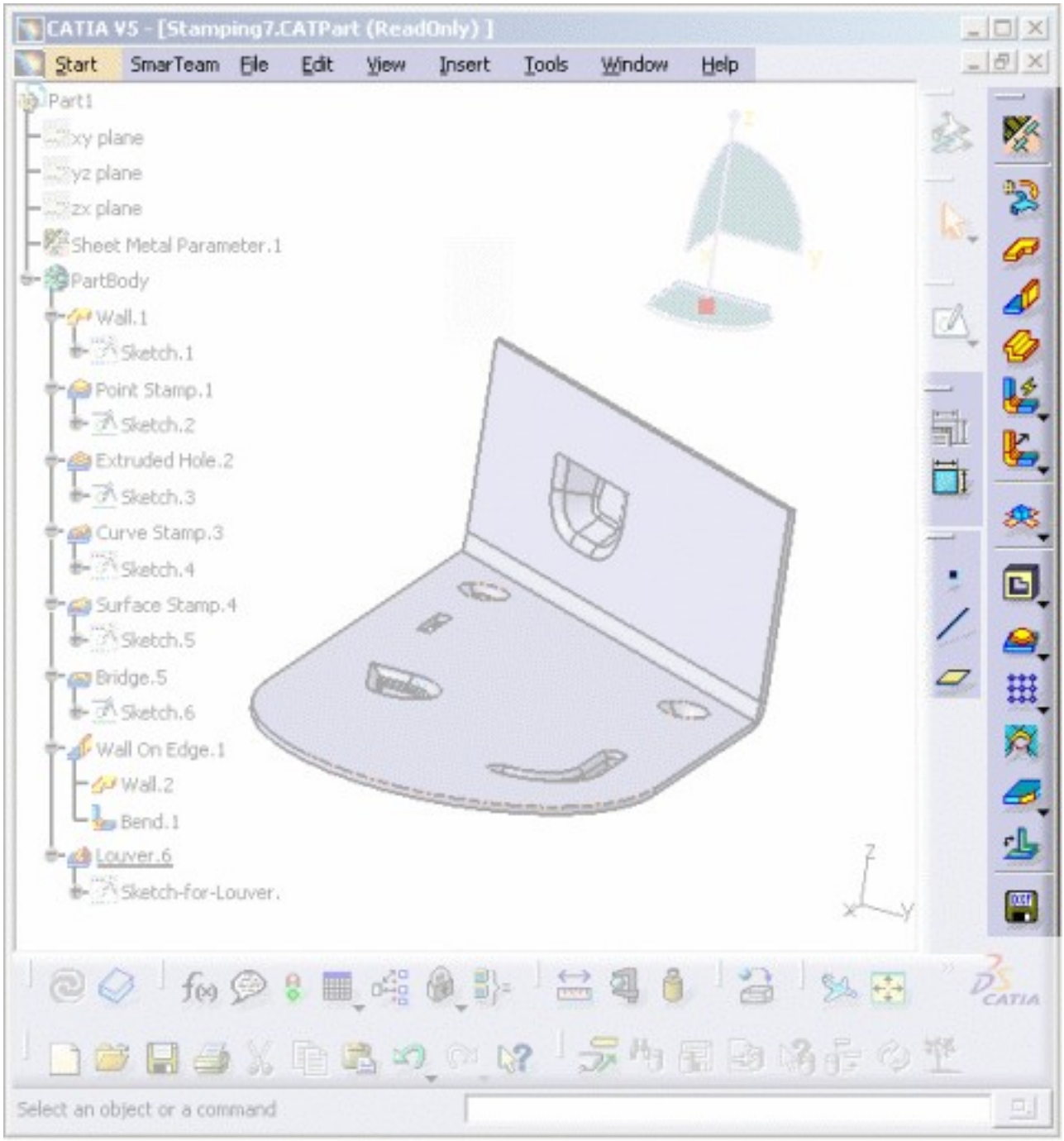
- You can select several sketches/curves/points to be mapped at a time.
- Mapped curves can be created across several walls and bends.

Workbench Description

Sheet Metal Design
NEW Sheet Metal Design

Sheet Metal Design

The Sheet Metal Design application window looks like this:
Click the hotspots to display the related documentation.



Menu Bar
Sheet Metal Toolbar
Constraints Toolbar
Reference Elements Toolbar

Specification Tree

Menu Bar

The various menus and menu commands that are specific to Sheet Metal Design are described below.

Start	File	Edit	View	Insert	Tools	Windows	Help

Tasks corresponding to general menu commands are described in the *Infrastructure User's Guide*. Refer to the Menu Bar section.

Insert

For...	See...
Sketcher...	Refer to Sketching in the *Sketcher User's Guide*.
Sheet Metal Parameters...	Managing the Default Parameters
Walls Recognition...	Creating Walls from an Existing Part
Wall...	Creating Walls from a Sketch
Wall on Edge...	Creating Walls from an Edge
Extrusion...	Extruding
Bends	Insert -> Bends
Swept Walls	Insert -> Swept Walls
Unfold	Insert -> Unfold
ShePocket	Insert -> ShePocket
Stampings	Insert -> Stampings
Patterns	Insert -> Patterns
CornerRelief...	Creating a Local Corner Relief
Corners	Insert -> Corners



Mapping...	Mapping Curves
Save As DXF...	Saving As DXF
Constraints	Setting Constraints in the *Part Design User's Guide*
Advanced Replication Tools	Insert -> Replication Tools
Instantiate From Document...	Instantiating PowerCopy Features

Insert -> Bends



For...	See...
Automatic Bends	Generating Bends Automatically
Bend	Creating Bends From Walls
Conic Bend	Creating Conical Bends
Bend	Generating a Bend from a Line

Insert -> Swept Walls



For...	See...
Flange	Creating a Flange
Hem	Creating a Hem
Tear Drop	Creating a Tear Drop
Swept Flange	Creating a Swept Flange

Insert -> Unfold



For...	See...
Unfold	3D View
MultiView	Concurrent Access

Insert -> ShePocket



For...	See...
Cutout...	Creating a Cutout
Split...	Splitting Geometry

Insert -> Stampings



For...	See...
Point Stamp	Creating a Point Stamp
Extruded Hole	Creating an Extruded Hole
Curve Stamp	Creating a Curve Stamp
Surface Stamp	Creating a Surface Stamp
Bridge	Creating a Bridge
Louver	Creating a Louver
Stiffening Rib	Creating a Stiffening Rib
User Stamping	Creating User-Defined Features

Insert -> Patterns

For...	See...
Rectangular Pattern	Creating Rectangular Patterns



Circular Pattern	Creating Circular Patterns
User-Defined Pattern	Creating User-Defined Patterns

Insert -> Corners



For...	See...
Corner	Creating Corners
Chamfer	Creating Chamfers

Insert -> Replication Tools



For...	See...
PowerCopy Creation	Creating PowerCopy Features
PowerCopy Save in Catalog	Saving PowerCopy Features

Up Next

Sheet Metal Toolbar



 See Managing Default Parameters

 See Creating Walls from an Existing Part

 See Creating Walls from a Sketch

 See Creating Walls from an Edge

 See Extruding

 See Creating Bends



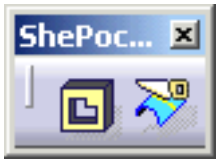
 See Creating Swept Walls



 See Unfolding the Part



 See Pockets



 See Stamping



 See Patterning



 See Creating a Local Corner Relief

 See Creating Corners

 See Creating Chamfers

 See Curve Mapping

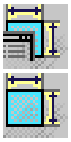


 See Save As DXF

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Constraints Toolbar



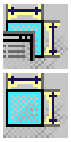


See Setting Constraints from the *Part Design User's Guide*

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Constraints Toolbar





See Setting Constraints from the *Part Design User's Guide*

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Reference Elements Toolbar



 See Creating Points

 See Creating Lines

 See Creating Planes

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Specification Tree

Within the Sheet Metal Design workbench, you can generate a number of features that are identified in the specification tree by the following icons.

Further information on general symbols in the specification tree are available in Symbols Used in the Specification Tree.

 Sheet Metal Parameters

 Wall

 Wall On Edge

 Extrude

 Automatic Bends

 Bend

 Conical Bend

 Flat Bend

 Flange

 Hem

 Tear Drop

 User-defined Flange

 Cutout

 Split

 Point Stamp

 Extruded Hole

 Curve Stamp

 Surface Stamp

 Bridge

 Rectangular Pattern

 Circular Pattern

 User-Defined Pattern

 Corner Relief

 Corner

 Chamfer

 Mapping

 Point

 Line

 Plane

 Louver

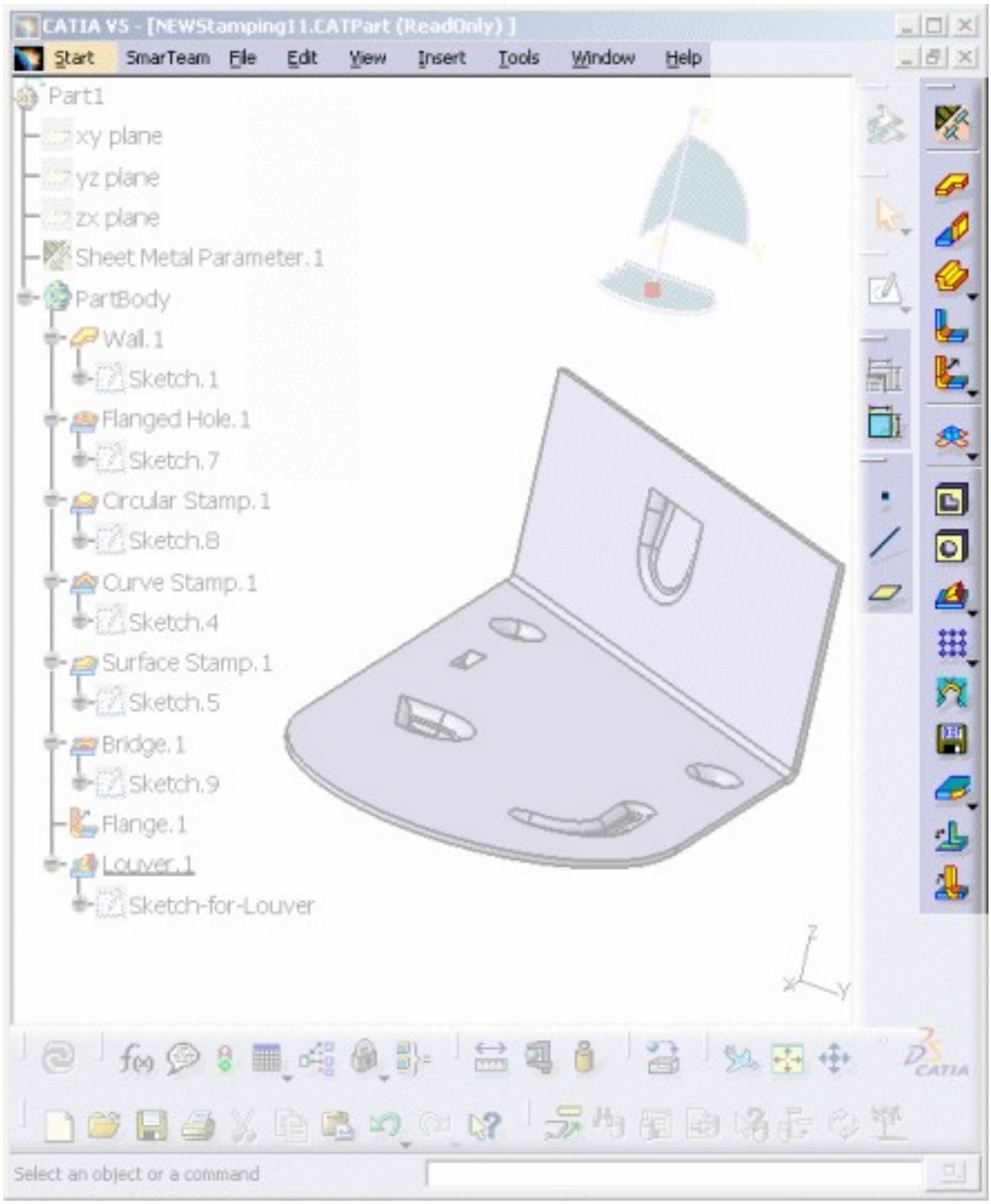
 Stiffening Rib

 User-Defined Stamp

Back Up

NEW Sheet Metal Design

The NEW Sheet Metal Design application window looks like this:
Click the hotspots to display the related documentation.



Menu Bar
NEW Sheet Metal Toolbar

Constraints Toolbar
Reference Elements Toolbar
Specification Tree

Menu Bar

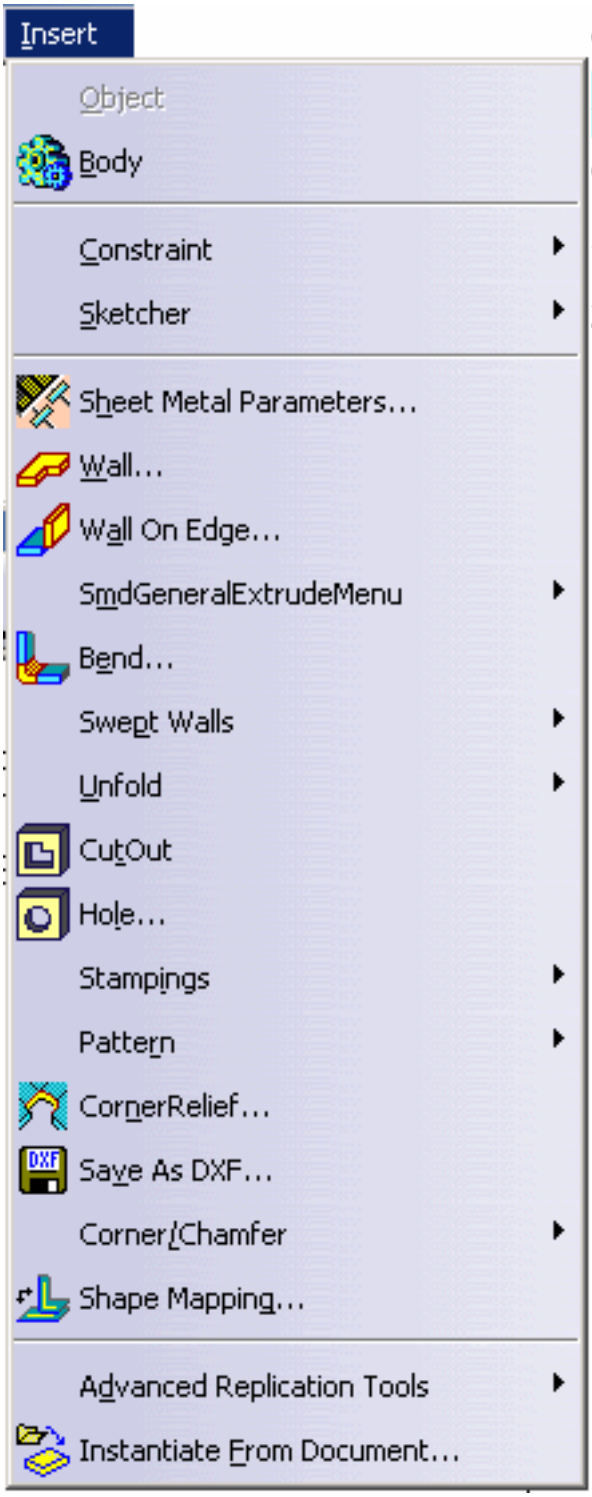
The various menus and menu commands that are specific to Sheet Metal Design are described below.

Start	File	Edit	View	Insert	Tools	Windows	Help

Tasks corresponding to general menu commands are described in the *Infrastructure User's Guide*. Refer to the Menu Bar section.

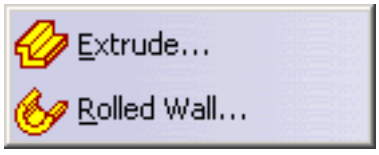
Insert

For...	See...
Constraints	Setting Constraints in the *Part Design User's Guide*
Sketcher	Refer to Sketching in the *Sketcher User's Guide*.
Sheet Metal Parameters...	Managing the Default Parameters
Wall...	Creating Walls from a Sketch
Wall on Edge...	Creating Walls from an Edge
SmdGeneralExtrudeMenu	Insert -> Extrudes
Bend...	Creating Bends From a Line
Swept Walls	Insert -> Swept Walls
Unfold	Insert -> Unfold
CutOut	Creating a Cutout
Hole...	Creating a Hole
Stampings	Insert -> Stampings
Patterns	Insert -> Patterns



CornerRelief...	Creating a Local Corner Relief
Save As DXF...	Saving As DXF
Corner/Chamfer	Insert -> Corners
Shape Mapping...	Mapping Elements
Advanced Replication Tools	Insert -> Replication Tools
Instantiate From Document...	Instantiating PowerCopy Features

Insert -> Extrudes



For...	See...
Extrude...	Extruding
Rolled Wall...	Creating Rolled Walls

Insert -> Swept Walls

For...	See...



Flange	Creating a Flange
Hem	Creating a Hem
Tear Drop	Creating a Tear Drop
Swept Flange	Creating a Swept Flange

Insert -> Unfold



For...	See...
Unfold	3D View
MultiView	Concurrent Access

Insert -> Stampings



For...	See...
Flanged Hole...	Creating a Flanged Hole
Bead...	Creating a Bead
Circular Stamp...	Creating a Circular Stamp
Surface Stamp...	Creating a Surface Stamp
Bridge...	Creating a Bridge
Flanged CutOut...	Creating a Flanged Cutout
Stiffening Rib...	Creating a Stiffening Rib
Curve Stamp...	Creating a Curve Stamp
User Stamp...	Creating User-Defined Features
Louver...	Creating a Louver

Insert -> Patterns



For...	See...
Rectangular Pattern	Creating Rectangular Patterns
Circular Pattern	Creating Circular Patterns
User-Defined Pattern	Creating User-Defined Patterns

Insert -> Corners

For...	See...



Corner...	Creating Corners
Chamfer...	Creating Chamfers

Insert -> Replication Tools



For...	See...
PowerCopy Creation	Creating PowerCopy Features
PowerCopy Save in Catalog	Saving PowerCopy Features

Up Next

NEW Sheet Metal Toolbar



 See Managing Default Parameters

 See Creating Walls from a Sketch

 See Creating Walls From An Edge

 See Extruding

 See Creating Rolled Walls



 See Creating Bends From a Line

 See Creating Swept Walls



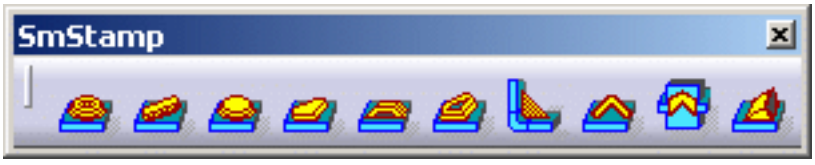
 See Unfolding the Part



 See Creating a Cutout

 see Creating a Hole

 See Stamping





 See Patterning

 See Creating a Local Corner Relief


See Creating Corners


See Creating Chamfers


See Mapping Elements




See Save As DXF

Back Up Next

Reference Elements Toolbar



 See Creating Points

 See Creating Lines

 See Creating Planes

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Specification Tree

Within the Sheet Metal Design workbench, you can generate a number of features that are identified in the specification tree by the following icons.

Further information on general symbols in the specification tree are available in Symbols Used in the Specification Tree.

 Sheet Metal Parameters

 Wall

 Wall On Edge

 Extrude

 Rolled Wall

 Bend

 Flange

 Hem

 Tear Drop

 User-defined Flange

 Cutout

 Hole

 Flanged Hole

 Bead

 Circular Stamp

 Surface Stamp

 Bridge

 Flanged Cutout

 Stiffening Rib

 Rectangular Pattern

 Circular Pattern

 User-Defined Pattern

 Corner Relief

 Corner

 Chamfer

 Mapping

 Point

 Line

 Plane

 Flat Bend

 Curve Stamp

 User-Defined Stamp

 Louver

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Customizing For NEW SheetMetal Design

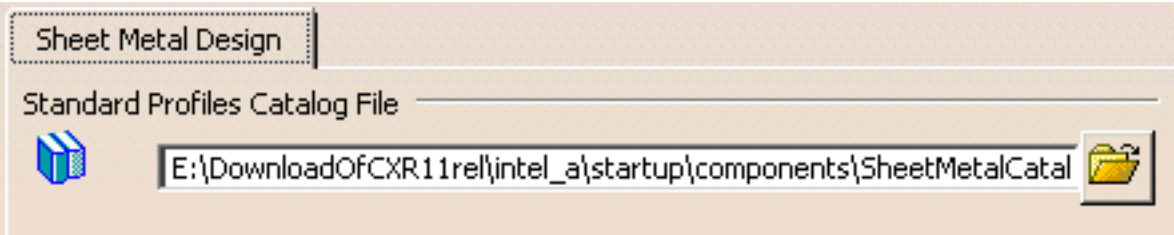
This section describes how to customize different settings specific to the NEW SheetMetal Design workbench. The settings described here deal with permanent setting customization.

Customizing General Settings
Customizing Standard Files

Customizing General Settings

This task explains how to customize the NEW Sheetmetal Design General Settings.

1. Select the **Tools -> Options** menu item.
   The Options dialog box appears.
2. Click **Mechanical Design** category then the **NEW Sheet Metal Design** subcategory.



The default path is displayed in the Standard Profiles Catalog Files. You can modify it by clicking the **Browse** icon .

If no catalog path has been defined prior to entering the Catalog Browser command, the default catalog is selected and is path automatically added to the Standard Profiles Catalog Files field.

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Customizing Standard Files

This section describes how to customize settings.
The task described here deals with permanent setting customizing.

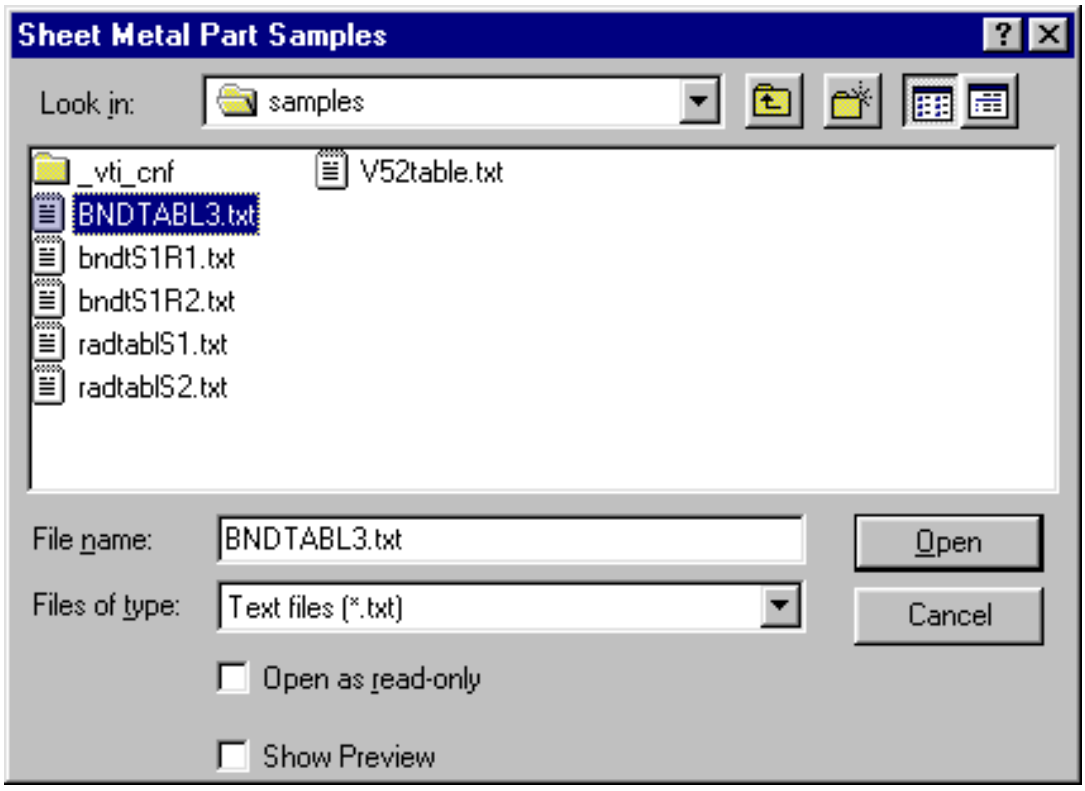
Using Sheet Metal Standards Files

This task explains how to access company standards files.

Open a new document.

1. Click the **Sheet Metal Parameters** icon .
   The Sheet Metal Parameters dialog box opens.

2. Select the **Sheet Standards Files...** button.
   The Sheet Metal Part Samples window is displayed.

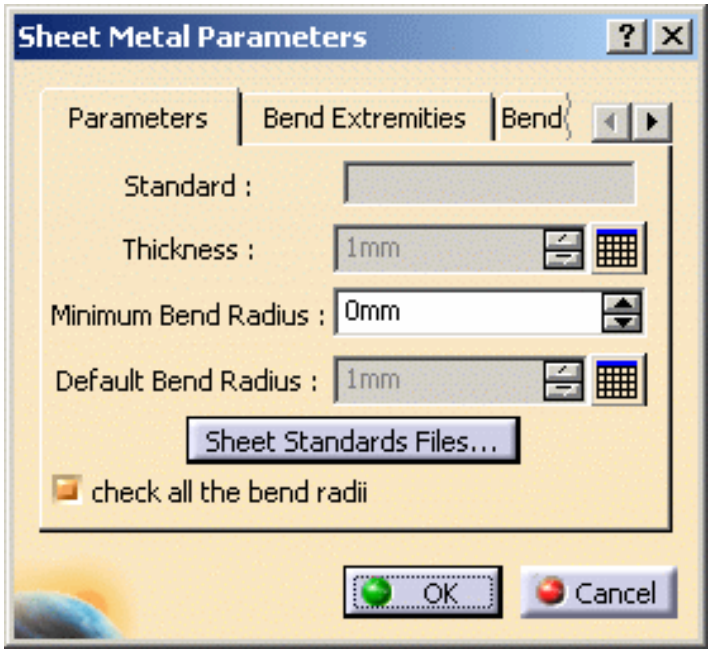


3. Indicate the path to the Sheet Metal table.

These files are available under .xls or .txt format.

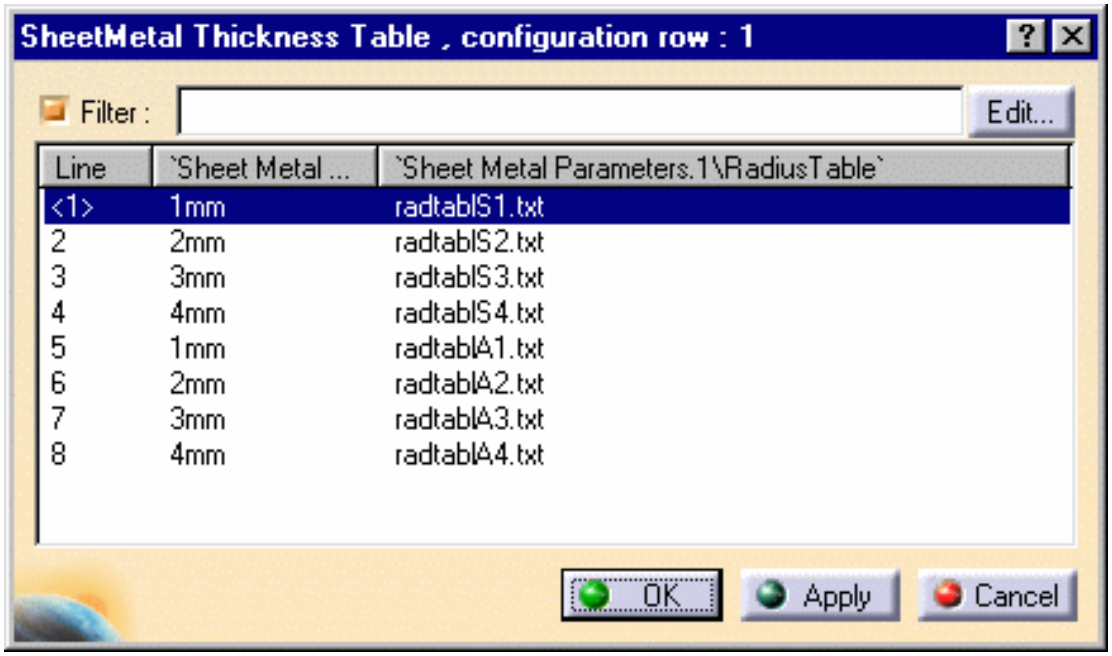
4. Click **Open**.

   In the Sheet Metal Parameters dialog box, the **Design Table** icon  appears opposite the Thickness and Bend radius fields.



The parameters are now in gray, indicating that you can no longer modify the values.

5. Click the Thickness Design Table icon  and select the line containing the appropriate parameters (for example Line 1).



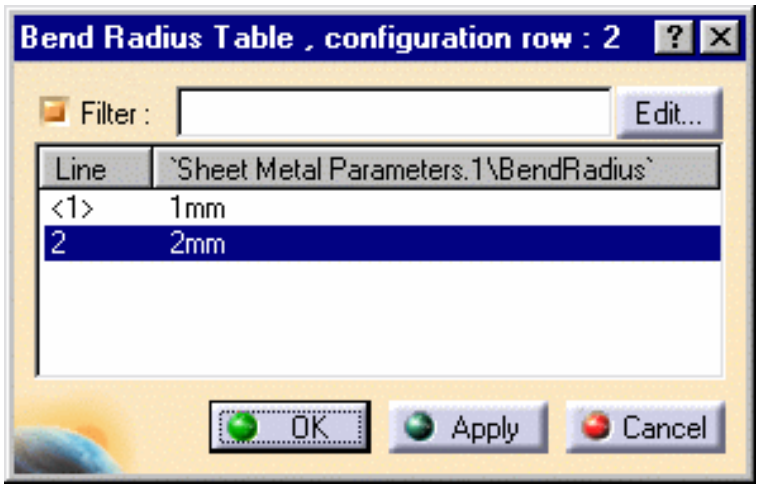
Using the **Tools -> Options -> General -> Document** tab, **Other Folders** option, you can specify where the files are located. Refer to Document.

This scenario can work when the .CATPart document and all reference table files (Design & Radius) are located in the same directory. This directory is the current one when the Design table is created, and also when the .CATPart is open.

However, generally speaking, you must reference the complete path indicating where the radius table files are to be found in the RadiusTable column. In this case, regardless of the current directory, the correct tables are located when re-opening the .CATPart document.

6. Click **OK**.
   The parameter values are updated in the Sheet Metal Parameters dialog box.

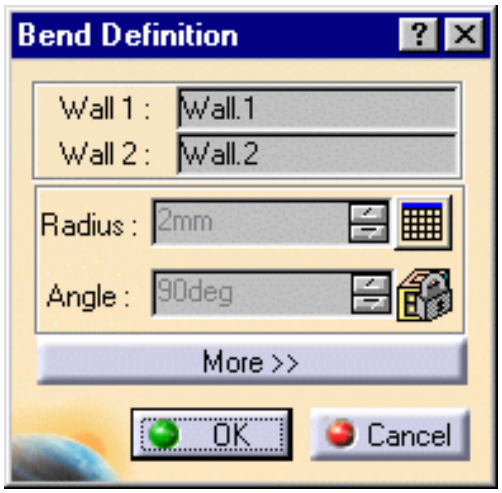
7. Click the Bend Radius Design Table icon .
8. Select line 2 and click OK. The parameter values are updated in the Sheet Metal Parameters dialog box.



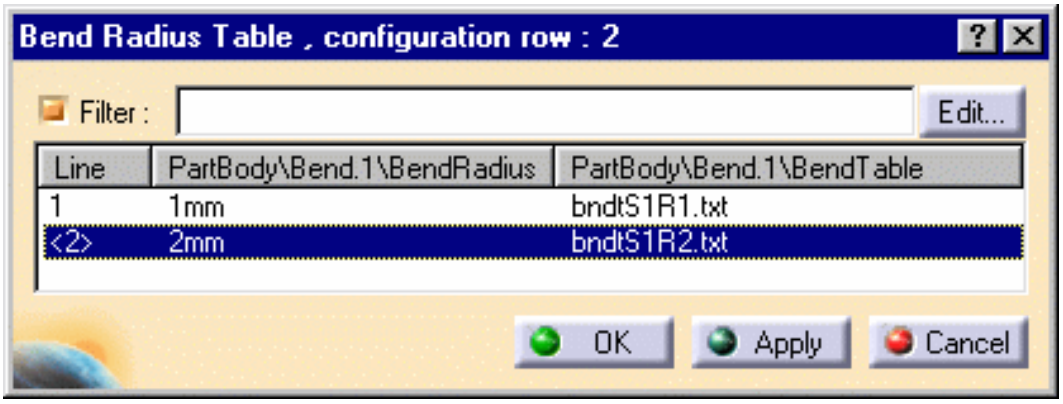
9. Create a bend.

   The Bend Definition dialog box displays a design table for the Bend Radius.

   The default mode, that is to say the formula: **Bend Radius = Part Radius** is deactivated.



Let's see the Bend Radius Table, using this icon .



It shows the Bend Radius and the corresponding Bend Table.

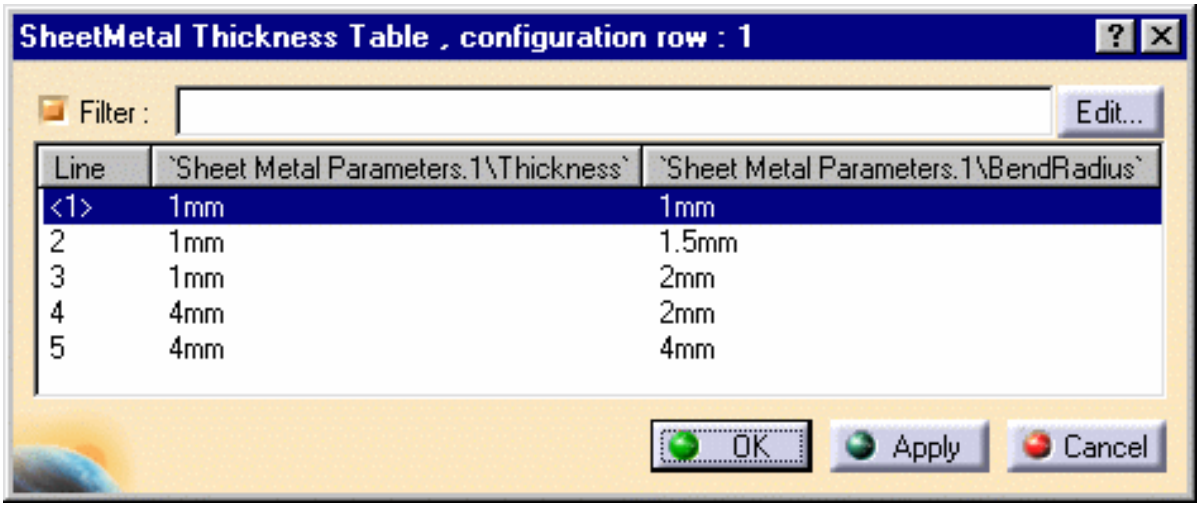
10. Click OK.

If the **Angle** value is contained in the **Bend Table**, the **Bend Allowance** uses the corresponding value.
If not, the **Bend Allowance** is computed according to the KFactor.

Using the Sheet Metal Design Tables:

Steps 1 to 4 are identical.

5. Click the Design Table icon and select a line.



6. Click **OK**.
The parameter values are updated in the Sheet Metal Parameters dialog box.

At that time, the parameters Thickness and Bend radius are driven by the design table. They are now in gray, indicating that you can no longer modify the values.
Note that if you create a bend, there is no design table: it's the formula which is used.

To disable the access to design tables:

- Select the **Tools -> Options -> Part -> Display** tab and check **Relations:**

  the **Design Table** icon  is displayed in the specification tree.

- Right-click this icon: the contextual menu appears.
- Select **SheetMetal Thickness Table object -> Deactivate**

The relation is no longer used but still exists.
It can be activated at any time.

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Glossary

B C E F P R S W

B

bend A feature joining two walls

bend extremity Axial relimitation for a straight bend

C

corner relief A feature created at the intersection of several bends or of a bend and a wall allowing to remove matter so that the part can fold/unfold without overlapping.

cutout A feature corresponding to an opening through a feature.
The shape of the opening corresponds to the extrusion of a profile.

E

extrusion A feature created by extruding a profile and adding thickness

F

flange A feature created by sweeping a profile along a spine.
The different flanges or swept walls available are: simple and swept flange, hem and tear drop.

P

pattern A set of similar features repeated in the same feature or part

profile An open or closed contour including arcs and lines created by the **Profile** command in the Sketcher workbench

R

reference wall The first created wall; when unfolding the part, it is the fixed wall.

S

stamping A feature created by embossing sheet metal.
The different stampings available are:
point, extruded hole, curve, surface, bridge, louver and stiffening rib.

W

wall A feature created by adding thickness to a profile

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automatic bends

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creating
workbench
Generative Drafting
Sheet Metal Design

Version 5 Release 12
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Aerospace Sheetmetal Design



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- Using this Guide
- Where to Find More Information

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Getting Started
- Entering the Aerospace SheetMetal Design Workbench
- Defining the Aerospace SheetMetal Parameters
- Creating a Web from a Sketch
- Creating a Surfacic Flange on a Web
- Creating a First Cutout
- Extracting Drawings from the Aerospace SheetMetal Design Part

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- Managing the Default Parameters
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  - Computing the Bend Allowance
  - Defining the Compensations
- Creating a Web
- Creating a Surfacic Flange
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- Unfolding
  - Folded/Unfolded View Access
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Glossary

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Preface

Aerospace Sheetmetal provides an associative feature-based modeling enabling to design sheetmetal parts in concurrent engineering between the unfolded or folded part representation.

Aerospace Sheetmetal allows you to define a part using predefined features. Both the folded geometry and the flattened geometry can be computed from the feature specifications.

Using this Guide
Where to Find More Information

Preface

Aerospace Sheetmetal provides an associative feature-based modeling enabling to design sheetmetal parts in concurrent engineering between the unfolded or folded part representation.

Aerospace Sheetmetal allows you to define a part using predefined features. Both the folded geometry and the flattened geometry can be computed from the feature specifications.

Using this Guide
Where to Find More Information

Using this Guide

This guide is intended for the user who needs to become quickly familiar with the CATIA Version 5 Aerospace Sheetmetal Design product. The user should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars.

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Where to Find More Information

Prior to reading this book, we recommend that you read the *Infrastructure User Guide.*

The *Part Design*, the *Assembly Design* and the *Generative Drafting*, the *Sketcher* and the *Generative Shape Design* documentations may prove useful.

Finally, you can read the *Sheet Metal Design (NEW)* documentations to find out more about that product and to fully use the interoperability between these products.

Conventions

Back Up

What's New?

This table identifies what new or improved capabilities have been documented in Version 5 Release 12 of the Aerospace SheetMetal Design workbench.

New Functionalities

Defining the Compensations

Customizing Standards Files To Define Methods for Compensations

Creating a Stiffening Rib

Mapping Elements

Creating Corners

Creating Chamfers

Patterning

- Creating Rectangular Patterns
- Creating Circular Patterns
- Creating User-Defined Patterns

Integration With Part Design

Enhanced Functionalities

Creating a Surfacic Flange

A new compensation tab enables you to define compensations for joggles or flange sides

Creating a Joggle

You can apply compensations on a single, double, or twin joggle

Defining the Compensations

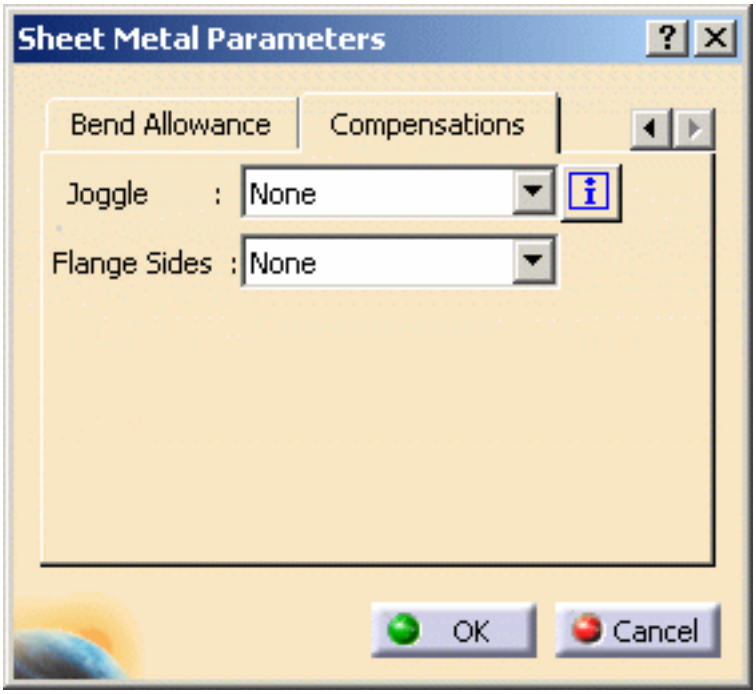
This section shows how to select the appropriate method to define compensations when on the joggle or the flange sides.
Compensation is a modification of the standard calculation of the unfolding process in order to represent at best the material behavior.

You first need to define which method to apply by customizing design tables.
To do so, please refer to the Customizing Standards Files To Define Methods for Compensations chapter.

1. Click the **SheetMetal Parameters** icon .

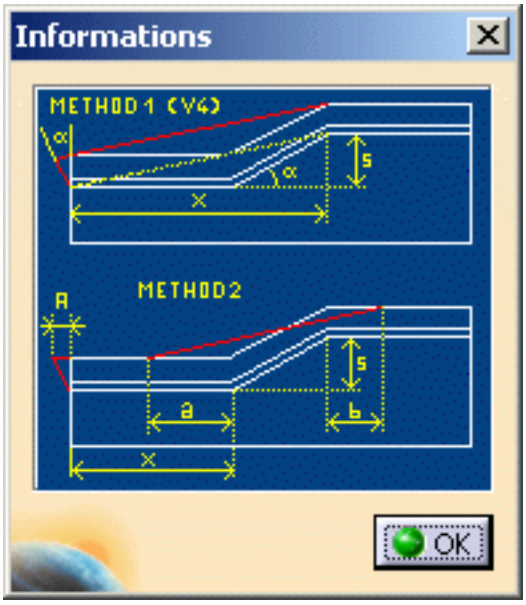
   The Sheet Metal Parameters dialog box is displayed.

   The fifth tab concerns the compensations.



2. In the **Joggle** combo list, select the method as defined in the SheetMetal Standard files (lien):

- **None**: no compensation is applied
- **Method 1** (= Method V4)
- **Method 2**



If the method you chose is not the one defined in the SheetMetal Standard Files, a warning message is issued prompting you to select another file.

You can click the information icon to display a schema explaining both methods.

3. In the **Flange Sides** combo list, select how the sides will be computed:

- **None**: no compensation is applied
- **Manual: Angle**: the deformation is computed according to an angle
- **Manual: Length**: the deformation is computed according to a length

4. Click OK in the dialog box to validate the compensations parameters.



Back Up

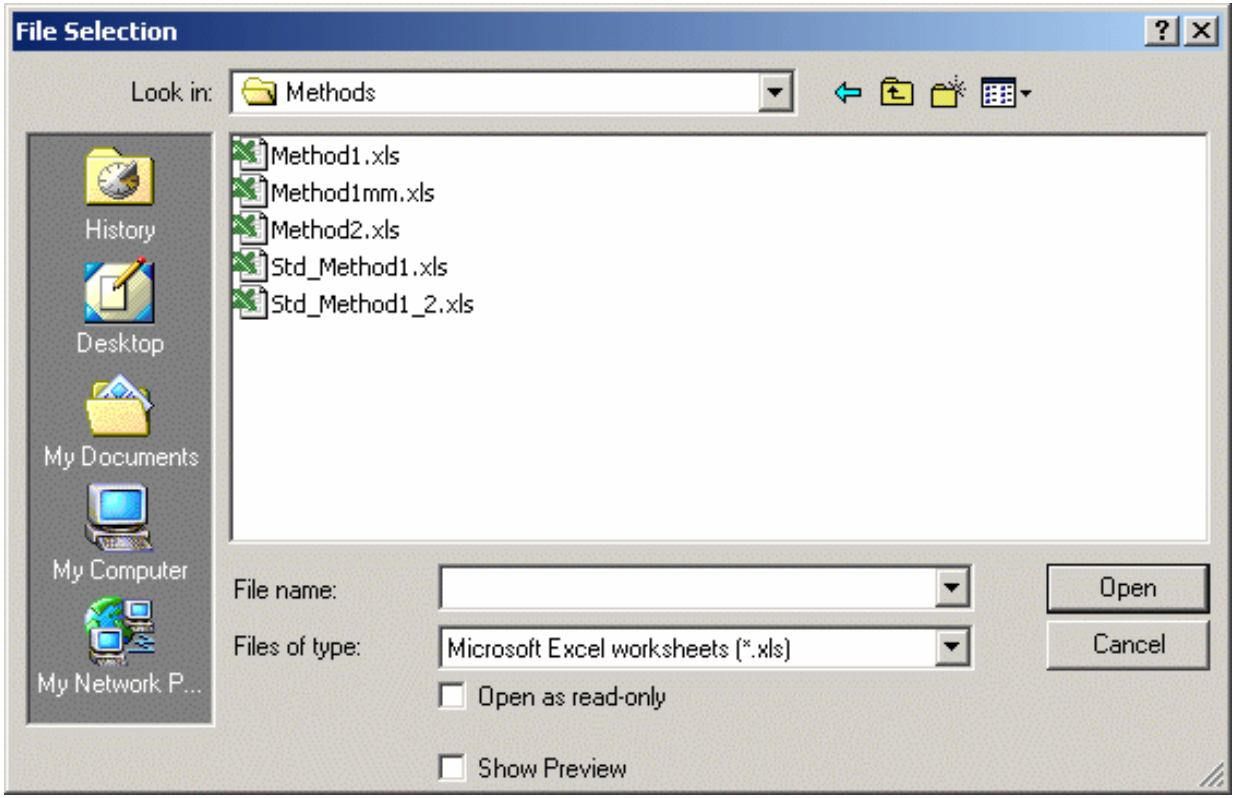
Customizing Standards Files To Define Methods for Compensations

This task explains how to access company standards files in order to define methods for joggle and sides compensations.

Open a new document.

1. Click the **Sheet Metal Parameters** icon .
   The Sheet Metal Parameters dialog box opens.

2. Select the **Sheet Standards Files...** button. The File Selection window is displayed.



3. Indicate the path to the Sheet Metal methods (Std_Method1.xls or Std_Method1_2.xls)

These files are available under .xls format.

Two methods are available to enable the joggle compensations.
Both use Design Tables.

Method 1 (=Method V4)

JoggleCompensation_C1 (in)	JoggleCompensation_C2 (in)	JoggleCompensation_C3 (in)	JoggleCompensation_C4 (in)	JoggleCompensation_C5 (in)	JoggleCompensation_Ra (in)
3	0.2	6	0.4	3	0.2

The path to this file can be found in the Std_Method1.xls associate file.

Method 2

Several files are needed.

JoggleCompensation_SMax (mm)	JoggleCompensation_a (mm)	JoggleCompensation_b (mm)	JoggleCompensation_Ra (mm)	JoggleCompensation_Rb (mm)
1	10	9	2	1.5
3	15	10	2.5	2
6	20	18	3	2.5
10	30	26	3.5	3
15	40	35	4	4
20	60	50	5	4.5
100000	70	55	6	5.5

JoggleCompensation_SMax

- If (S > 0 and S <= 1), then a = 5, b = 4, Ra = 1, Rb = 1.5
- If (S > 6 and S <= 10), then a = 30, b = 26, Ra = 3.5, Rb = 4
- If (S > 20), then a = 70, b = 55, Ra = 6, Rb = 7

The path to this file can be found in the Std_Method1_2.xls associate file.
Std_Method1_2.xls also contains paths for each defined thickness.

Thickness (mm)	JoggleCompensationMethod2	JoggleCompensationOnFlange
1	E:\www\spuCXR12\AslEnglish\aslug.doc\src\samples\	Flg\Method2Th1.xls
1.5		Flg\Method2Th15.xls
2		Flg\Method2Th2.xls

Here is an example of the file corresponding to a thickness of 1mm:



S corresponds to the depth of the joggle to compensate and X corresponds to the distance of the joggle from the flange side.

JoggleCompensation_SfMax

- If (S > 0 and S <= 1.5) **and** (X > 0 and X <= 4.5), then A = 0.55
- If (S > 0 and S <= 1.5) **and** (X > 24.5), then A = 0.95
- If (S > 1.5 and S <= 10) **and** (X > 4.5 et X <= 18.5), then A = 0.85
- If (S > 10) **and** (X > 24.5), then A = 1.5

4. Choose the appropriate file for the desired method.

5. Click **Open**.

In the Sheet Metal Parameters dialog box, if you chose Method 2, the **Design Table** icon  appears opposite the Thickness field.



Back Up

Patterning

This section explains and illustrates how to create various kinds of patterns on Aerospace Sheet Metal parts.


Create rectangular patterns: select the element to be duplicated, set the patterning type, and its parameters, and the reference direction


Create circular patterns: select the element to be duplicated, set the axial reference parameters, the reference direction, and possibly the crown definition


Create user-defined patterns: select the element to be duplicated, and the positioning sketch and anchor point


To have further information about patterns, refer to *Part Design User's Guide*.

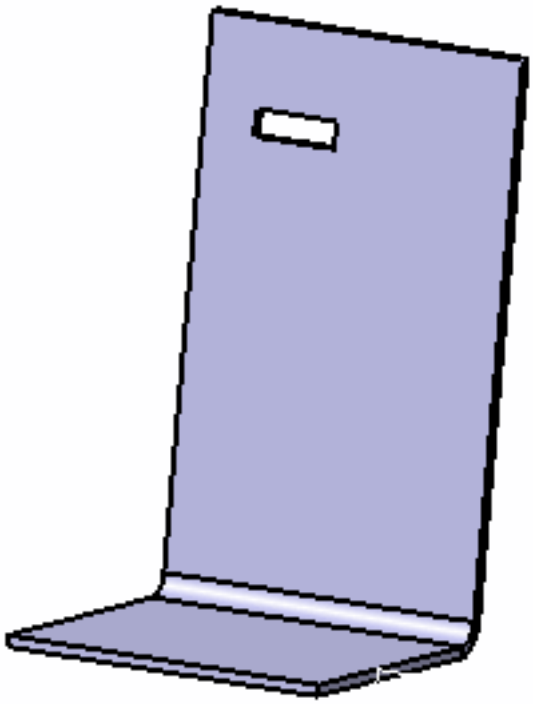
Back Up Next

Creating Rectangular Patterns

In this task, you are going to create rectangular cutouts according to a pattern. These features make the creation process easier.

You can only duplicate cutouts, holes, and stamps (except stiffening rib). These features must lie on a unique and planar surface.

Open the RectangularPattern1.CATPart document.
The part looks like this:



1. Select the rectangular cutout you want to duplicate.

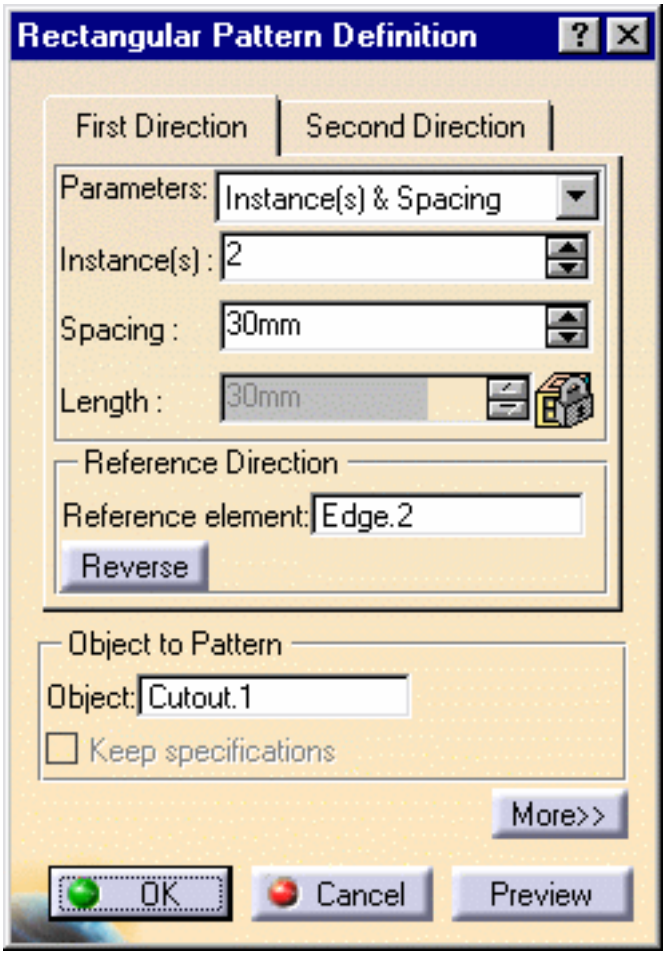
2. Click the **Rectangular Pattern** icon .

The **Rectangular Pattern Definition** dialog box is displayed. Each tab is dedicated to a direction to define the location of the duplicated feature.

3. Set the specification for the First Direction by selecting the first edge (**Edge.2**) as shown, to specify the first direction of creation.
An arrow is displayed on the flange.

The **Reverse** button enables to modify the direction.
You can also click the arrow in the 3D geometry.

4. Keep the **Instances & Spacing** options to define the parameters.
   Choosing these parameters types dims the **Length** field because the application no longer needs this specification to space the instances.



You can set the duplication parameters by choosing the number of instances, the spacing between instances, or the total length of the zone filled with instances. Three options are available:

1. **Instances & Length**: the spacing between instances is automatically computed based on the number of instances and the specified total length
2. **Instances & Spacing**: the total length is automatically computed based on the number of instances and the specified spacing value
3. **Spacing & Length**: the number of instances is automatically computed to fit the other two parameters.

For each of these cases only two fields are active, allowing you to define the correct value.

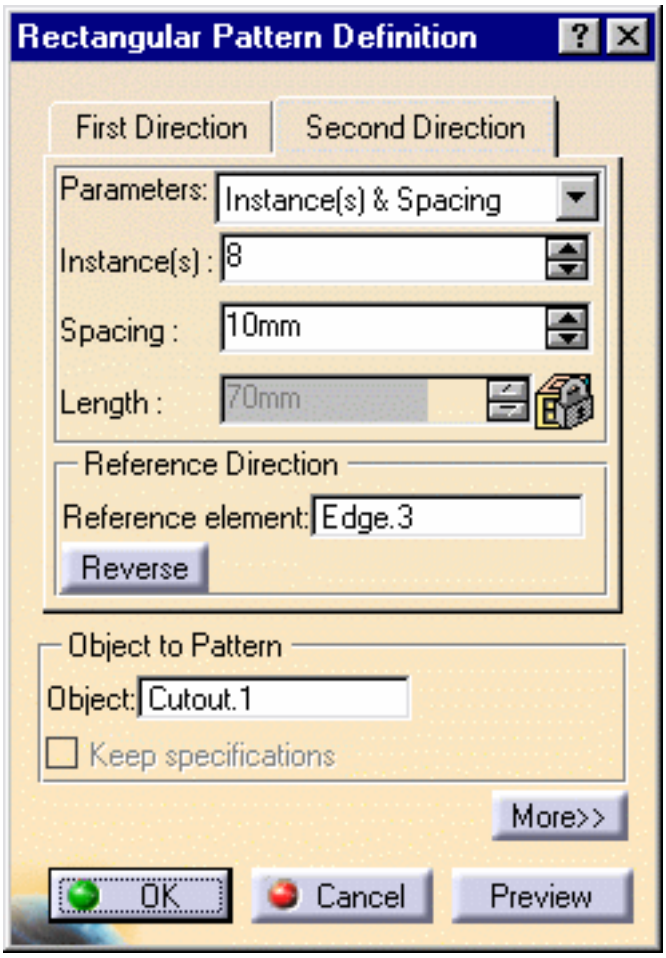
If you set **Instances & Length** or **Spacing & Length** parameters, note that you cannot define the length by using formulas.

Patterns should not go beyond the surface (this can be checked using the preview).

5. Enter 2 as the number of instances you wish to obtain in the first direction.
6. Define the spacing along the grid: enter 30mm.

Defining the spacing along the grid and the length of your choice, would make the application compute the number of possible instances and space them at equal distances.

7. Now, click the **Second Direction** tab to define the other parameters.



Note that defining a second direction is not compulsory. Creating a rectangular pattern defining only one direction is possible.

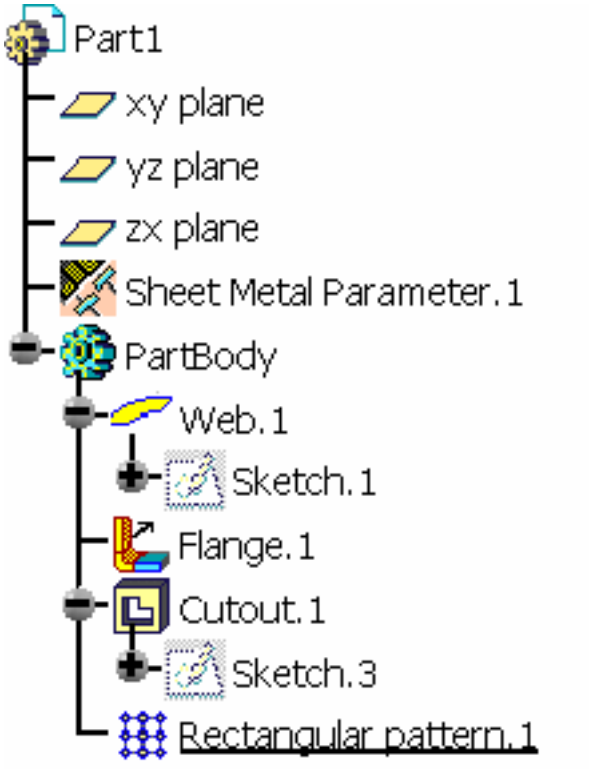
8. Select the second edge (**Edge.3**), as shown, to define the second direction.

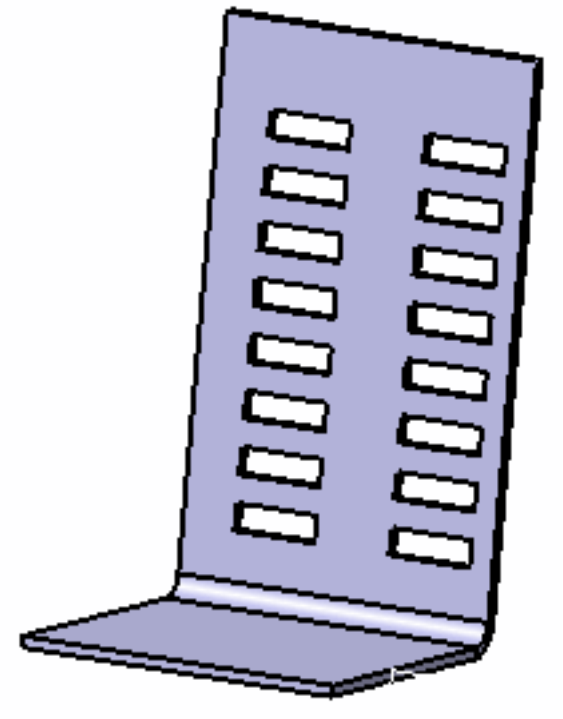
9. Keep the **Instances & Spacing** option: enter 8 and 10 mm in the appropriate fields.

   Additional cutouts have been aligned along this second direction.

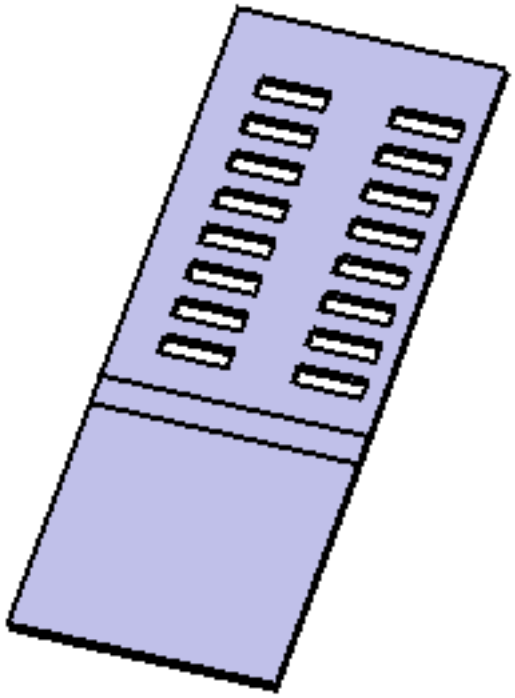
10. Click **OK** to repeat the cutouts.

After the update, the part looks like this:





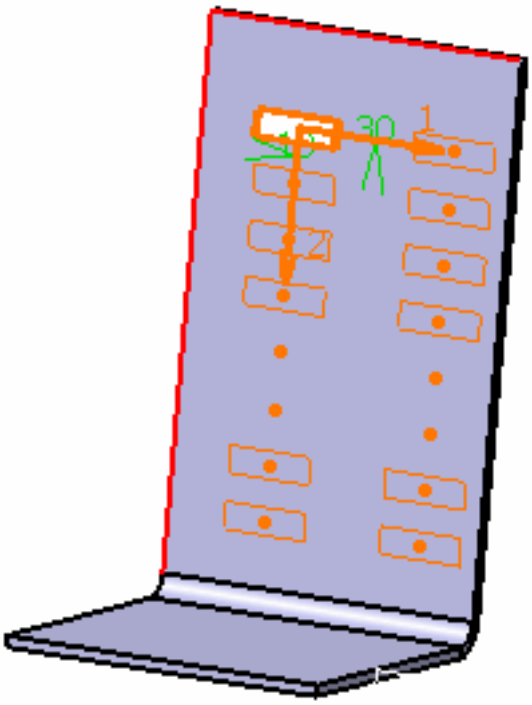
11. Select this icon  to unfold the part:
The pattern is updated on the unfolded view.



12. Click the More>> button to display further options.

The **Simplified representation** option lets you lighten the pattern geometry, when more than 10 instances are generated in one direction. What you need to do is just check the option, and click Preview. The system automatically simplifies the geometry:

You can also specify the instances you do not want to see by double-clicking the dots. These instances are then represented in dashed lines during the pattern definition and then are no longer visible after validating the pattern creation. The specifications remain unchanged, whatever the number of instances you view. This option is particularly useful for patterns including a large number of instances.



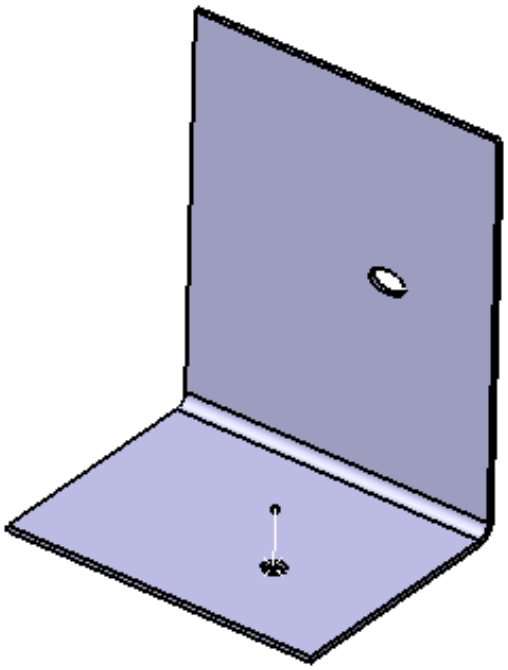
Up Next

Creating Circular Patterns

In this task, you are going to create circular cutouts according to a pattern.
These features make the creation process easier.

You can only duplicate cutouts, holes, and stamps (except stiffening rib).
These features must lie on a unique and planar surface.

Open the CircularPatterns1.CATPart document.
The part looks like this:

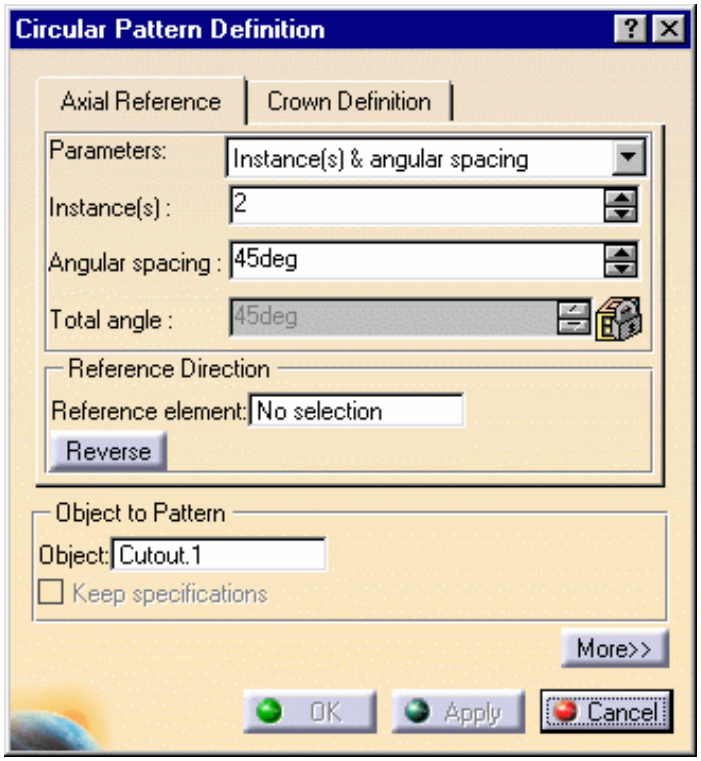


1. Select the circular cutout you want to duplicate.

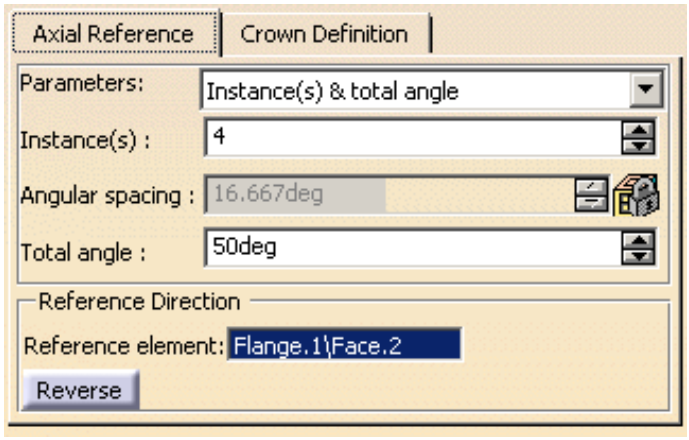
2. Click the **Circular Pattern** icon .

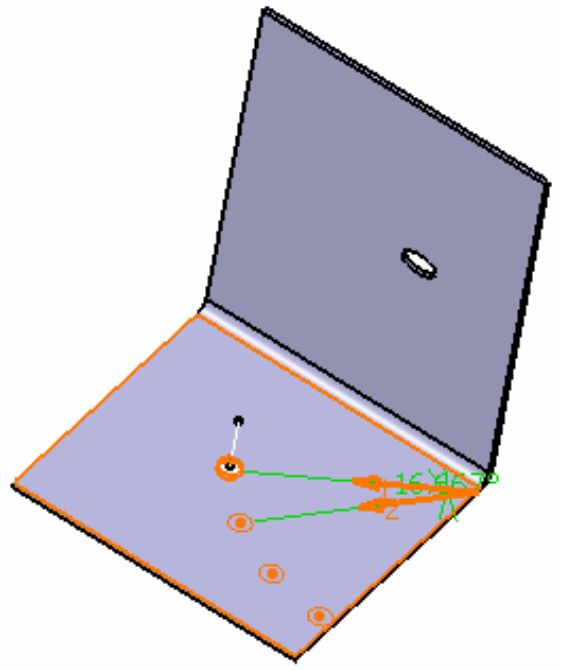
   The **Circular Pattern Definition** dialog box is displayed.

3. Define the **Axial Reference** by choosing the **Parameters** type, and reference direction.

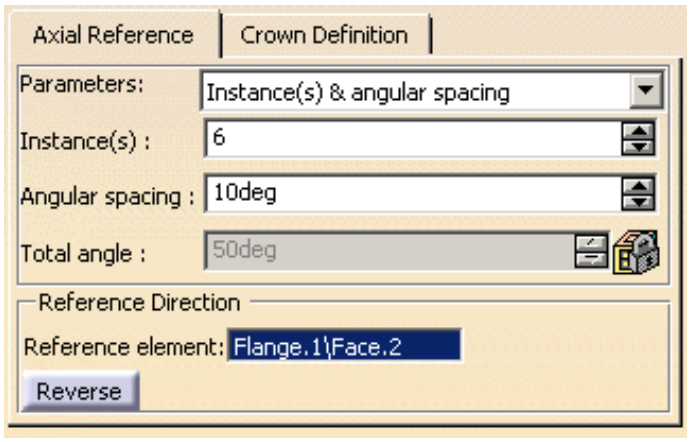


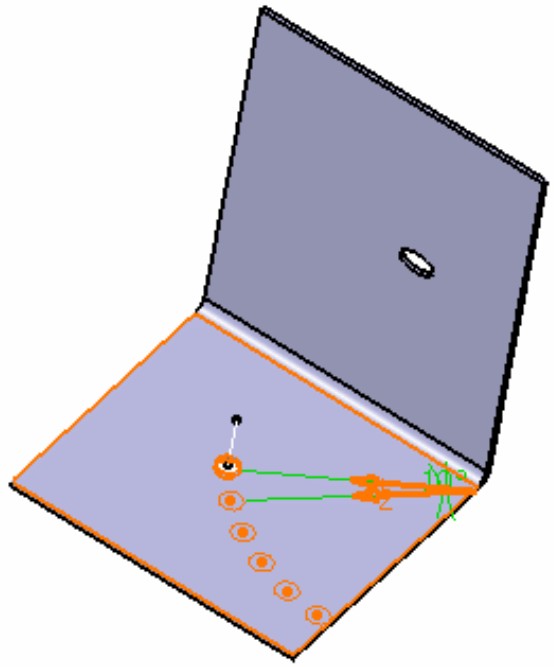
- **Instance(s) & total angle**: the number of patterns as specified in the instances field are created, in the specified direction, and evenly spread out over the total angle.



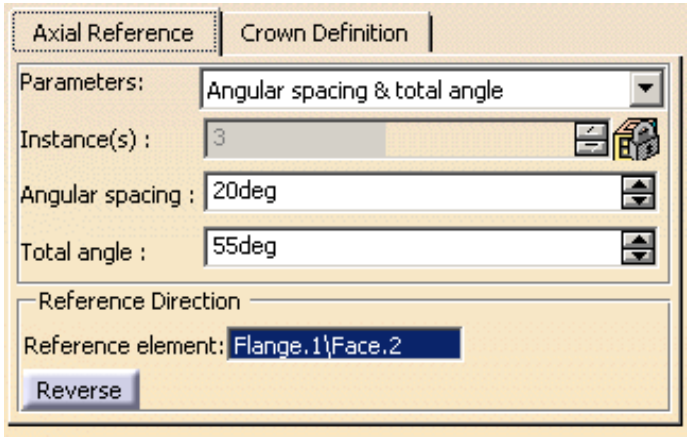


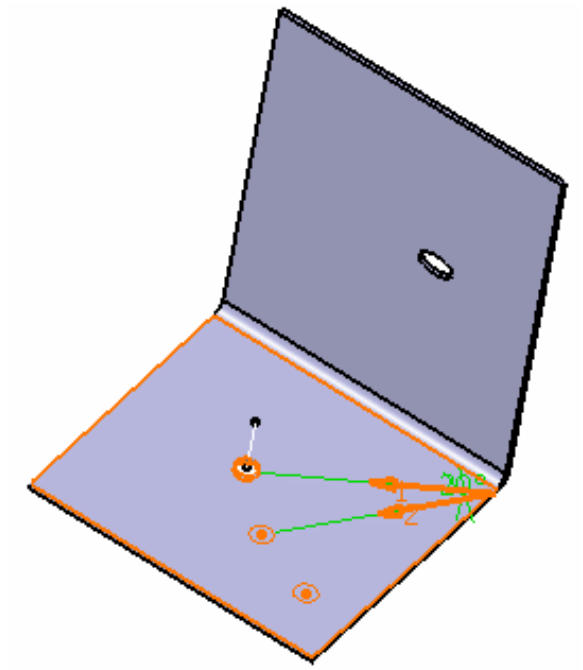
- **Instance(s) & angular spacing**: the number of patterns as specified in the instances field are created in the specified direction, each separated from the previous/next one of the angular angle value.



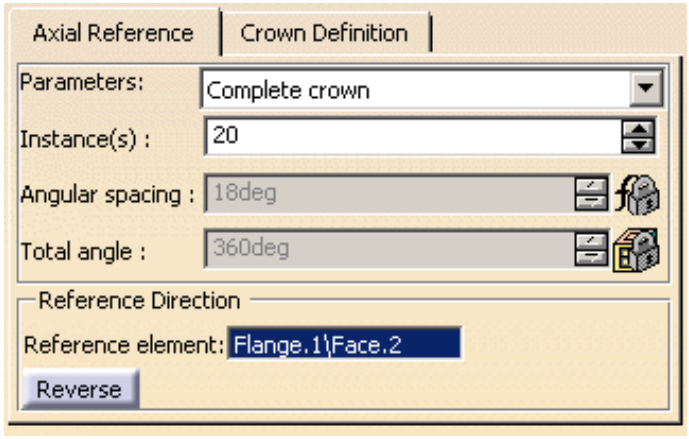


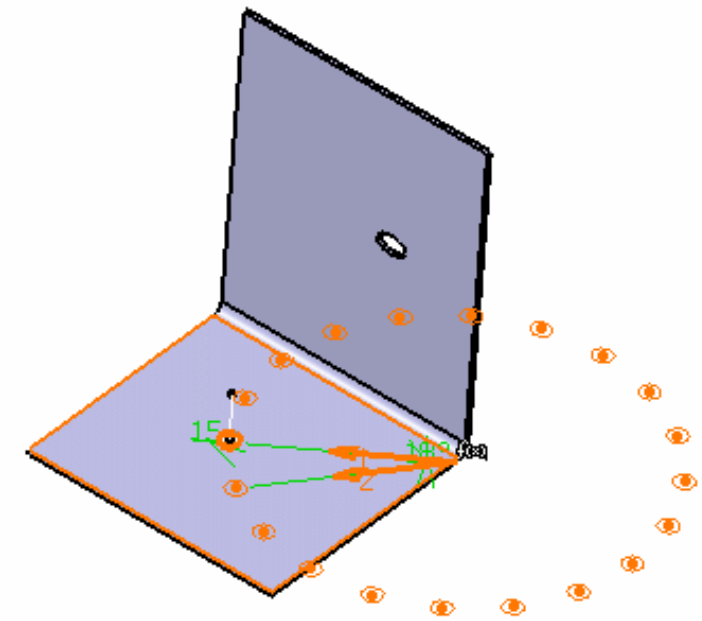
- **Angular spacing & total angle**: as many patterns as possible are created over the total angle, each separated from the previous/next one of the angular angle value.





- **Complete crown**: the number of patterns as specified in the instances field are created over the complete circle (360°).





If you set **Instance(s) & total angle** or **Angular spacing & total angle** parameters, note that you cannot define the length by using formulas.

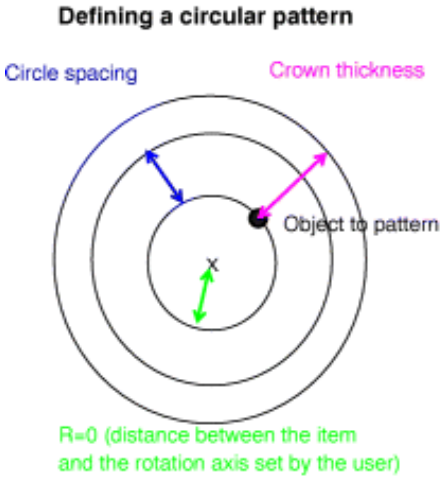
4. Click the Reference element and select the element defining the rotation axis.
   Here select the face on which lies the circular cutout.

- To define a direction, you can select an edge or a planar face.
  Should you select the face of a web, the rotation axis would be normal to that face.
- Click the **Reverse** button to inverse the rotation direction.

Now you are going to add a crown to this pattern.

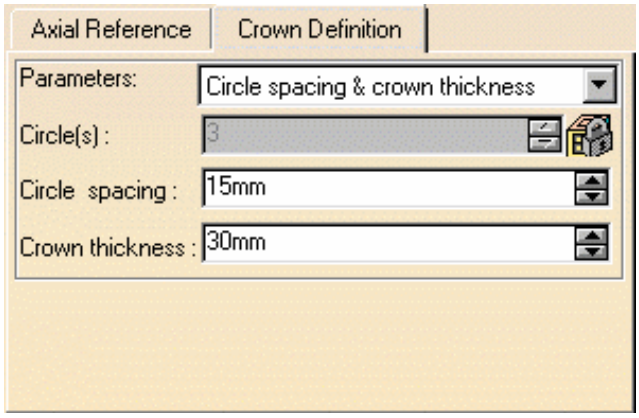
5. Click the **Crown Definition** tab, and choose which parameters you wish to define the crown.

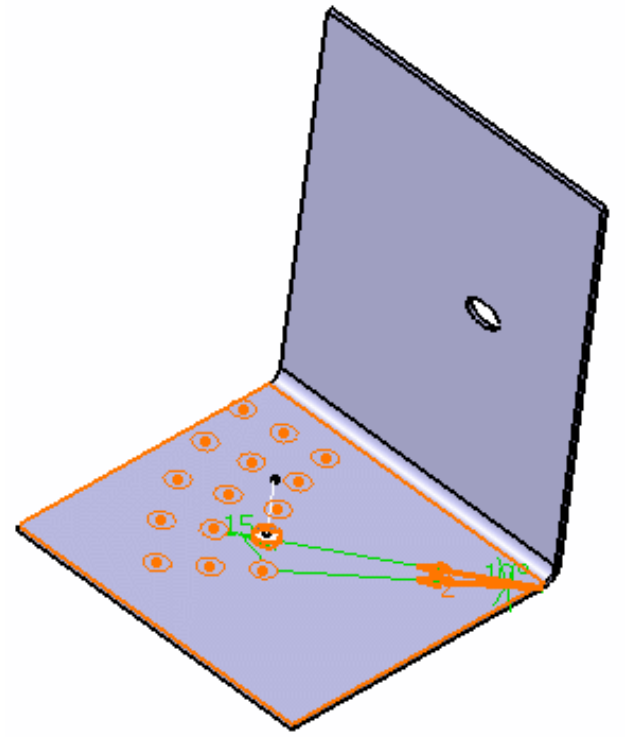
   This figure may help you define these parameters:



- Circle(s) and crown thickness: you define the number of circles and they are spaced out evenly over the specified crown thickness
- Circle(s) and circle spacing: you define the number of circles and the distance between each circle, the crown thickness being computed automatically
- Circle(s) spacing and crown thickness: you define the distance between each circle and the crown thickness, and the number of circles is automatically computed.

For example, using the values described above for the **Angular spacing & total angle** option, you could define the crown as:

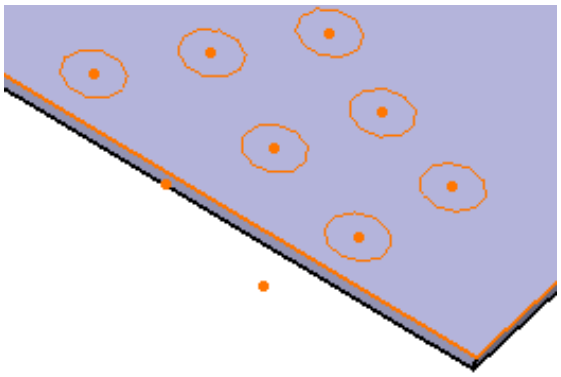




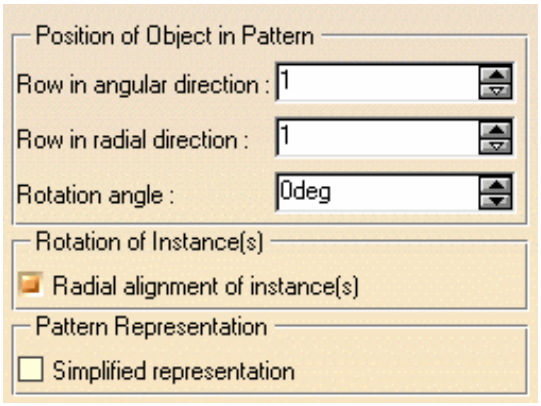
Note that one of the pattern goes beyond the flange (this can be checked using the preview).

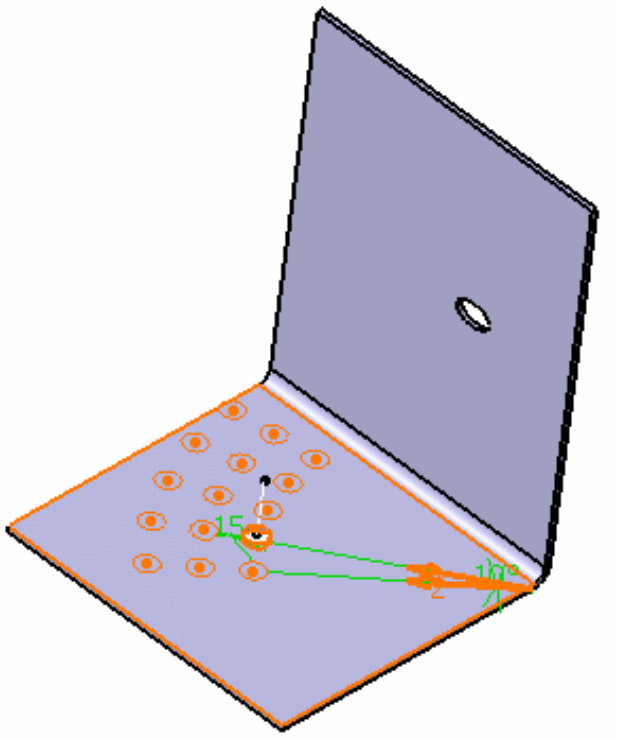
You can delete instances of your choice when creating or editing a pattern. To do so, just select the points materializing instances in the pattern preview.

The instance is deleted, but the point remains, as you may wish to click it again to add the instance to the pattern definition again.



6. Click the **More>>** button to display further options:





Using these options, you can change the position of the selected cutout within the crown. For example, if you set the **Row in angular direction** parameter to 4, this is what you obtain: the initially selected cutout is the fourth instance, based on the rotation direction, of the pattern.

Typically, in this case, you might want to edit the pattern and click again the instance that you removed above, to get a full pattern.

- The **Simplified representation** option lets you lighten the pattern geometry, when more than 10 instances are generated in one direction. What you need to do is just check the option, and click Preview. The system automatically simplifies the geometry:
  You can also specify the instances you do not want to see by double-clicking them . These instances are then represented in dashed lines during the pattern definition and then are no longer visible after validating the pattern creation. The specifications remain unchanged, whatever the number of instances you view. This option is particularly useful for patterns including a large number of instances.
- When checking the **Radial alignment of instances**, all instances have the same orientation as the original feature. When unchecked, all instances are normal to the lines tangent to the circle.

7. Click OK to create the pattern.

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Creating User-Defined Patterns

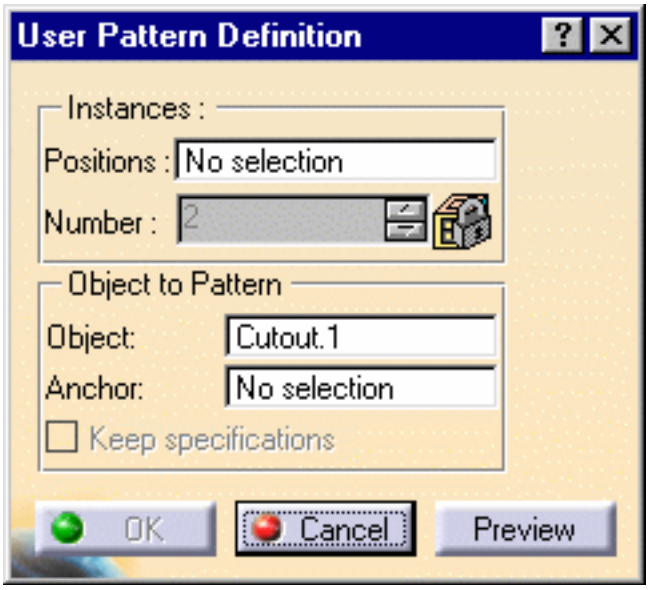
The User Pattern command lets you duplicate a feature, such as a sketch, a cutout, a stamp, or any other feature as many times as you wish at the locations of your choice. Locating instances consists in specifying anchor points. These points are sketches.
You can only duplicate cutouts, holes, and stamps (except stiffening rib).
These features must lie on a unique and planar surface.

Open the UserPatterns1.CATPart document.

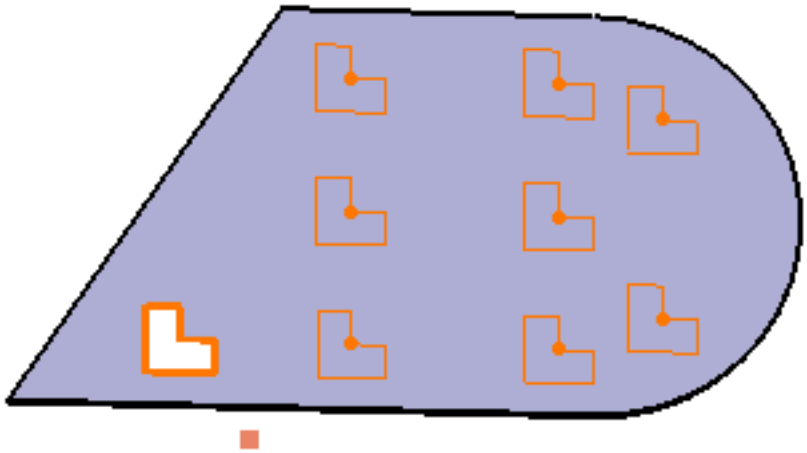
1. Select the feature to be duplicated.
   Here we selected the cutout.

2. Click the **User Pattern** icon .

   The User Pattern Definition dialog box is displayed.

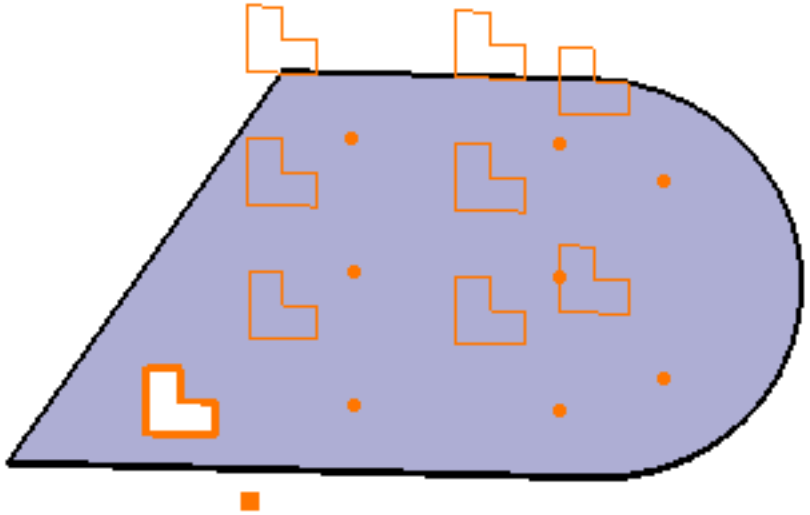


3. Select '**Sketch 3**' in the specification tree and click **Preview**.
   The sketch contains the points you need to locate the duplicated cutouts.

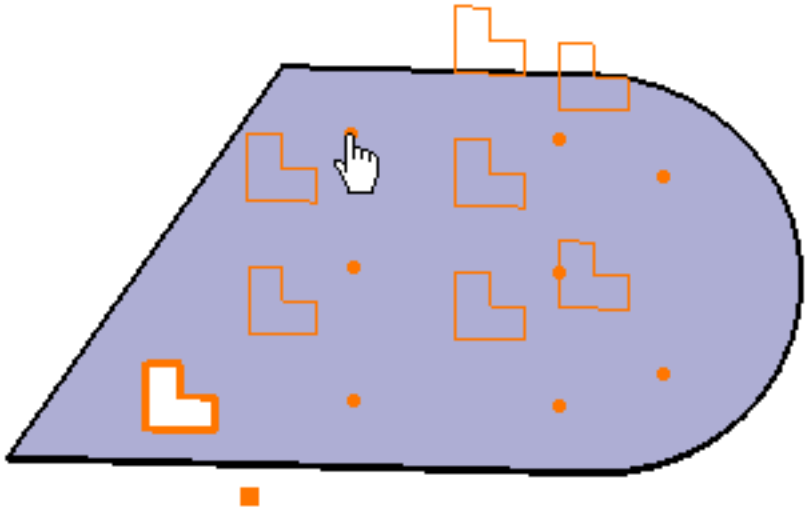


By default, the application positions each instance with respect to the center of gravity of the element to be duplicated. To change this position, use the anchor field: click the anchor field and select a vertex or a point.

4. Click inside the Anchor field and select the point (Point.1) to indicate a new reference location.



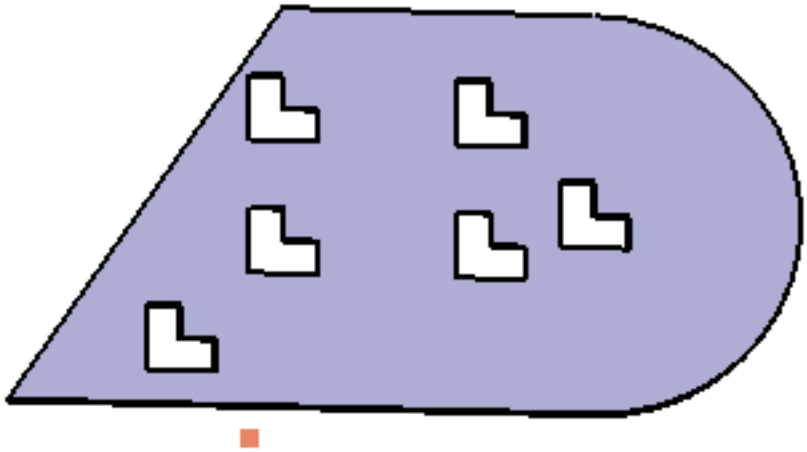
5. You can then click the points corresponding to the pattern instances to be removed.



Patterns should not go beyond the surface (this can be checked using the preview).

6. Click OK in the User Pattern Definition dialog box.

   Cutouts are created at the points of the sketch.



Would you need to unfold the part using the  icon, you would notice that the pattern is updated.

Please be careful concerning the content of the sketch selected to locate the instances of the User Pattern. The sketch should only include the points locating the instances of the selected reference feature. Therefore it is useful to create a point as "Construction Element" corresponding to the reference feature (in case of constraints for example), so that the instances are different from this feature.



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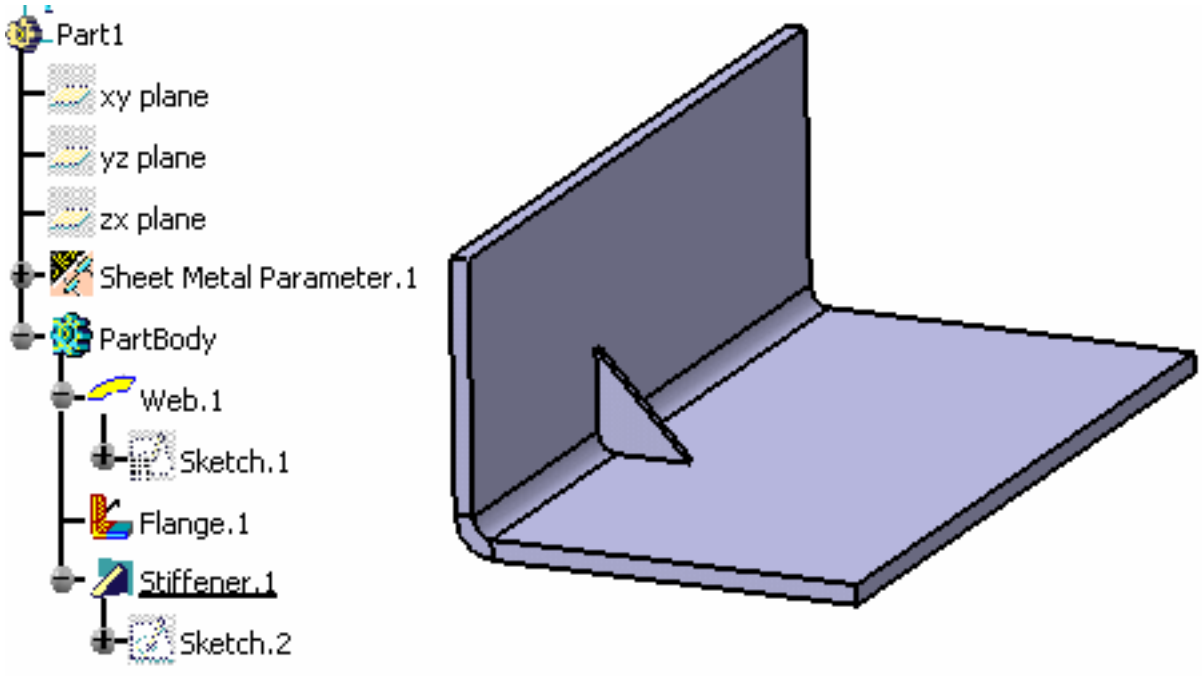
Integration With Part Design

You can open the Integration1.CATPart document.

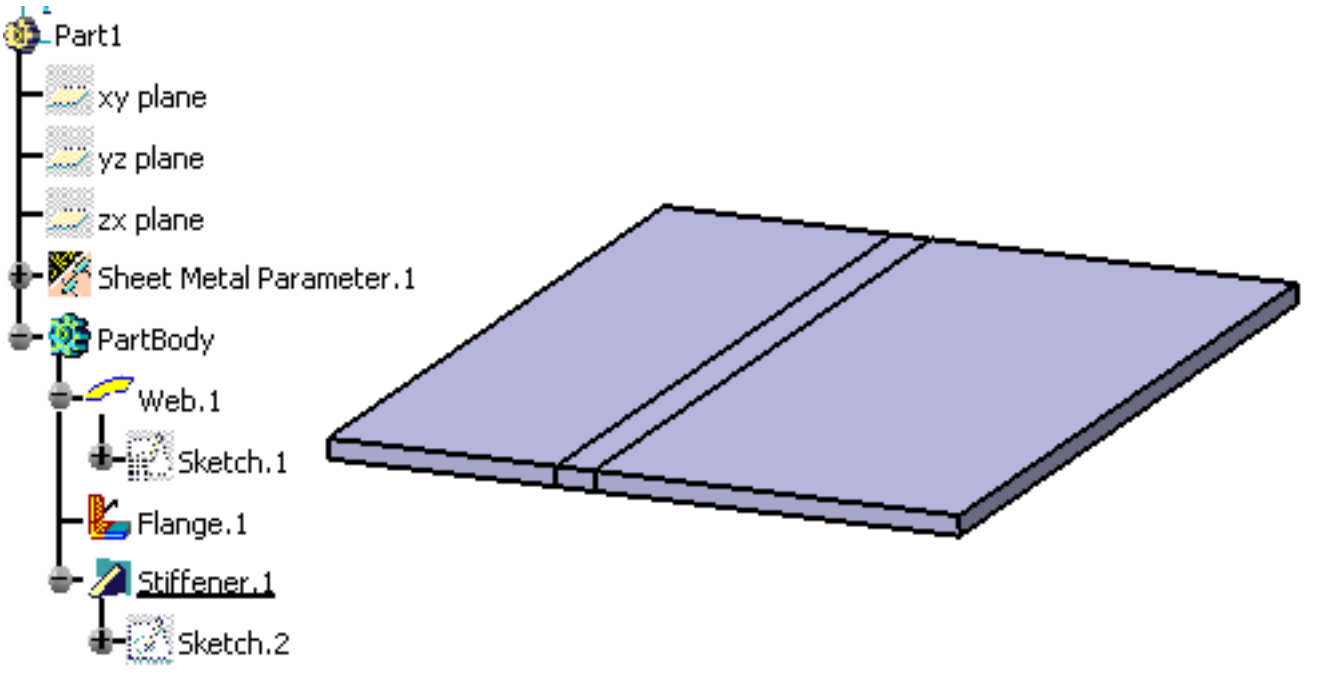
In a CATPart document, you may have Part Design features and Aerospace Sheet Metal features according to the following rules:

- Part Design features can be created before Aerospace Sheet Metal features.
- a Part Design feature can also be created after Aerospace Sheet Metal features as long as the part is in folded view.
- in the unfolded view, the Part Design feature will not be displayed.
- it is no longer possible to create Aerospace Sheet Metal features after this last Part Design feature in folded view.

1. Create two walls with an Automatic or Cylindrical Bend.
2. Switch to Part Design workbench.
3. Launch the Sketcher and draw an oblique line in the yz plane.
4. Click the icon  to create a **Stiffener**.

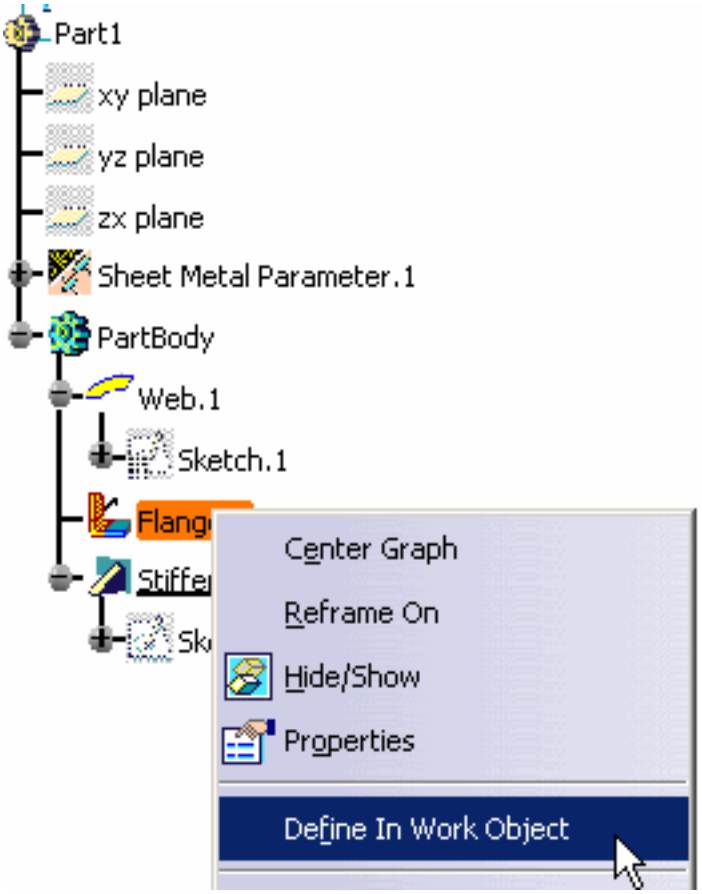


5. Switch to the Aerospace Sheet Metal workbench.
6. Click the **Unfold** icon .



The stiffener is not displayed on the unfolded view.

To add a new Aerospace Sheet Metal feature, select the Flange for example and right-click the **Define In Work Object** item.



The new Aerospace Sheet Metal feature will be added after the Flange but before the Stiffener.

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Creating a Surfacic Flange

This section explains how to create a surfacic flange on a web, or an existing surfacic flange (in that case, their fillets must not intersect).

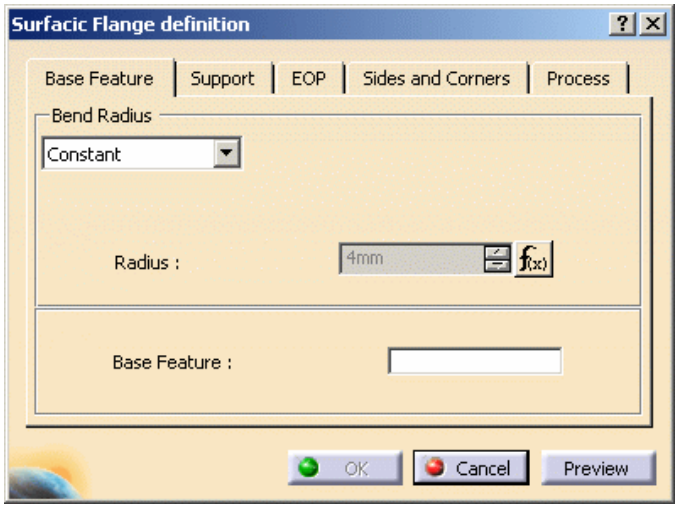
Open the Flange1.CATPart document.
Create a web as shown in the previous task.

Here are the different elements taken into account when creating a surfacic flange:

- Base Feature
- Support
- EOP
- Sides and Corners
- Process
- Compensations

1. Click the **Surfacic Flange** icon .

   The Surfacic Flange definition dialog box is displayed.



Base Feature

In the Base Feature tab, the Bend Radius is of **Constant** type. It is set to the default bend radius of the part.

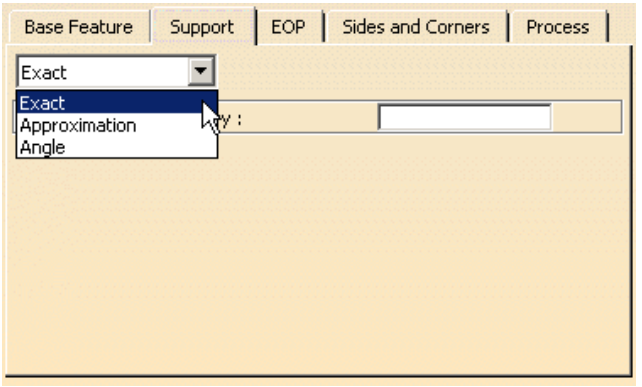
2. You can modify the fillet Radius value by changing the driving equation: click the icon.
   The Formula Editor dialog box opens, you can modify the dictionary and the parameters.
   Or you may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.

3. Choose the web as the Base Feature.

   Once you chose the base feature, the Support tab automatically displays.

Support

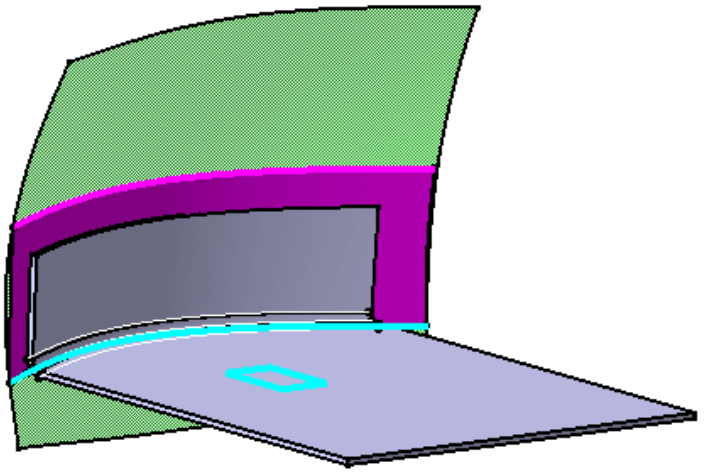
4. In the Support tab, choose the surfacic flange's geometrical support. It can either be a surface, a plane or a curve.

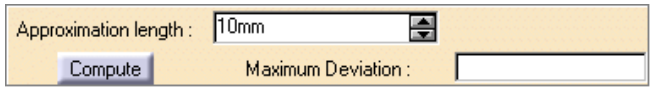


Make sure the support is big enough to be able to later define an EOP with a length from OML. The OML is a curve created by intersecting the flange support and a plane perpendicular to the web and normal to the OML.

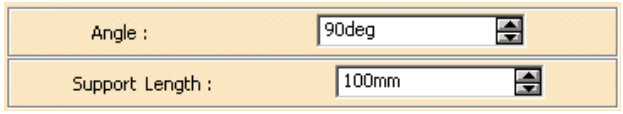
- Exact: the selected support is to be used for the creation of the surfacic flange.
- Approximation: the support surface is approximated using a ruled surface. This ruled surface is defined from two curves:
  - the OML (in light blue), computed at the intersection between the support surface and the web plane.
  - a curve parallel to the OML (in pink), computed at a distance equal to the approximation length

  This mode enables you to compute the maximum deviation between the support geometry and the approximated surface.

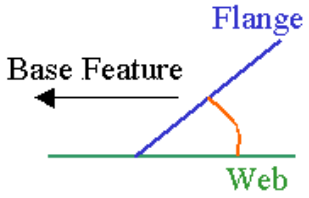




- Angle: the support of the surfacic flange can also be defined by a line, a curve, an angle or the edge of a base feature. The angle is constant and you can change its value using the spinners.



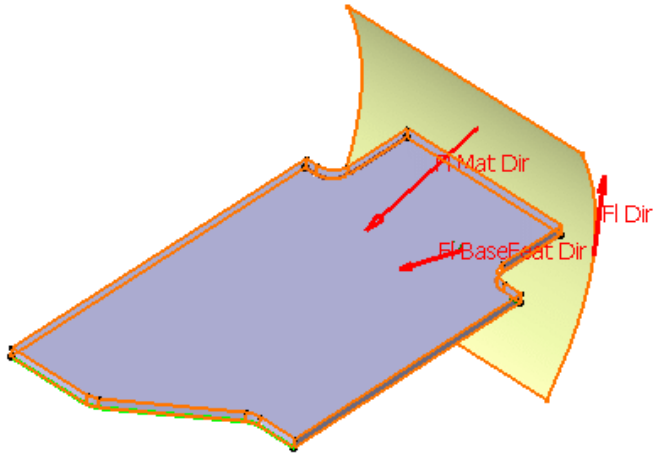
You can modify the Support Length generated by the curve and the angle. By default, the length is set to ten times the EOP (Edge of Part) length. If the default EOP length is higher than 100mm, you need to modify the surface length.



The red angle is the angle taken into account when creating the surfacic flange.

5. Define the vectors' directions.

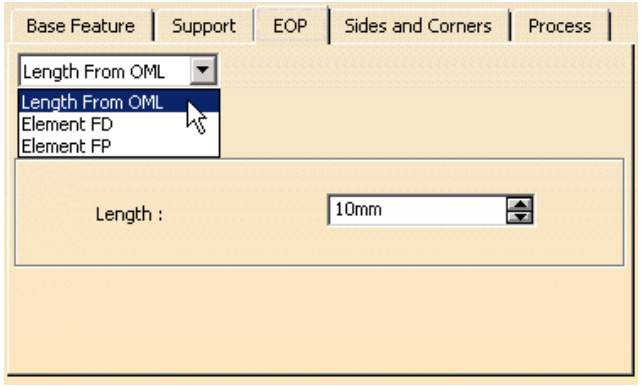
   The vectors show the Base Feature Direction, the Direction and the Material Direction according to the direction of the geometrical support of the surfacic flange.
   You can modify the directions by clicking the arrows.



The surfaces (or curves) used to define the support surface must be continuous in point and tangency.

EOP

6. In the EOP (Edge Of Part) tab, you can define either:

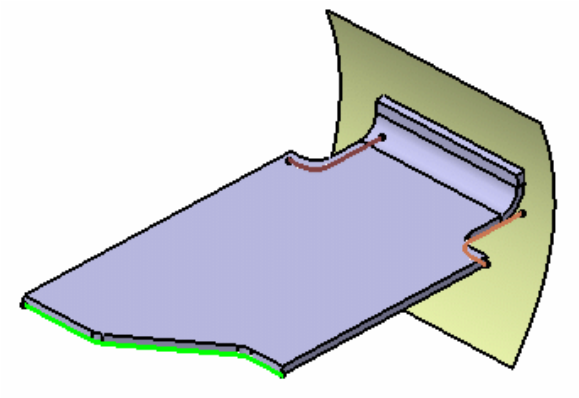


- a length from OML (Outer Mold Line): length between the curve defining the top of the surfacic flange and the OML,
- an element FD (Folded): boundary element (either a surface that intersects with its surface, or a sketch, or a wire projected on its surface),
- an element FP (Flat Pattern): curve or sketch defining the flattened contour of the surfacic flange.

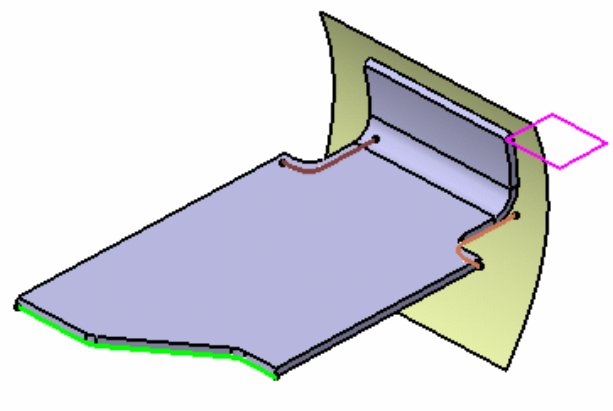
   The element FP **must** be included within the limits of the surfacic flange support when folded.

7. Click OK.

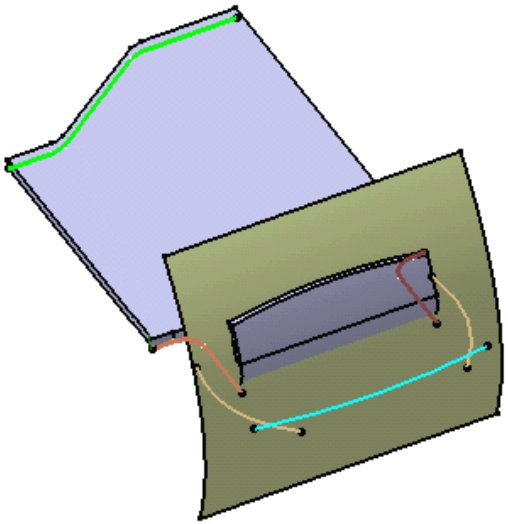
The Surfacic Flange (identified as Surfacic Flange.xxx) is created and the specification tree is updated accordingly.



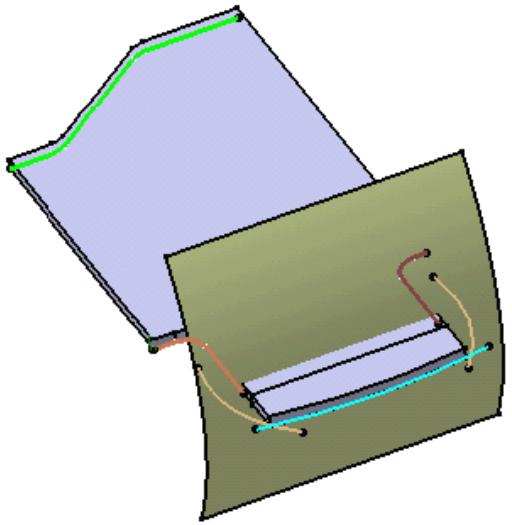
Surfacic Flange with a length from OML of 15 mm



Surfacic Flange with Plane.2 (in pink) as Element FD



Surfacic Flange with EOP FP (in light blue) as Element FP

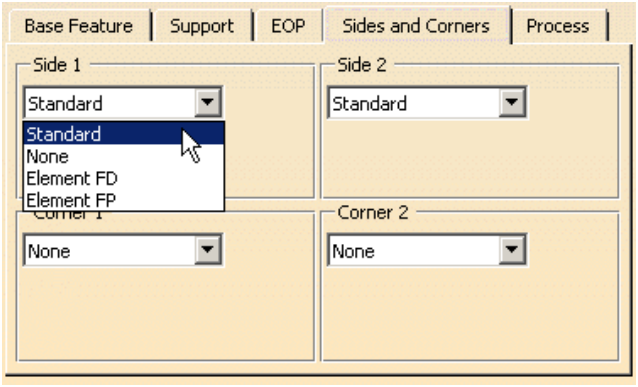


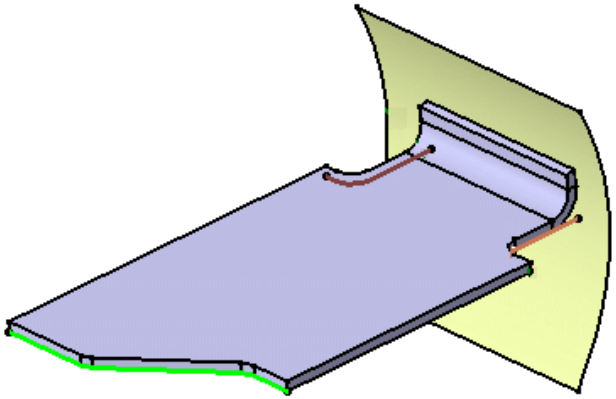
Unfolded view of the Surfacic Flange with EOP FP as Element FP.
See Unfolding.

Sides and Corners

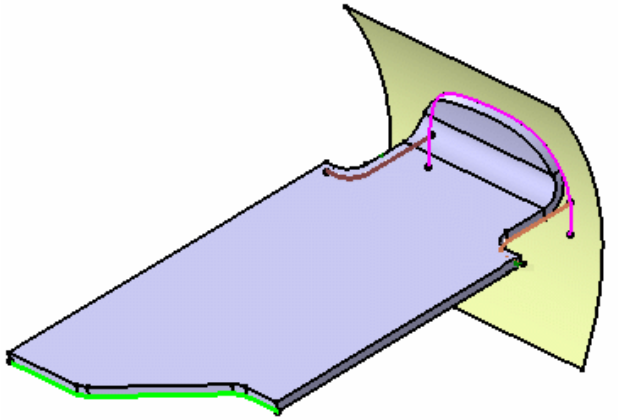
In the Sides and Corners tab, you can choose to define the following elements:

- **sides** (intersection between the Base Feature and a curve) as:
  - standard: they are automatically defined at the web limit and the perpendicular plans are kept (in this case, the user does not have to define them)
  - none: no side computed (only the EOP will define the contour of the Flange)
  - element FD (Folded): they are defined by a folded geometrical element (curve, plane or surface).
  - element FP (Flat Pattern): curve defining the flattened contour of the flange.

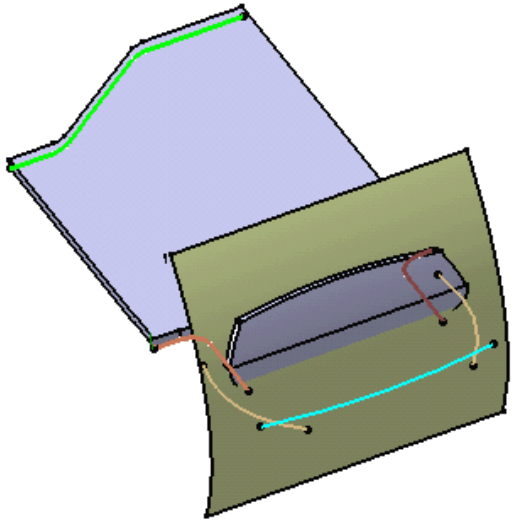




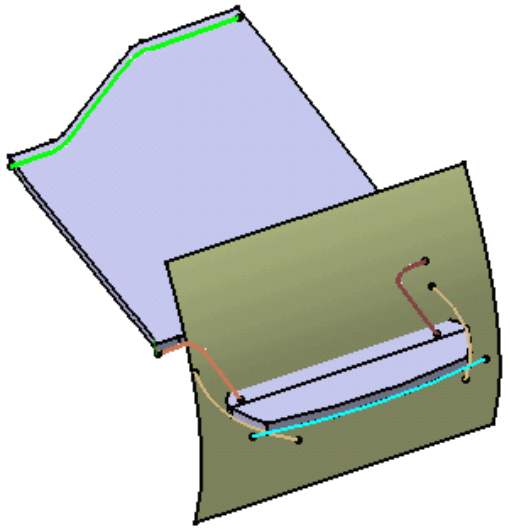
Surfacic Flange defined with a Length from OML of 10mm, and Side 1 and 2 as Standard



Surfacic Flange defined with EOP FD as Element FD, and Side 1 and 2 as None

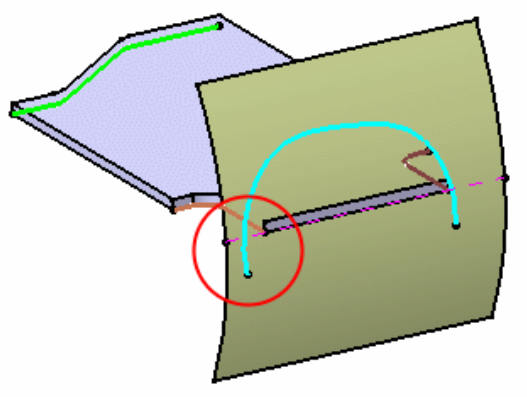


Folded view of the Surfacic Flange with EOP FP as Element FP, Side 1 FP and Side 2 FP (in light brown) as Side 1 and Side 2.
See Unfolding.

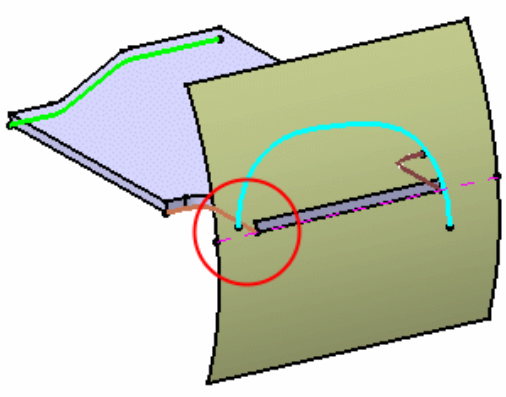


Unfolded view of the Surfacic Flange with EOP FP as Element FP, Side 1 FP and Side 2 FP (in light brown) as Side 1 and Side 2.
See Unfolding.

a. The following examples show two cases of a flange defined by an EOP FP or FD and Sides as None.

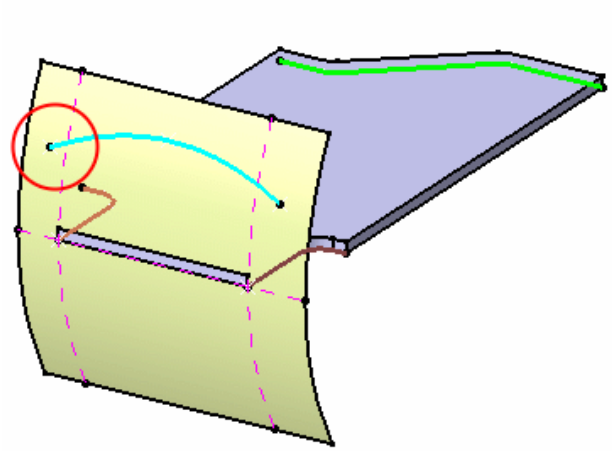


There is an intersection between the EOP and the web support -> the Surfacic Flange can be computed

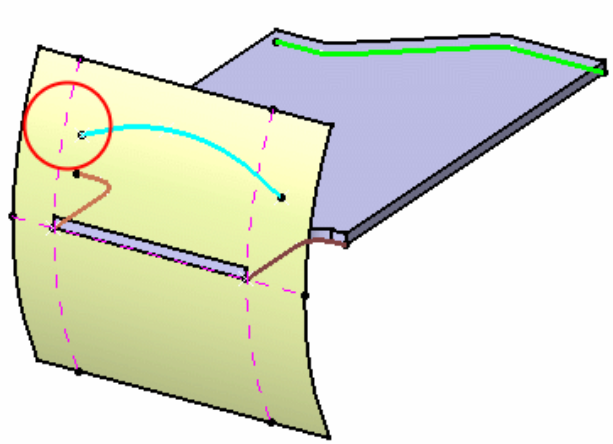


There is no intersection between the EOP and the web support -> the Surfacic Flange cannot be computed

b. The following examples show two cases of a flange defined by an element FD as the EOP and Standard sides.

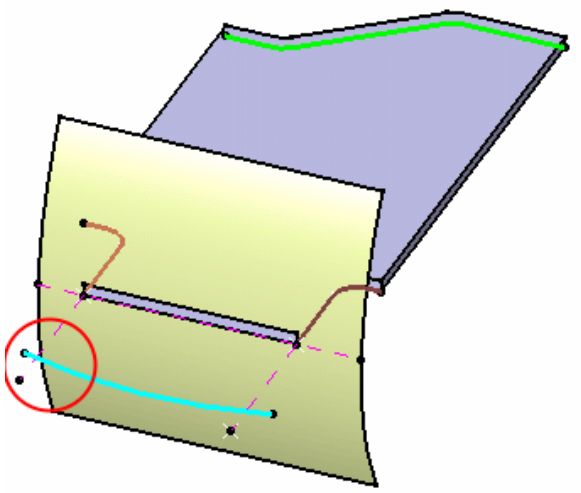


There is an intersection between the EOP and the side -> the Surfacic Flange can be computed

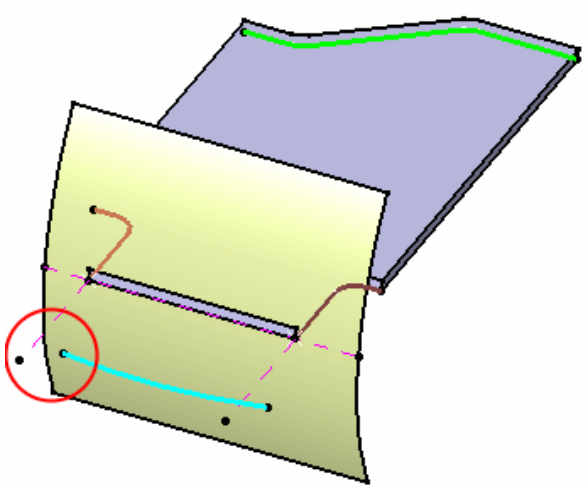


There is no intersection between the EOP and the side -> the Surfacic Flange cannot be computed

c. The following examples show two cases of a flange defined by an element FP as the EOP and Standard sides.

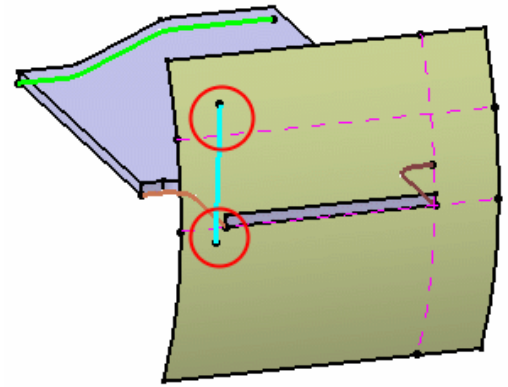


There is an intersection between the OML and the EOP -> the Surfacic Flange can be computed

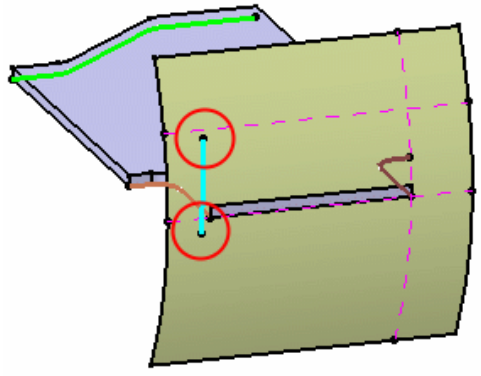


There is no intersection between the OML and the EOP -> the Surfacic Flange cannot be computed

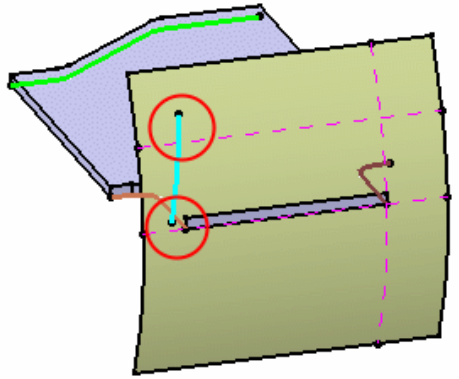
d. The following examples show three cases of a flange defined by an element FD as the side.



There is an intersection between the Element FD and the OML and between the side and the EOP -> the Surfacic Flange can be computed

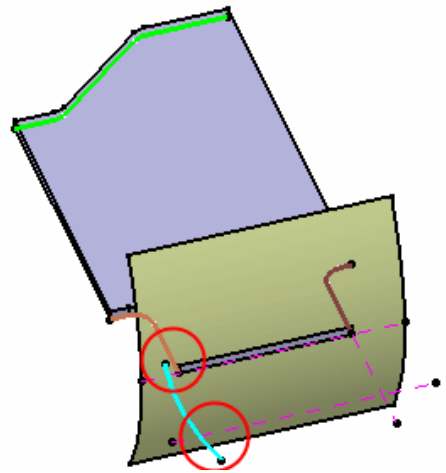


There is an intersection between the side and the OML but no intersection between the side and the EOP -> the Surfacic Flange cannot be computed

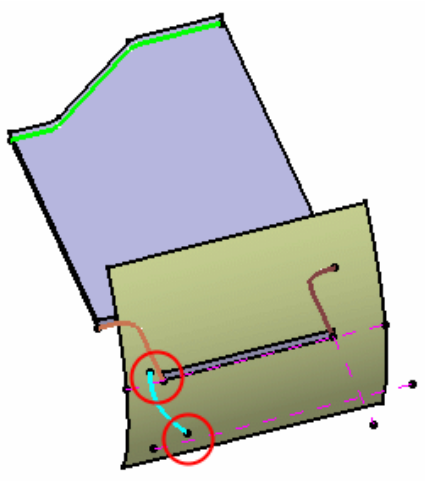


There is an intersection between the side and the EOP but no intersection between the side and the OML -> the Surfacic Flange cannot be computed

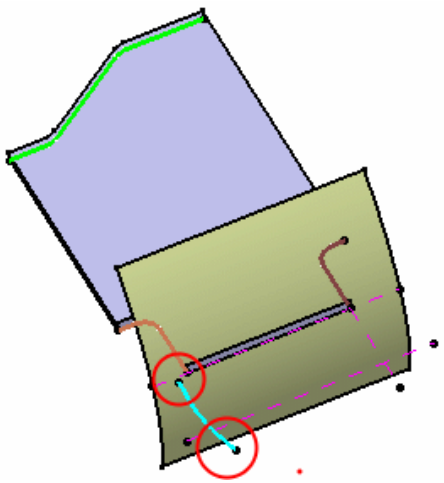
e. The following examples show three cases of a flange defined by an element FD as the side.



There is an intersection between the side and the EOP and between the side and the OML -> the Surfacic Flange can be computed

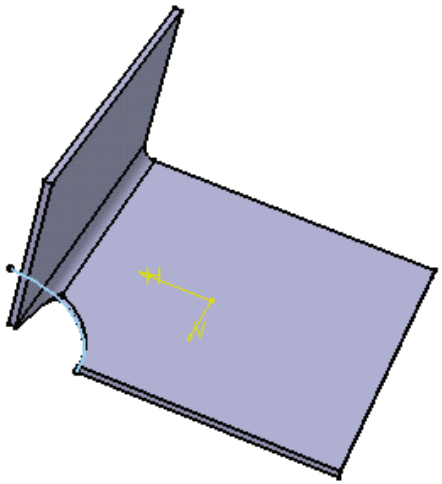


There is an intersection between the side and the OML but no intersection between the side and the EOP -> the Surfacic Flange cannot be computed

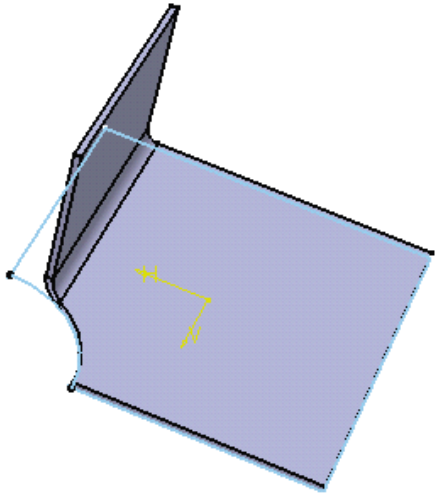


There is an intersection between the side and the EOP but no intersection between the side and the OML -> the Surfacic Flange cannot be computed

Standard sides are calculated on the first profile of a web that did not undergo any modifications. Any modification on the web will not be taken into account to create the surfacic flange's sides. For example, if you create a cutout on the web, then create a surfacic flange with standard sides, the latter will be calculated from the web's profile without the cutout (please note that the cutout's role is not to redefine the web)
For optimization reasons, we advise you to first create a sketch with the desired shape, then create the surfacic flange.

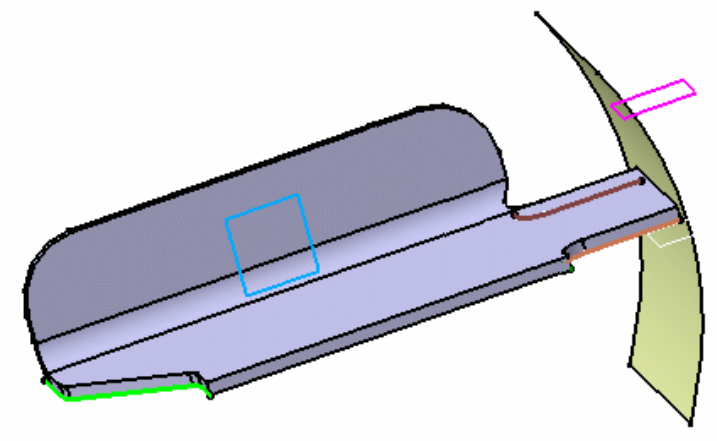


The example above shows a Surfacic Flange with standard sides calculated from the web modified by a cutout (the cutout, shown in blue, is ignored)



The example above shows a Surfacic Flange with standard sides calculated from the web defined entirely by a sketch (shown in blue)

- **corners** (profile defined between the EOP and the sides) as:
  - none: no corner computed (only the EOP is able to define the contour of the Surfacic Flange)
  - corner: between the side and the EOP (defined with a radius value)



The example above shows a Surfacic Flange defined with Sides 1 and 2 as Standard, and Corners 1 and 2 of 10mm each.

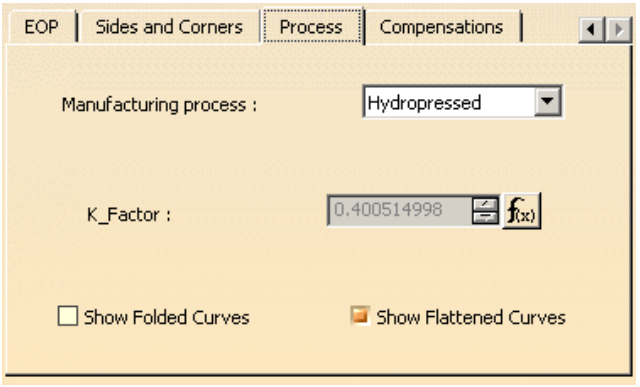
Process

In the Process tab, you can define the:

- Manufacturing process:
  - Hydropressed
  - BrakeFormed

- K_Factor: you can modify the K Factor as defined in the Sheet Metal Parameters dialog box by changing the driving equation.
  Click the f(x) icon. The Formula Editor dialog box opens, you can modify the dictionary and the parameters. Or you may need to deactivate the formula using the contextual menu on the field and choosing **Formula -> Deactivate** before editing the value.

You can also choose to display the characteristic curves either on the folded view (**Show Folded Curves**), and/or on the flattened view (**Show Flattened Curves**) of the part.



Compensations

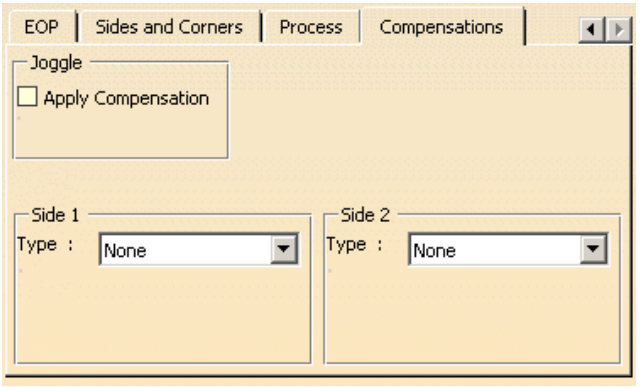
In the Compensations tab, you can define compensations for the:

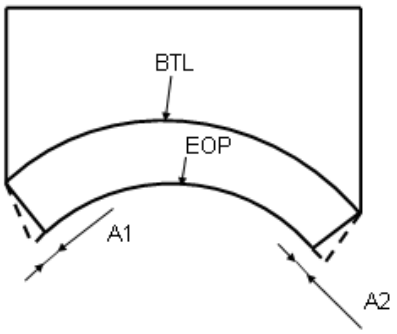
- Joggle:
  - check the **Apply Compensation** button when creating or editing the joggle.
    See Creating a Joggle for further information.

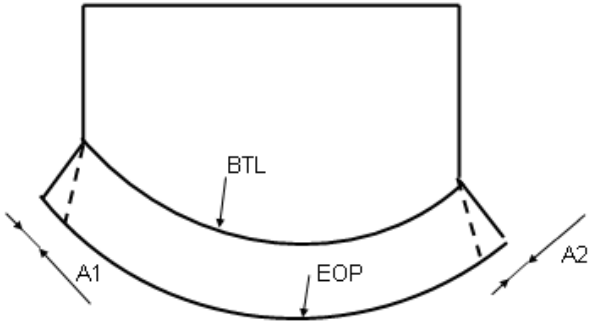
- Flange Sides (Side and Side 2).
  - Define the type:
    - **None**: no compensation is applied
    - **Manual: Angle**: the deformation is computed according to an angle
    - **Manual: Length**: the deformation is computed according to a length parallel to the BTL.
  - Define the **Angle** in the case of a **Manual: Angle** compensation. A negative angle adds

material, and a positive angle removes material.

- Define the **Length** in the case of a **Manual: Length** compensation

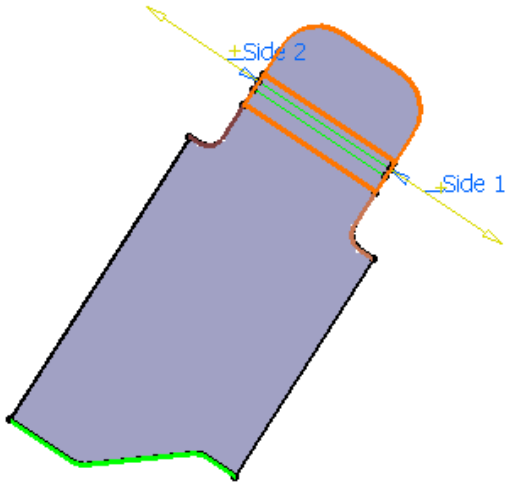


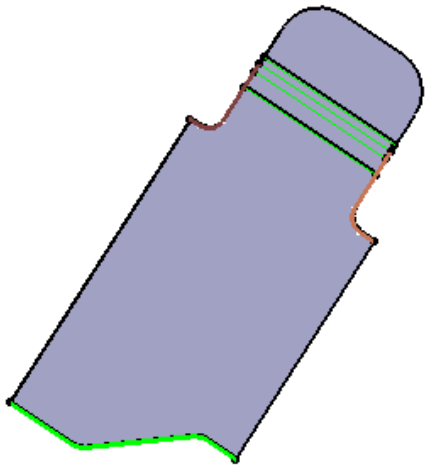




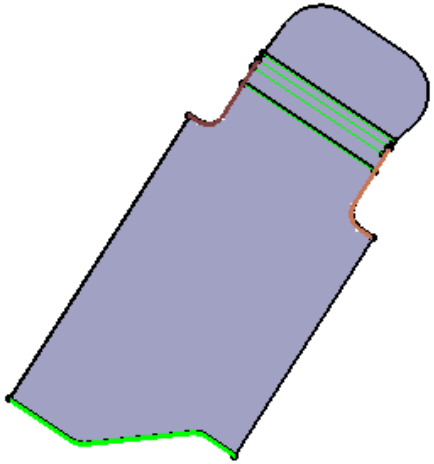
The values of the modification are the angles A1 and A2.

- Compensations can be created either on the folded or flattened part, but they only apply on the flattened part.
- Compensations can be modified independently on each flange.
- The + sign (in yellow in the 3D geometry) shows that material will be added to the sides.

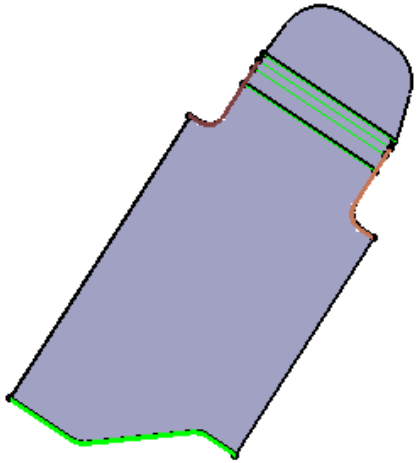




Unfolded Surfacic Flange defined with Corners 1 and 2 of 10mm each, and no compensation for Side 1 and Side 2



Unfolded Surfacic Flange defined with Corners 1 and 2 of 10mm each, a Manual: Angle compensation of -20deg for Side 1 and no compensation for Side 2



Unfolded Surfacic Flange defined with Corners 1 and 2 of 10mm each, a Manual: Angle compensation of 20deg for Side 1 and -10deg for Side 2

- In the case the user does not define a flange side, the latter is automatically computed at the Web limit, perpendicular to the OML.
- In the case no corner is defined, the side and the EOP are simply relimiting each other
- The sides of the fillet are continuous in tangency with the contour of the web and the sides of the flange.

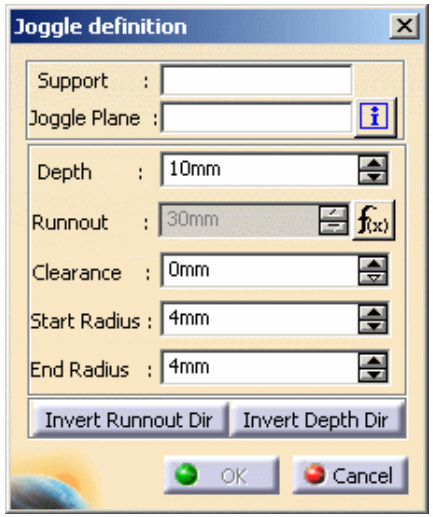
Creating a Joggle

This task explains how to create a joggle, that is a feature which causes the flange to be locally deformed.

The joggle is a feature which cannot exist alone, it is always defined on a flange.

Open the Joggle1.CATPart document.
Create a flange as shown in the previous task.

1. Click the **Joggle** icon .



2. Select the flange as the support.

The **Support** of the joggle is not automatically set to the last created flange.

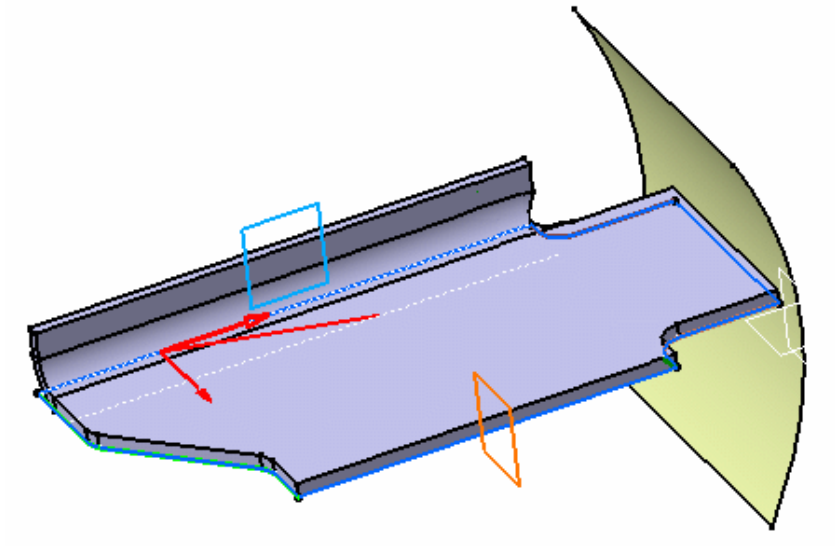
3. Choose a plane as the joggle plane, here we choose Plane3.

   The blue curve defines the boundary of the web.

The vectors show you the joggle directions:

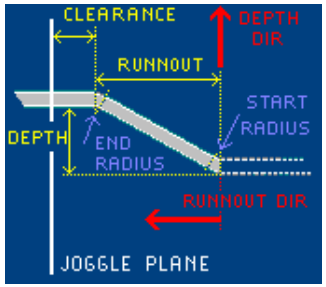
- The vector on the flange support determines the depth direction

- The vector on the joggle plane determines the side on which the joggle is to be created



In case there are several intersections between the flange and the plane, the closest intersection is chosen.

You can click the icon to display a schema showing the joggle parameters to be defined.



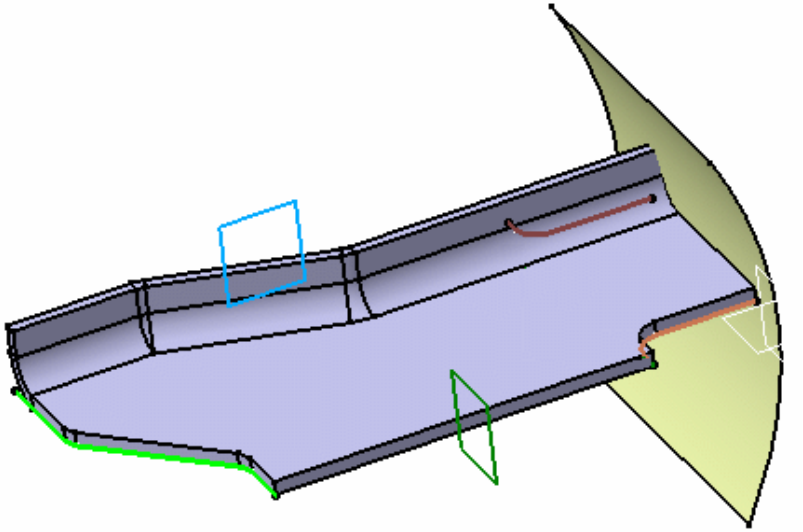
4. You can modify the following parameters of the joggle by clicking the up and down arrows.
   - depth: offset from the support surface (shown as dotted lines)

The dotted lines must remain inside the blue curve.

   - runout: length of the offset, between the original surface of the flange and the new surface (joggle)
   - clearance: length added to the offset at the joggle starting plane
   - start radius: fillet between the runout and the flange
   - end radius: fillet between the runout and the offset

5. Click OK.

The joggle (identified as Joggle.xxx) is created and the specification tree is updated accordingly.



- If you modify the depth, the runout adjusts automatically.
- You can reverse the runout direction either by clicking the red arrow or by clicking the **Invert Runout Dir** button in the dialog box.
- You can reverse the depth direction either by clicking the red arrow or by clicking the **Invert Depth Dir** button in the dialog box.

Applying Compensations

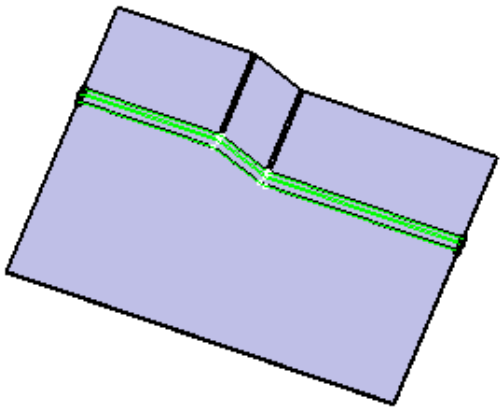
You can apply compensations when creating the flange or editing the joggle.

Compensations can be created either on the folded or flattened part, but they only apply on the flattened part.

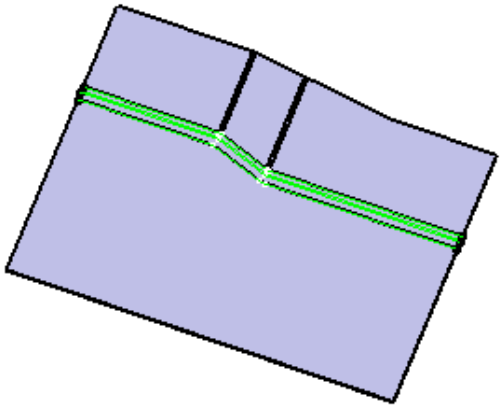
Open the Joggle2.CATPart document for a single joggle, Joggle.3CATPart for a twin joggle, and Joggle4.CATPart for a double joggle.
Standard Files must have been previously imported and a method for compensations defined.

1. Double-click the **flange** supporting the joggle to edit it.
2. In the Compensations tab, click the **Apply Compensations** button.

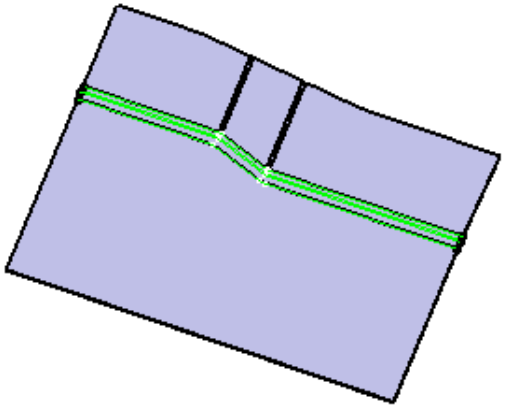
On a single joggle



Unfolded single joggle without compensations

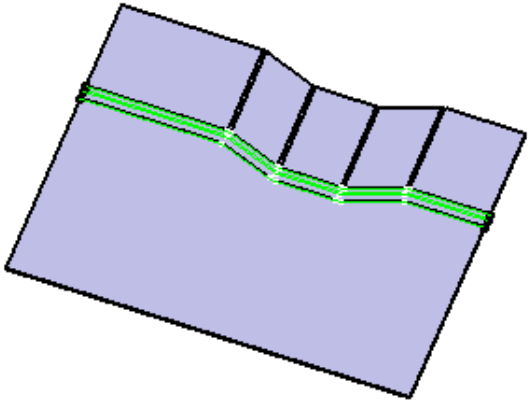


Unfolded single joggle with compensations relying on Method 1

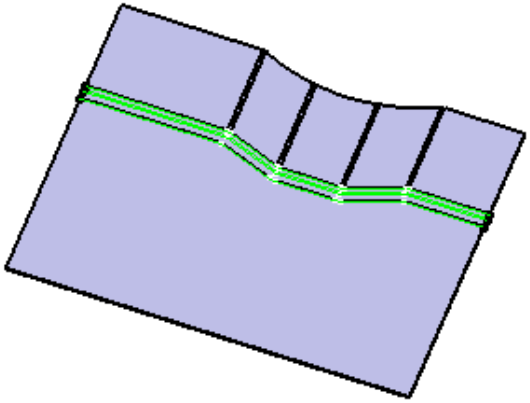


Unfolded single joggle with compensations relying on Method 2

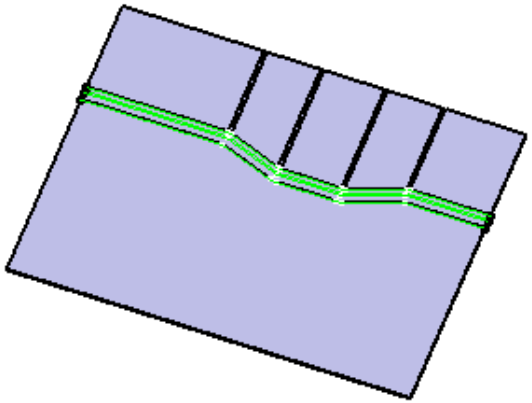
On a twin joggle



Unfolded twin joggle without compensations

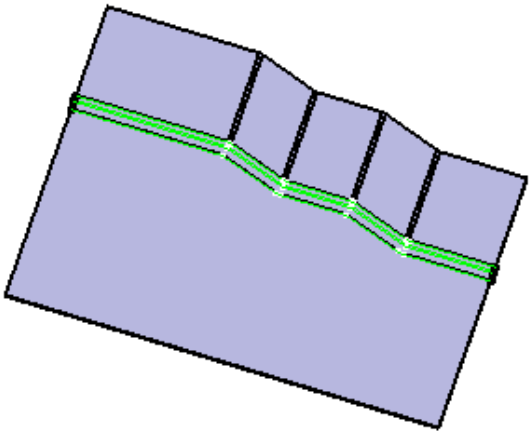


Unfolded twin joggle with compensations relying on Method 1

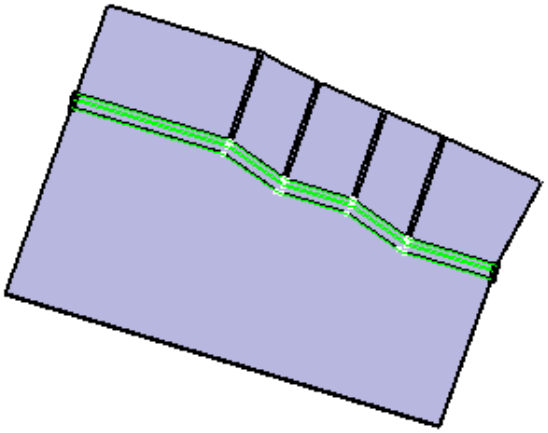


Unfolded twin joggle with compensations relying on Method 2

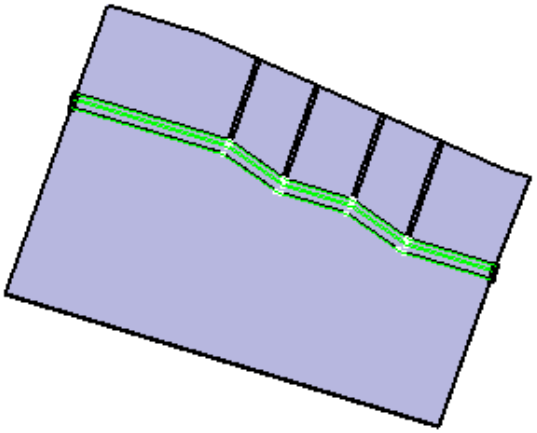
On a double joggle



Unfolded double joggle without compensations



Unfolded double joggle with compensations relying on Method 1



Unfolded double joggle with compensations relying on Method 2

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Getting Started

Before getting into the detailed instructions for using Version 5 CATIA - Aerospace Sheet Metal Design, the following tutorial provides a step-by-step scenario demonstrating how to use key functionalities.

The main tasks proposed in this section are:

Entering the Aerospace SheetMetal Design Workbench
Defining the Aerospace SheetMetal Parameters
Creating a Web from a Sketch
Creating a Surfacic Flange on a Web
Creating a First Cutout
Extracting Drawings from the Aerospace SheetMetal Design Part

 All together, these tasks should take about 20 minutes to complete.

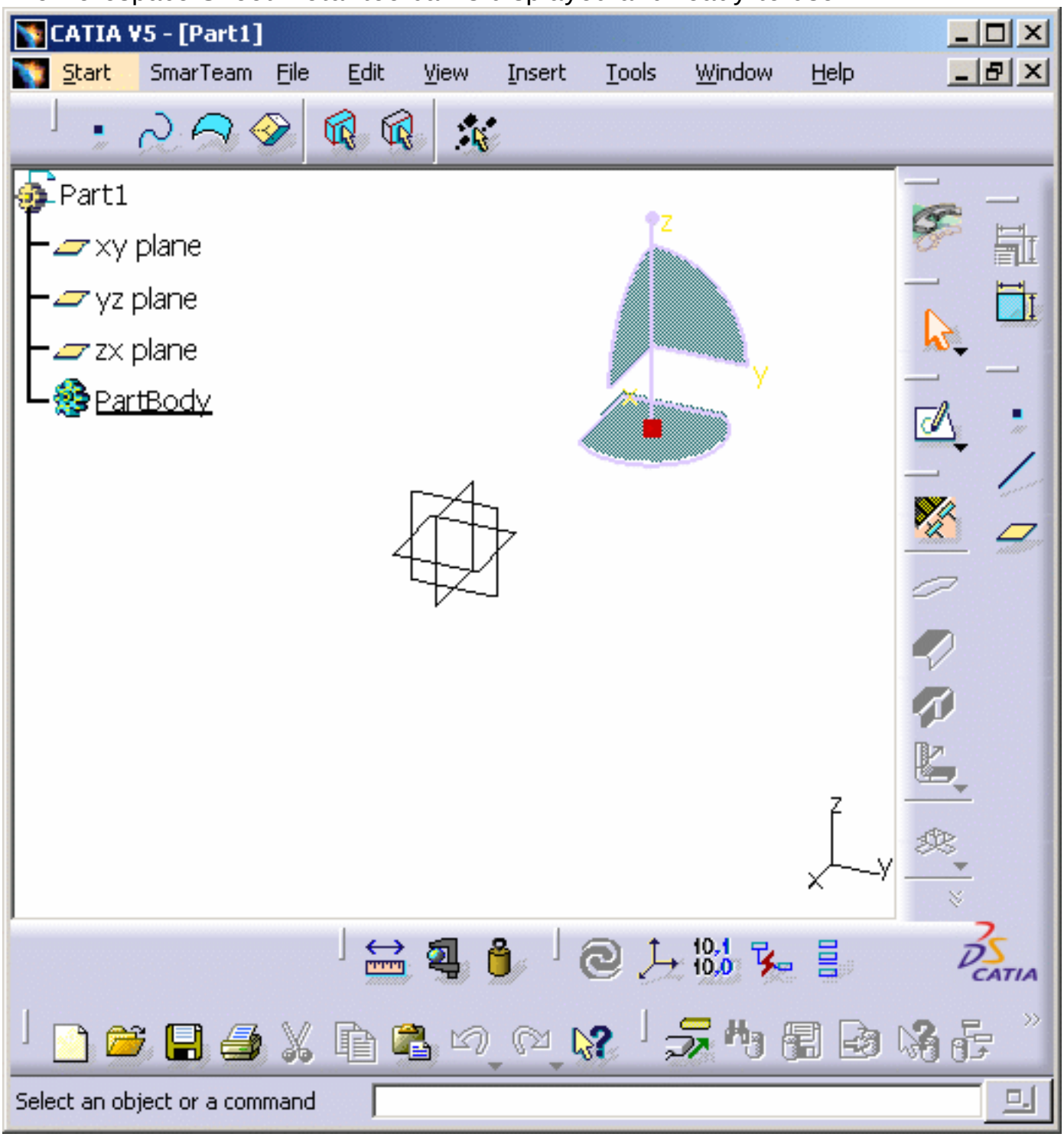
Entering the Aerospace SheetMetal Design Workbench

The Aerospace Sheet Metal Design functions are available when you are in the Part environment. Several functions are integrated from the Part Design workbench.

This task shows how to enter the workbench.

Choose the **Mechanical Design -> Aerospace Sheet Metal Design** item from the **Start** menu.

The Aerospace Sheet Metal toolbar is displayed and ready to use.





You may add the **Aerospace Sheet Metal Design** workbench to your Favorites, using the **Tools -> Customize** item. For more information, refer to the *Infrastructure User's Guide.*

If you wish to use the whole screen space for the geometry, remove the specification tree clicking off the **View -> Specifications Visible** menu item or pressing F3.

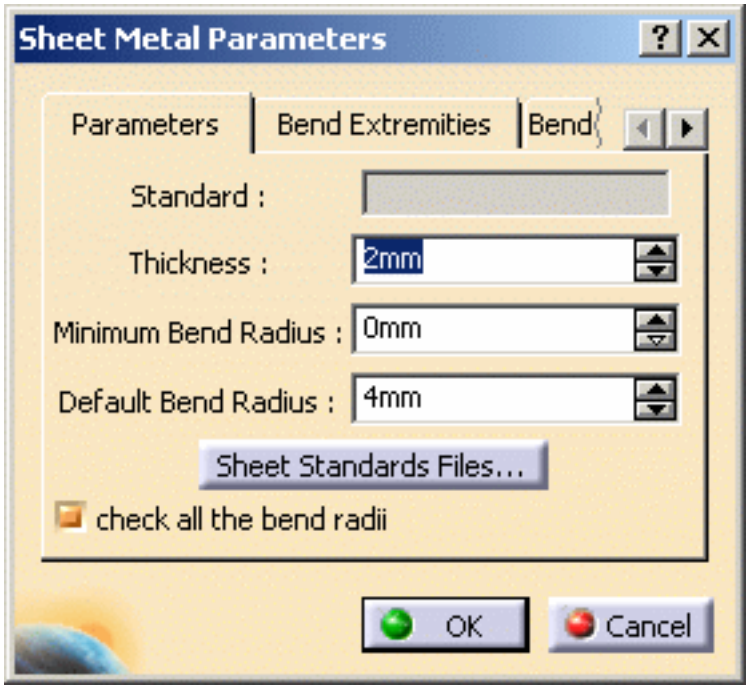
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Defining the Aerospace SheetMetal Design Parameters

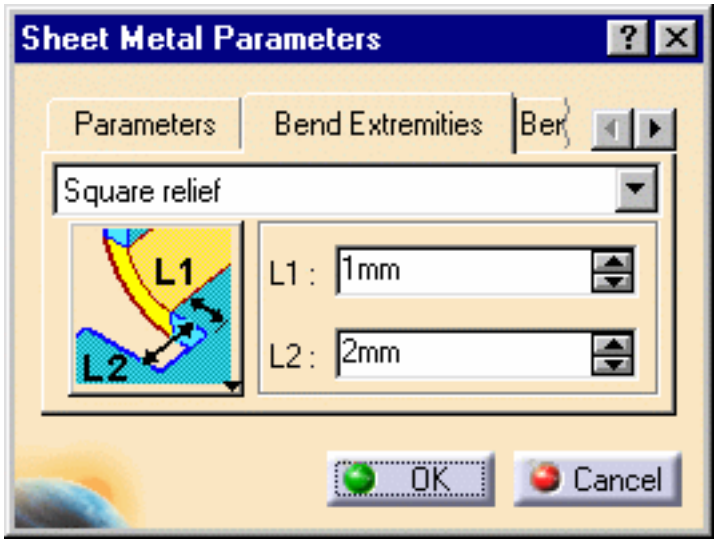
This task shows you how to configure the Aerospace SheetMetal parameters.

1. Click the **Sheet Metal Parameters** icon .

   The Sheet Metal Parameters dialog box is displayed.



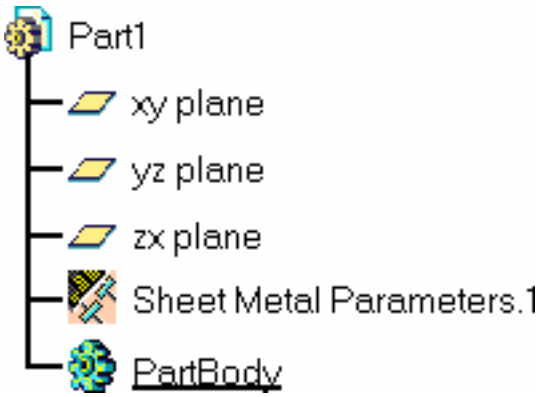
2. Enter 1mm in the **Thickness** field.
3. Enter 5mm in the **Default Bend Radius** field.
4. Select the **Bend Extremities** tab.



5. Select **Tangent** in the Bend Extremities combo list.

An alternative is to select the bend type in the graphical combo list.

6. Click **OK** to validate the parameters and close the dialog box.
   The **Sheet Metal Parameters** feature is added in the specification tree.



The other two tabs are not used in this scenario.

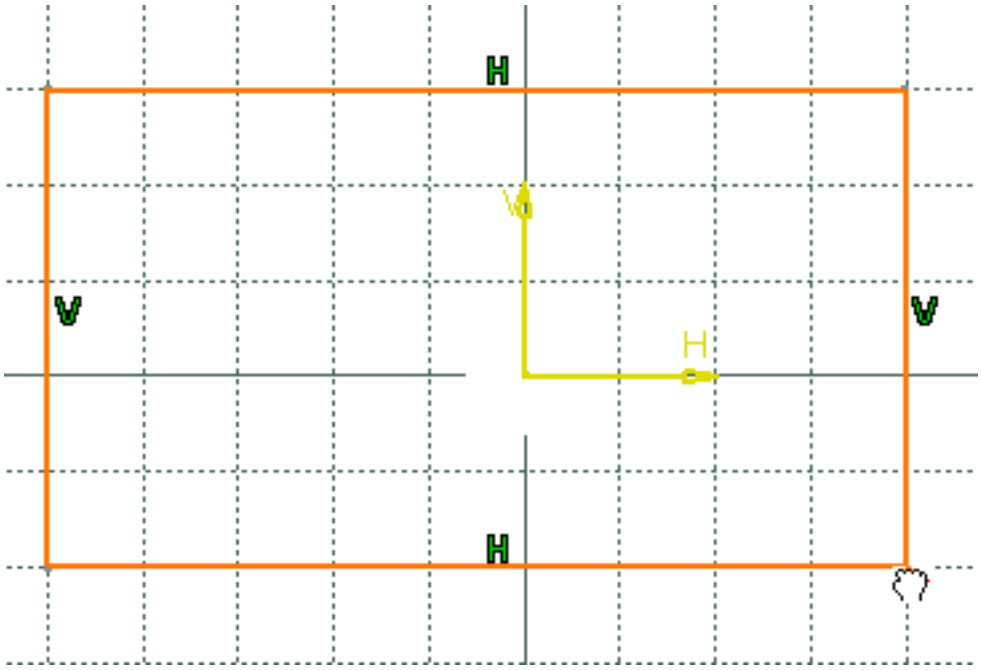
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Creating a Web from a Sketch

This section explains how to create a web.

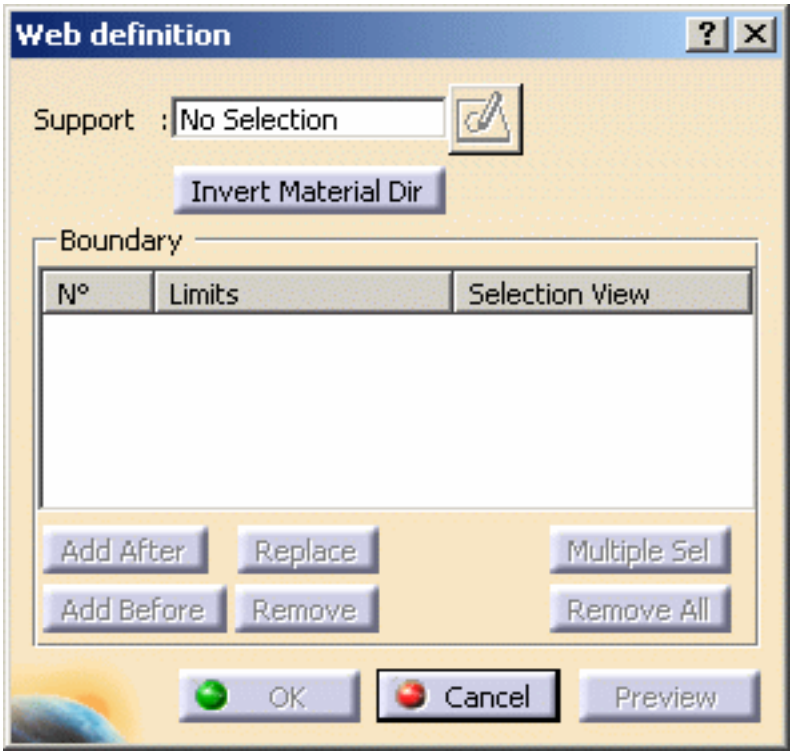
The web is the main feature of an Aerospace Sheetmetal part: there is always one (and only one) web.

1. Click the **Sketcher** icon .
2. Select the xy plane.
3. Click the **Rectangle** icon in the Profile toolbar to create the contour of the web.
4. Click to create the first point and drag the cursor.
5. Click to create the second point: the rectangle profile is displayed.
6. Click the **Exit workbench** to return to the 3D world.

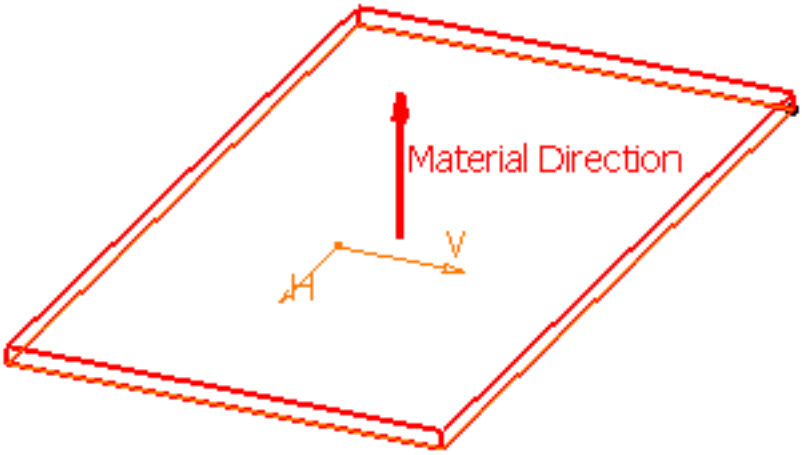


7. Click the **Web** icon .

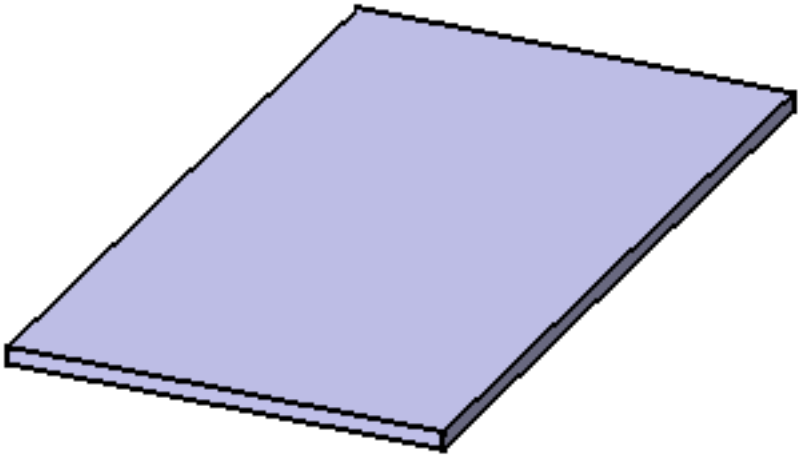
   The Web definition dialog box is displayed.



8. Select the sketch you just created as the support of the web. A preview of the web appears.
9. Click OK to create the web.



Here is the web.



You can click the **Sketcher** icon  to edit the sketch.



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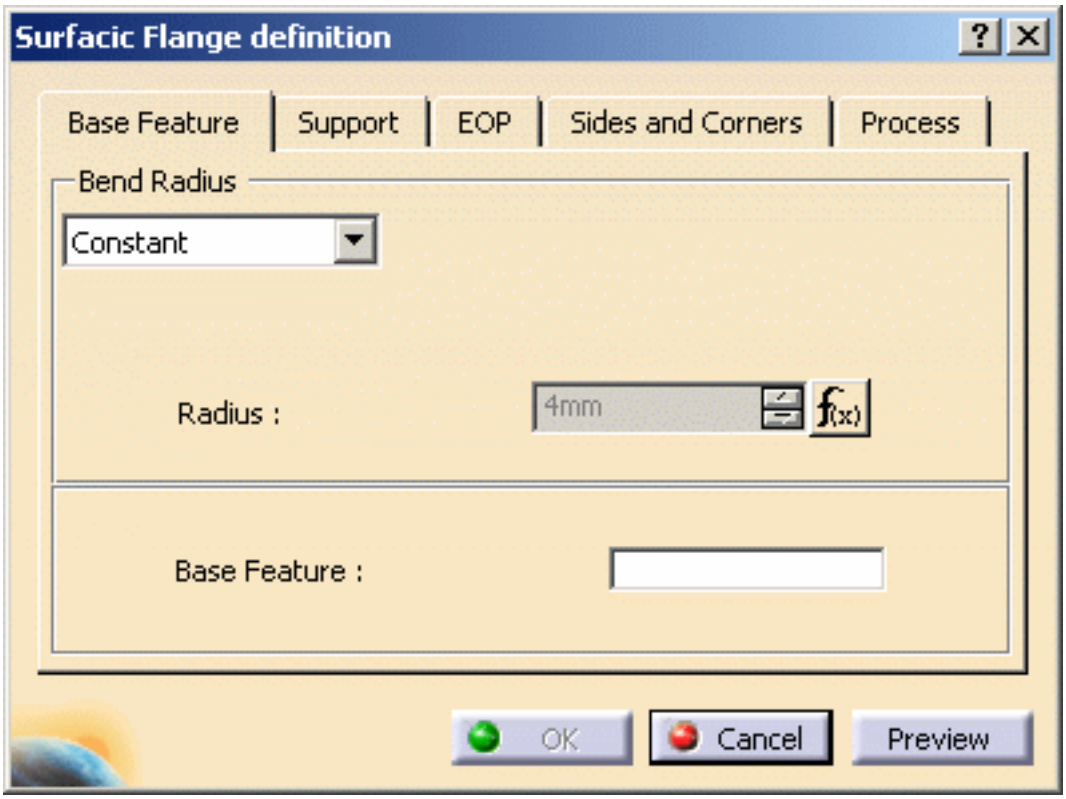
Creating a Surfacic Flange on a Web

This section explains how to create a flange on a web, that is a feature which enables to stiffen the part.

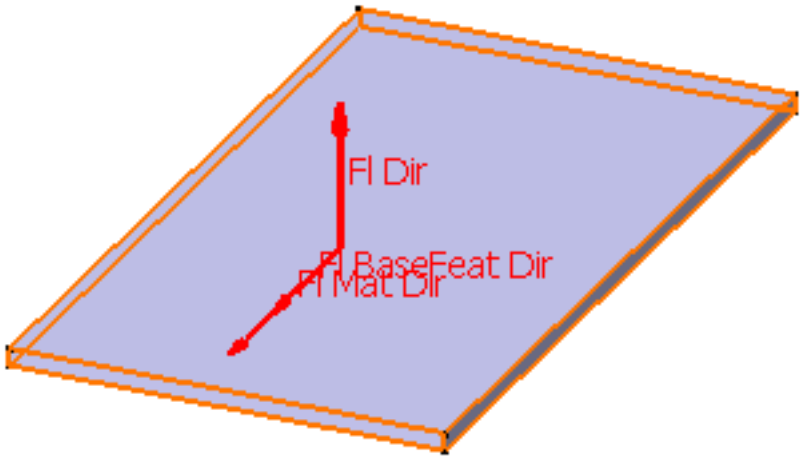
The web is still open from the previous task.

1. Click the **Surfacic flange** icon .

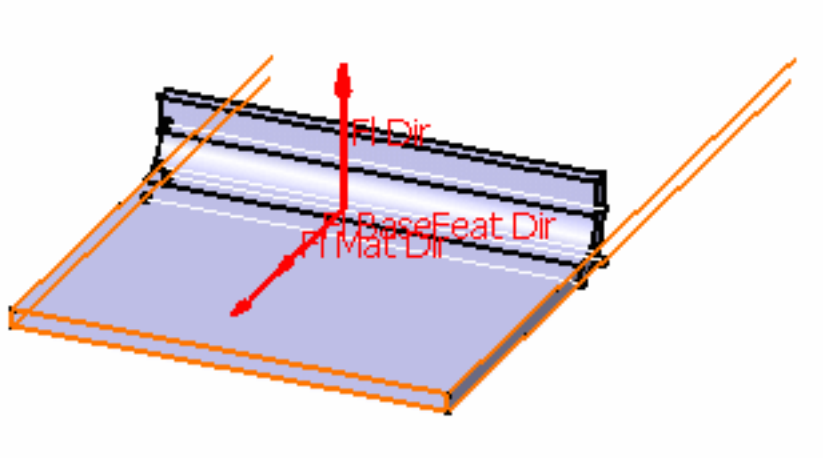
   The Surfacic Flange definition dialog box is displayed.



2. Choose the web as the Base Feature.
3. Choose the yz plane as support.

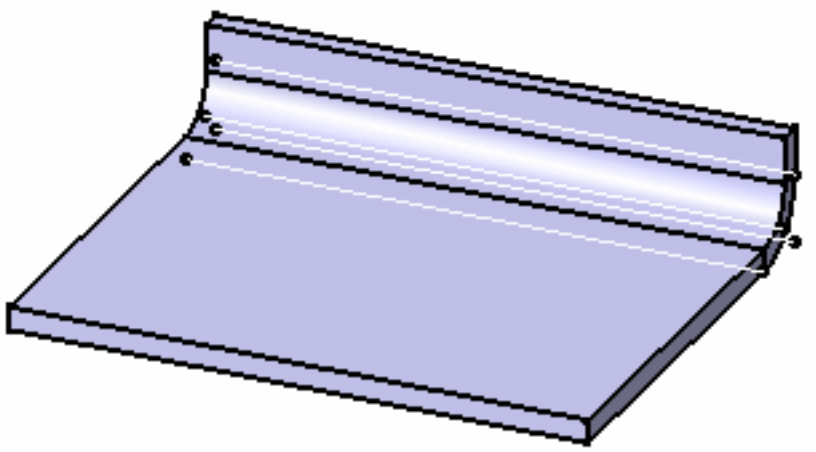


4. Click **Preview** to see the surfacic flange.



5. Click OK to create the surfacic flange.

   Here is the surfacic flange.





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Creating a CutOut

In this task, you will learn how to:

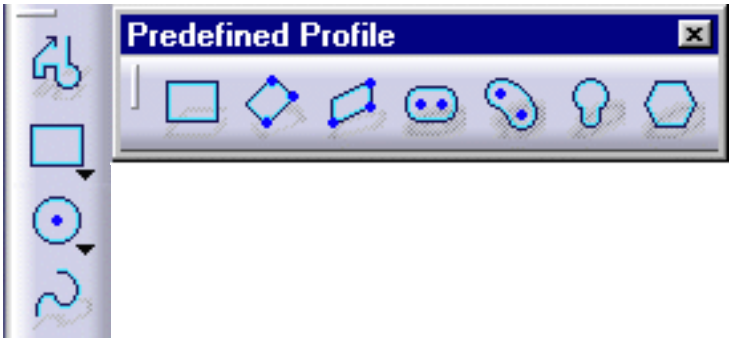
- open a sketch on an existing face
- define a contour on the face in order to create a cutout.

You can create a cutout defined either by a sketch or an open geometry.

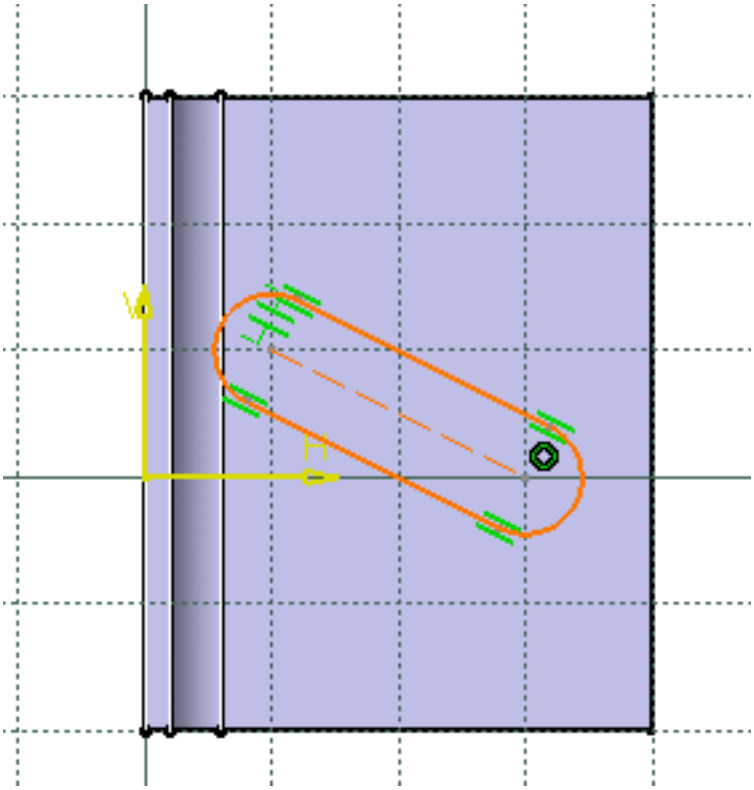
The flange is still open from the previous task.

1. Select the surface from the geometry area to define the working plane.
2. Click the **Sketcher** icon .
3. Click the **Elongated Hole** icon to create the contour.

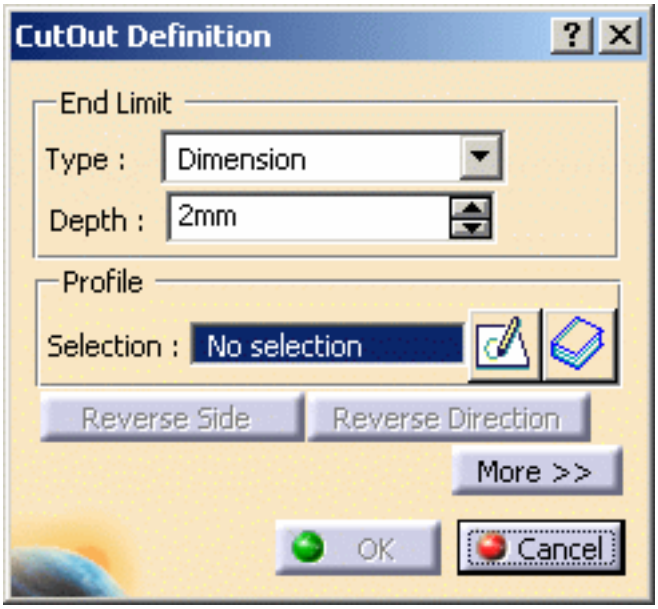
To access the oblong profile, click the black triangle on the **Rectangle** icon.
It displays a secondary toolbar.



4. Click to create the first point and drag the cursor.
5. Click to create the second point.
   The first semi-axis of the profile is created.
6. Drag the cursor and click to create the third point.
   The second semi-axis is created and the oblong profile is displayed.

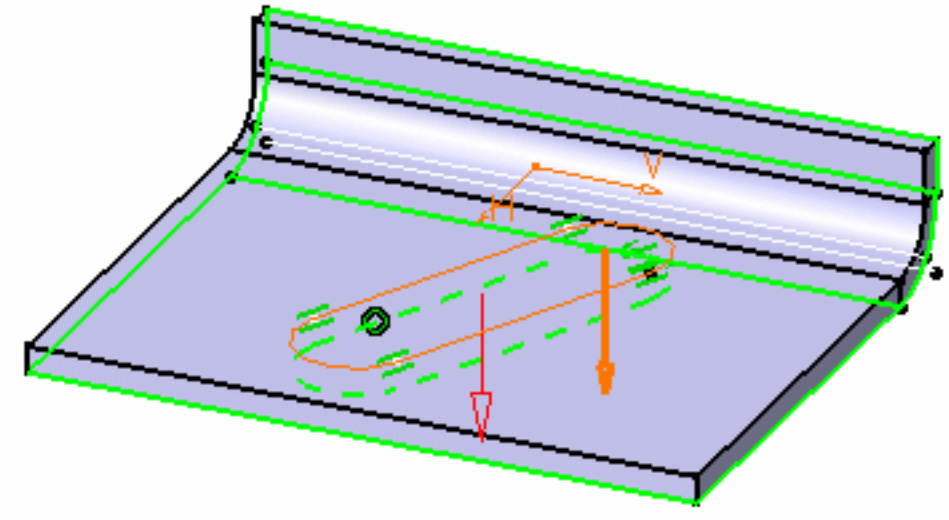


7. Click the **CutOut** icon .



The CutOut Definition dialog box is displayed and a cutout is previewed with default parameters.

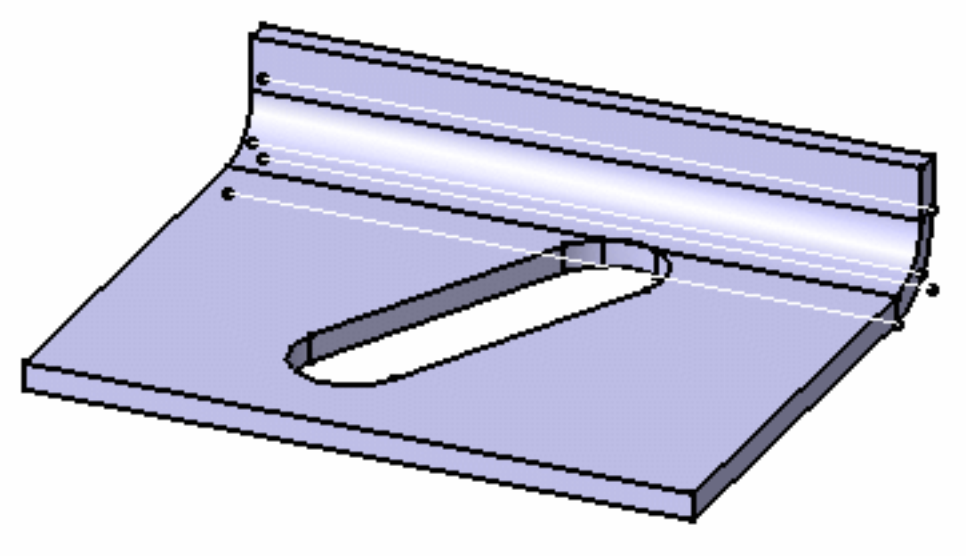
The vectors show the side and the direction of the cutout.



8. Select the **Dimension** type to define the limit of your cutout.

9. Click Ok.

   Here is your cutout.



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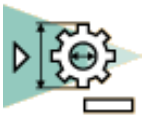
Extracting Drawings from the Aerospace SheetMetal Design Part

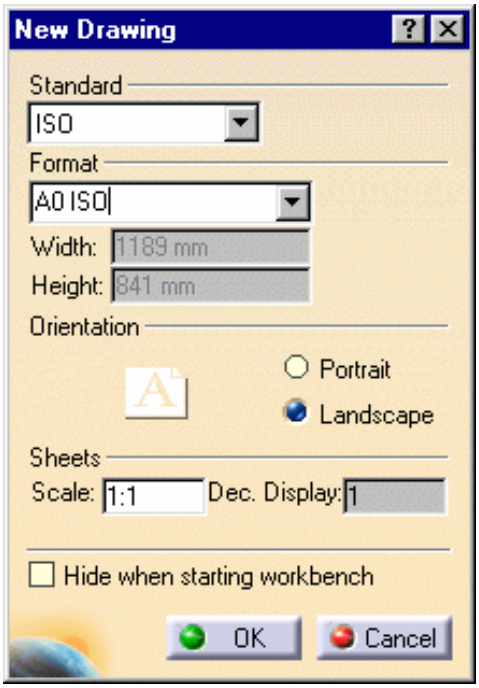
This task shows how to create the Aerospace SheetMetal Design Part views in the Generative Drafting workbench.

The Sheet Metal part is displayed.

1. Click  or select **File -> New...**
2. Select **the Drawing** type and click OK.

   The Generative Drafting workbench is launched.
   The New Drawing dialog box opens.





3. Click OK.

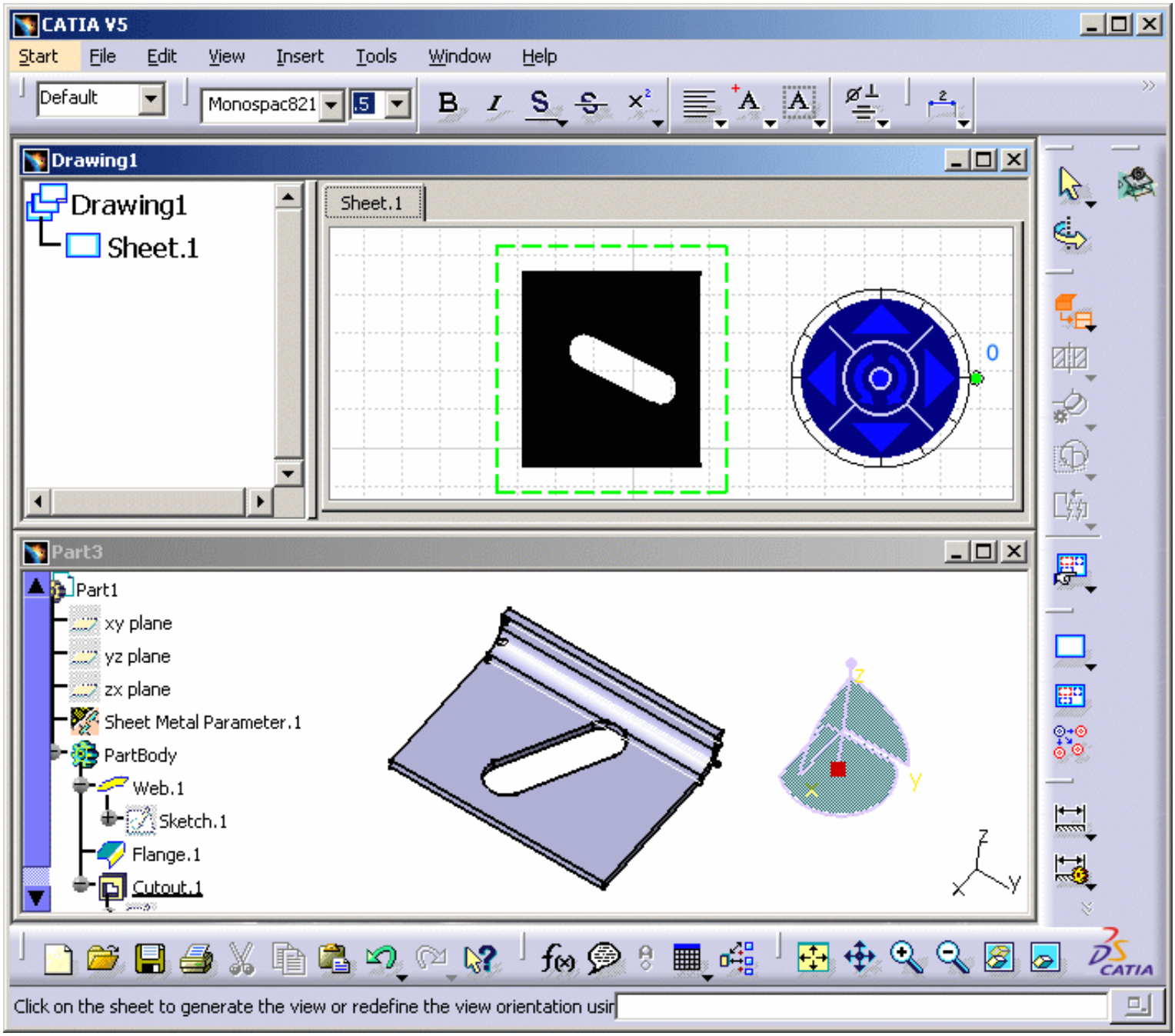
   For more information about this workbench, refer to *Generative Drafting User's Guide*.

4. The drawing sheet appears.
5. Tile the windows horizontally using the **Window -> Tile Horizontally** menu item.
6. Select the Unfolded View icon  in the Projections toolbar from Generative Drafting Workbench.

   This icon is added to the Projections toolbar provided the Sheet Metal workbench is present.

7. Choose the **xy plane** in the Sheet Metal specification tree.
   The unfolded view is displayed with the bends axes and limits.

Eventually, the Drafting sheet looks like this:



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User Tasks

Managing the Default Parameters
Creating a Web
Creating a Surfacic Flange
Creating a Joggle
Creating Swept Walls
Unfolding
Creating a CutOut
Creating a Hole
Creating Stamping Features
Creating a Local Corner Relief
Creating Constraints
Mapping Elements
Creating Corners
Creating Chamfers
Patterning
Reference Elements
Displaying Characteristic Curves
Looking For Aerospace SheetMetal Features
Browsing the Sheetmetal Catalog
Integration With Part Design

Managing the Default Parameters

This section explains and illustrates how to use or modify various kinds of features. The table below lists the information you will find.


Using Aerospace Sheetmetal Design assumes that you are in a CATPart document.


Edit the parameters: select the Parameters tab and define the element thickness and bend radius values.


Compute the bend allowance: select the Bend Allowance tab and define the allowance value (K factor).


Define the compensations: select the Compensations tab ans define the compensations for the joggle and the sides.

Please refer to the Customizing chapter to define the Sheet Standards Files.

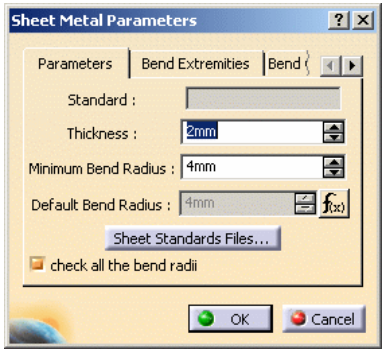
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Editing the Sheet and Tool Parameters

This section explains how to change the different sheet metal parameters needed to create your first feature.

1. Click the **Sheet Metal Parameters** icon .

   The Sheet Metal Parameters dialog box is displayed.



2. Change the **Thickness** if needed.

3. Change the **Minimum Bend Radius** if needed.
   The Minimum Bend radius defines the minimum internal radius allowing the creation of a bend. If the bend has a bend radius smaller than the one defined here, it will not be created.

4. Change the **Default Bend Radius** if needed.
   The Default Bend Radius corresponds to the internal radius and is linked by default to the creation of the flanges.

You can set the value to 0 to create bend with no radius. If using the DIN standard, the KFactor automatically sets to 0 as well.

Convention dictates that the inner angle between the two elements is used to define the bend.
It can vary from 0° to 180° exclusive. This angle is constant and the bend axis is rectilinear.

5. Click **OK** to validate the Sheet Metal Parameters.

The Standard field displays the Standard to use with the part, if implemented. The name of this standard file is defined in a Design Table.

When the **Check all the bend radii** button is checked, and you click OK in the Sheet Metal Parameters dialog box, existing bend radii are checked and a list displays flanges or bends that do not use the minimum Bend Radius value as defined in step 3. Therefore, they will not be modified.

Parameters can be defined in a Design Table. To do so, press the **Sheet Standards Files...** button to access to the company defined standards, if need be. For more information, refer to the Customizing Standard Files section.

All parameters hereafter, or only some of them, can be defined in this Design Table:

Sheet Metal Parameters	**Column associated in the Design Table**	**Definition**
Standard in Sheet Metal Parameters	SheetMetalStandard	sheet reference name
Thickness	Thickness	sheet thickness
Minimum Bend Radius	MinimumBendRadius	minimum bend radius
Default Bend Radius	DefaultBendRadius	default bend radius
K Factor	KFactor	neutral fiber position
Radius Table	RadiusTable	path to the file with all available radii

Whenever both Radius Table and Default Bend Radius are defined in the Design Table, only the Radius Table will be taken into account for the bend creation.

Standard Names For Holes	**Column associated in the Design Table**	**Definition**
Clearance Hole	ClearanceHoleStd	path to the Clearance Hole Standard file
Index Hole	IndexHoleStd	path to the Index Hole Standard file
Manufacturing Hole	ManufacturingHoleStd	path to the Manufacturing Hole Standard file
Fastener Hole	FastenerHoleStd	path to the Fastener Hole Standard file
Standard Names For Stamps	**Column associated in the Design Table**	**Definition**
Flanged Hole	ExtrudedHoleStd	path to the Flanged Hole Standard file
Bead	BeadStd	path to the Bead Standard file
Circular Stamp	CircularStampStd	path to the Circular Stamp Standard file
Surface Stamp	SurfaceStampStd	path to the Surface Stamp Standard file
Flanged CutOut	FlangedCutoutStd	path to the Flanged CutOut Standard file
Curve Stamp	CurveStampStd	path to the Curve Stamp Standard file

When a parameter refers to a path, another sub-Design Table will be associated to the corresponding feature.

1. Here is an example for the use of a bend allowance table:

Main Sheet Metal Parameters Design Table

	A	B	C	D
1	SheetMetalStandard	Thickness (mm)	RadiusTable	
2	AG 3412	2	RadiusTableForThickness2.xls	
3	AG 3824	4	RadiusTableForThickness4.xls	
4				
5				

Radius Table For Thickness 2

This table defines available all bend radii for a thickness of 2 mm. A design table will be created on the Default Bend Radius of the Sheet Metal Parameters and on the Radius of each bend.

	A	B
1	BendRadius (mm)	BendTable
2	1	BendTableT2R1.xls
3	2	BendTableT2R2.xls
4	4	BendTableT2R4.xls
5		

Bend Table for Thickness 2 and Bend Radius 1

Whenever a bend is created, a radius table will be associated. If the configuration "Bend Radius = 1mm" is selected, a new design table (the Bend Table) will be created from BendTableT2R1.xls in order to compute the bend allowance.

According to the open angle, the bend deduction will be read in the Allowance column or interpolated if necessary.

	A	B	C
1	OpenAngle (deg)	Allowance (mm)	
2	25	-1.942	
3	90	-3.644	
4	160	-0.534	
5			

2. Here is an example for the use of a hole standard file:

Main Sheet Metal Parameters Design Table

	A	B	C	D	E	F	G	H	I
1	SheetMetalStandard	Thickness (mm)	MinimumBendRadius (mm)	DefaultBendRadius (mm)	KFactor	ClearanceHoleStd	FastenerHoleStd	IndexHoleStd	ManufacturingHoleStd
2	AG 3412	2	0	4	0.36	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls
3	ST 5123	3	1	5	0.27	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls	HoleForAero.xls
4									
5									

Hole Standard

Whenever a hole is created, a design table will associate its radius with a standard name.

	A	B
1	StandardName	Diameter (in)
2	M1	0.39
3	M2	0.65
4	M3	0.89
5	M4	0.25
6	M5	0.56
7		

3. Here is an example for the use of a stamp standard file:

Main Sheet Metal Parameters Design Table

	A	B	C	D	E	F	G	H	I
1	SheetMetalStandard	SurfaceStampStd	CurveStampStd	CircularStampStd	BeadStd	BridgeStd	FlangedCutoutStd	ExtrudedHoleStd	StiffeningRibStd
2	AG 3412	SurfaceStampAG3412.xls	CurveStampAG3412.xls	CircularStampAG3412.xls	BeadAG3412.xls	BridgeAG3412.xls	FlangedCutoutAG3412.xls	ExtrudedHoleAG3412.xls	StiffeningRibAG3412.xls
3	ST 5123	SurfaceStamp5123.xls	CurveStampST5123.xls	CircularStampST5123.xls	BeadST5123.xls	BridgeST5123.xls	FlangedCutoutST5123.xls	ExtrudedHoleST5123.xls	StiffeningRibST5123.xls

Whenever a stamp is created, a design table will associate its dimension with a standard name.

- **Surface Stamp**

	A	B	C	D	E
1	StandardName	Height (mm)	Angle (deg)	Radius1 (mm)	Radius2 (mm)
2	S1	6	80	2	2
3	S2	8	75	1	1
4					

- **Curve Stamp**

	A	B	C	D	E	F
1	StandardName	Height (mm)	Length (mm)	Angle (deg)	Radius2 (mm)	Radius1 (mm)
2	C1	4	6	75	1	1
3	C2	5	7	80	1	1
4						

- **Circular Stamp**

	A	B	C	D	E	F
1	StandardName	Diameter (mm)	Height (mm)	Angle (deg)	Radius1 (mm)	Radius2 (mm)
2	C1	10	6	80	2	2
3	C2	20	5	85	1	1
4						

- **Bead**

	A	B	C	D	E
1	StandardName	SectionRadius(mm)	EndRadius(mm)	Height(mm)	Radius1 (mm)
2	Bead04	4	6	4	2
3	Bead09	9	10	5	3
4					

- **Flanged Cutout**

	A	B	C	D
1	StandardName	Height (mm)	Angle (deg)	Radius1 (mm)
2	F1	6	80	2
3	F2	8	75	1
4				

- **Flanged Hole**

	A	B	C	D
1	StandardName	Diameter (mm)	Height (mm)	Angle (deg)
2	D20	20	6	90
3	D15	15	6	70
4				

- **Stiffening Rib**

	A	B	C	D	E
1	StandardName	Angle (deg)	Radius2 (mm)	Length (mm)	Radius1 (mm)
2	S1	80	2	30	2
3	S2	75	1	35	2
4					



Up Next

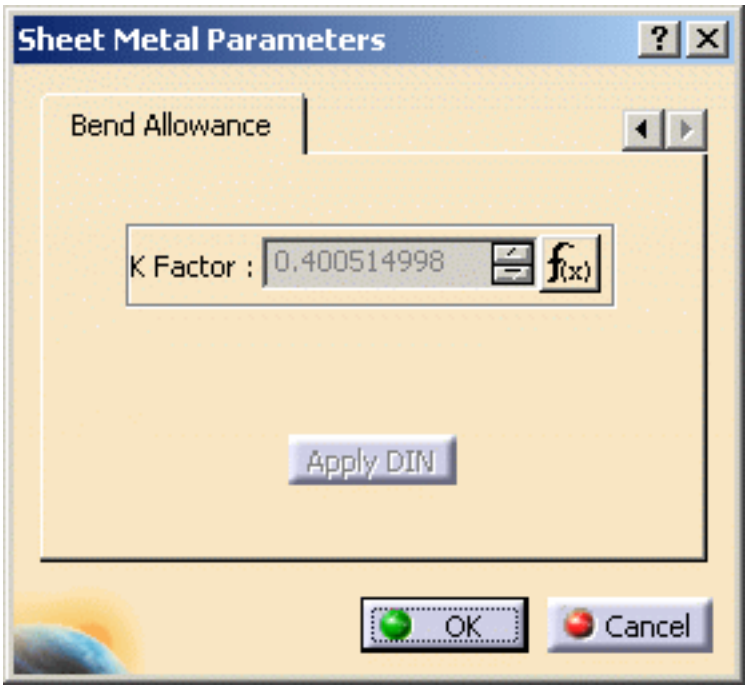
Computing the Bend Allowance

This section explains the calculations related to folding/unfolding operations.

1. Click the **SheetMetal Parameters** icon .

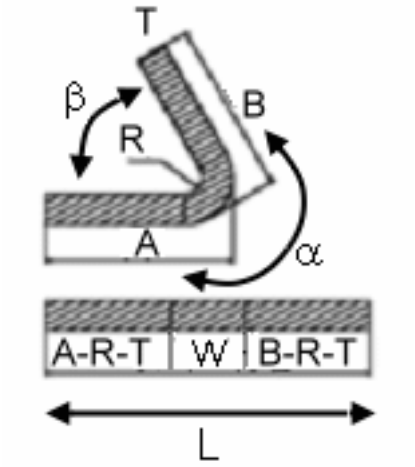
   The Sheet Metal Parameters dialog box is displayed.

   The fourth tab concerns the bend allowance.

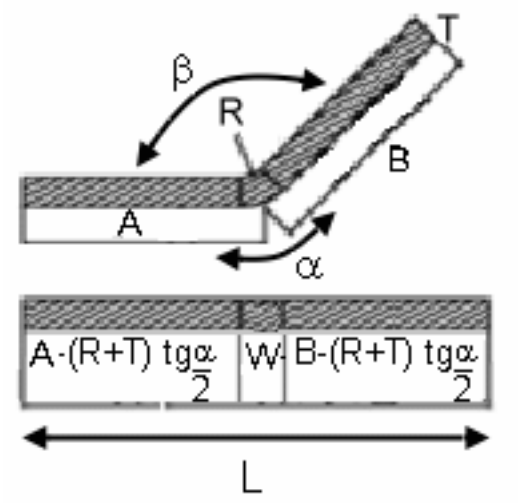


Bend Allowance

The bend allowance corresponds to the unfolded bend width.



bend < 90°

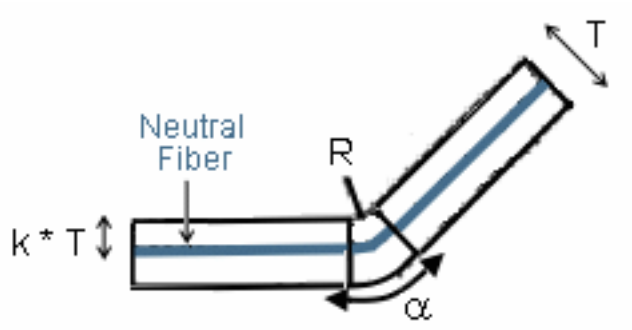


bend > 90°

L is the total unfolded length
A and **B** the dimensioning lengths as defined on the above figure. They are similar to the DIN definition.

- **K Factor**

Physically, the neutral fiber represents the limit between the material compressed area inside the bend and the extended area outside the bend. Ideally, it is represented by an arc located inside the thickness and centered on the bend axis.
The K factor defines the neutral fiber position:



$$W = \alpha * (R + k * T)$$

where:

W is the bend allowance
R the inner bend radius
T the sheet metal thickness
α the inner bend angle in radians.

If β is the opening bend angle in degrees:

$$\alpha = \pi * (180 - \beta) / 180$$

When you define the sheet metal parameters, a literal feature defines the default K Factor and a formula is applied to implement the **DIN** standard. This standard is defined for thin steel parts. Therefore the K Factor value ranges between 0 and 0.5.
The DIN definition for the K factor slightly differs.

$$W = \alpha * (R + k' * T/2)$$

Therefore **k' = 2 * k** and ranges from 0 to 1.

This formula can be deactivated or modified by right-clicking in the K factor field and choosing an option from the contextual menu. It can be re-activated by clicking the Apply DIN button. Moreover, the limit values can also be modified.

When a bend is created, its own K Factor literal is created.
Two cases may then occur:

a. If the Sheet Metal K Factor has an activated formula using the default bend radius as input parameter, the same formula is activated on the bend K Factor replacing the default bend radius by the local bend radius as input.
b. In all other cases, a formula "equal to the Sheet Metal K Factor" is activated on the local bend K Factor.
This formula can also be deactivated or modified.

Bend Deduction

When the bend is unfolded, the sheet metal deformation is thus represented by the bend deduction **V,** defined by the formula:

L = A + B + V

(refer to the previous definitions).

Therefore the bend deduction is related to the K factor using the following formula:

V = α * (R + k * T) - 2 * (R + T) * tan (min(π/2,α) / 2)

This formula is used by default. However, it is possible to define bend tables on the sheet metal parameters. These tables define samples: thickness, bend radius, open angle, and bend deduction. In this case, the bend deduction is located in the appropriate bend table, matching thickness, bend radius, and open angle. If no accurate open angle is found, an interpolation will be performed.

When updating the bend, the bend deduction is first computed using the previously defined rules. Then the bend allowance is deduced using the following formula:

W = V + 2 * (R + T) * tan (min(π/2,α) / 2)



When the bend deduction is read in the bend table, the K factor is not used.



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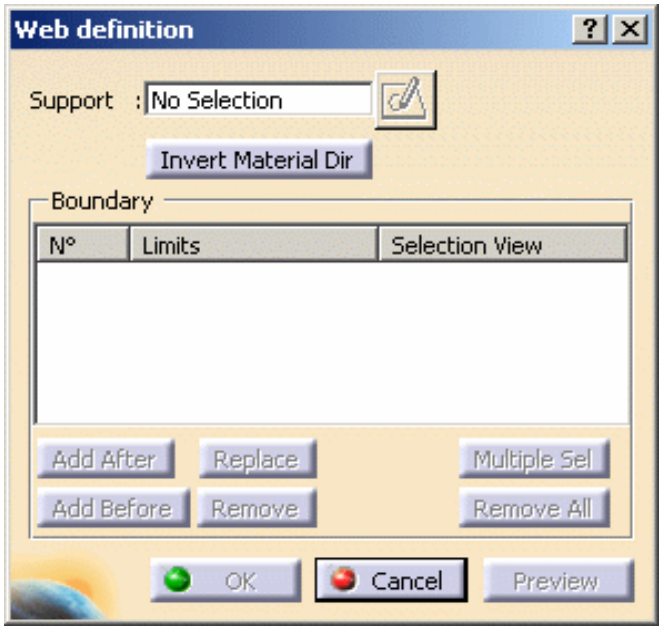
Creating a Web

This section explains how to create a web.

Open the Web1.CATPart document.

1. Click the **Web** icon .

   The Web definition dialog box is displayed.



2. In the **Support** field, select the support geometry in the specification tree. It can either be:

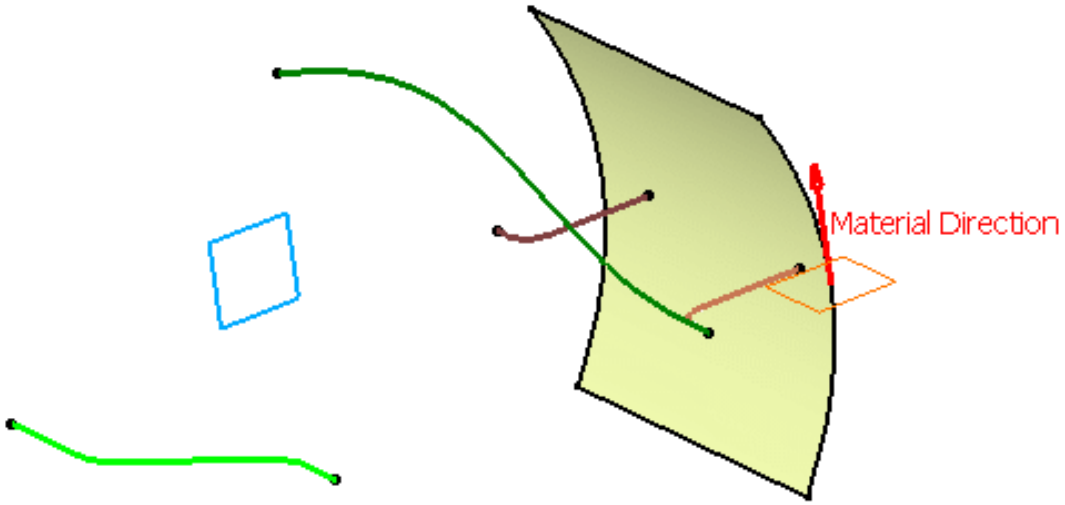
- a plane (example from the *Web from open geometry* open body)

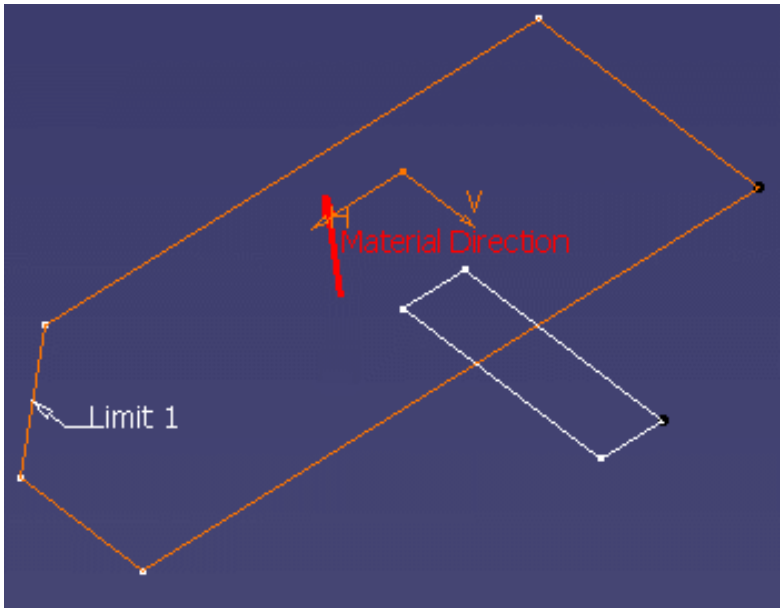
  The Material Direction is displayed, perpendicular to the geometrical support. You can reverse the direction by clicking the arrow.

- a closed sketch (example from the *Web from closed sketches* open body).

  In our example, there are two closed sketches: the web will be calculated on the smallest part of the second sketch.

  The Material Direction is displayed, perpendicular to the geometrical support. You can reverse the direction by clicking the arrow.





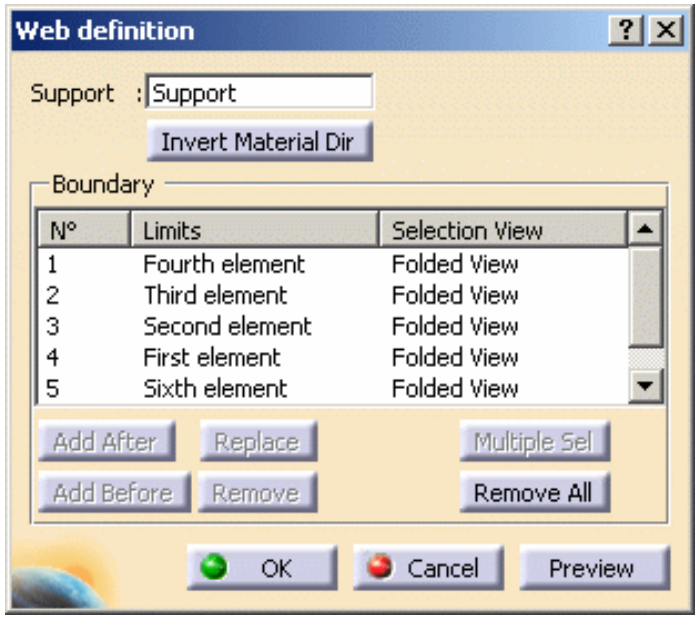
You can click the **Invert Material Dir** button to reverse the material direction of the web.

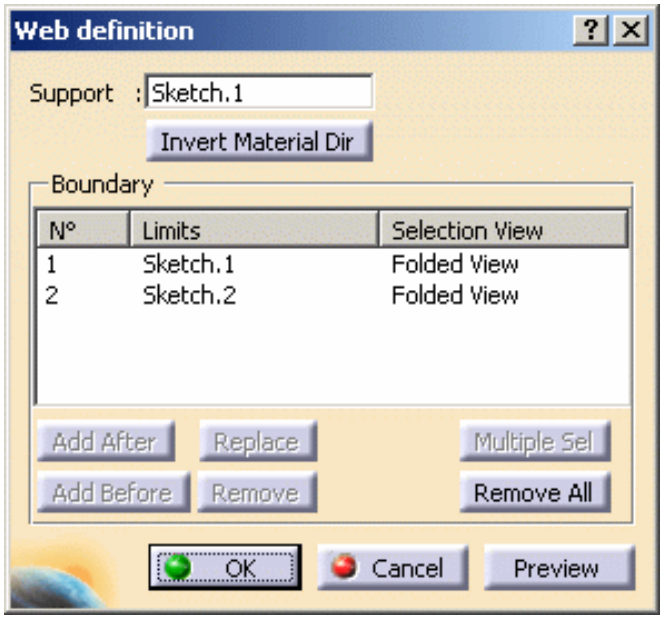
3. In the Boundary field, in the case of an open geometry, select the elements that limit the support geometry. It can either be:

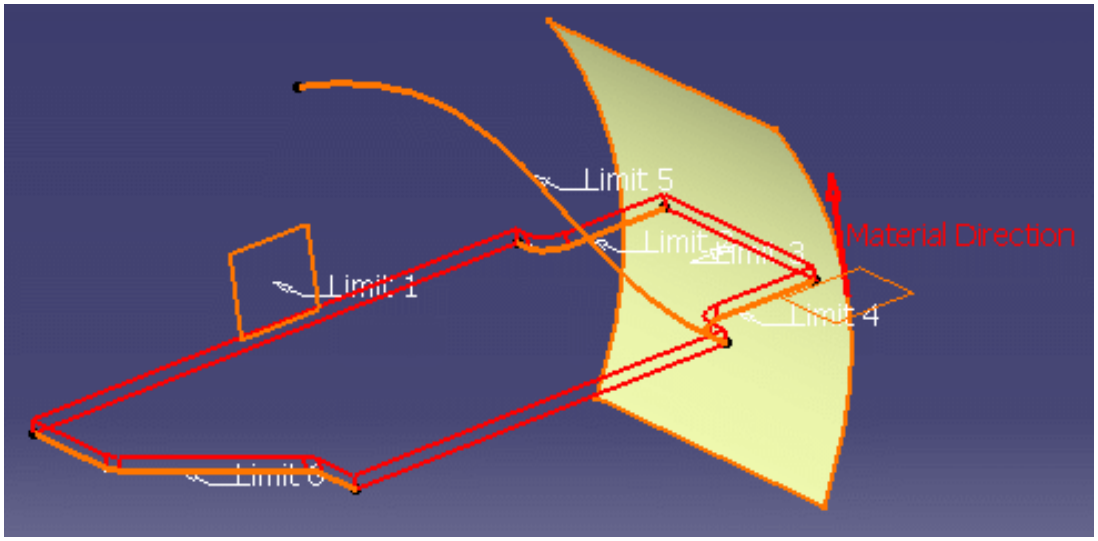
- a list of elements (curves, surfaces, or planes)
- one or more sketches

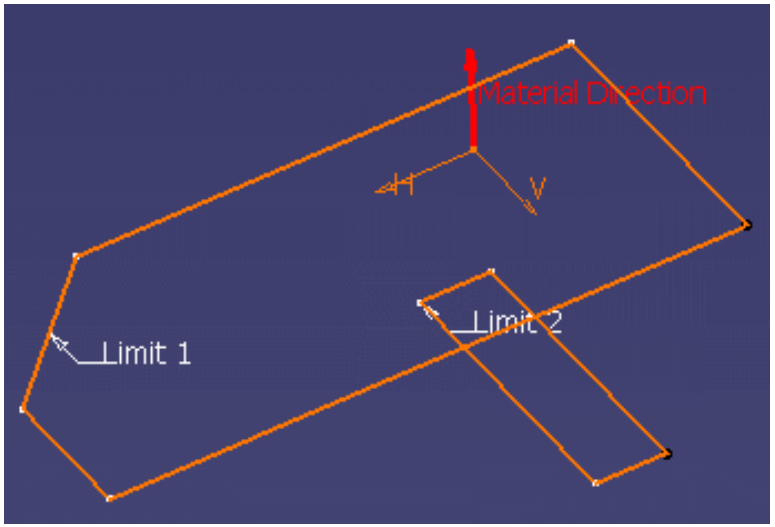
The elements must be selected consecutively.

They are displayed in the Boundary frame, in the order you chose them, as well as in the 3D geometry.









When a closed profile can be built, a light preview of the web is available. Otherwise, click **Preview**.

You can modify the selection by selecting an existing limit and using the following buttons to:

- add a limit after the selected limit (**Add After**)
- add a limit before the selected limit (**Add Before**)
- replace a limit (**Replace**)
- remove a limit (**Remove**)
- remove all limits (**Remove All**)
- select several limits to modify the existing limit (**Multiple Sel**). This option is available once you have selected **Add After**: the Limits to Add dialog box appears to let you select the limits.

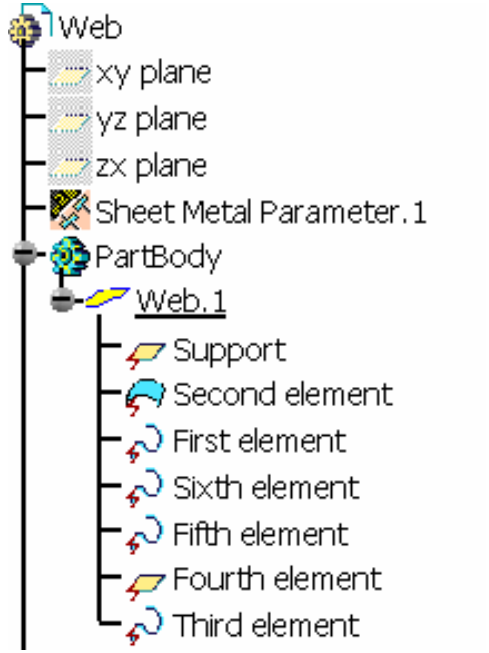
Once you have modified the selection, a light preview is available. You can click the Preview button to display the result of the web.

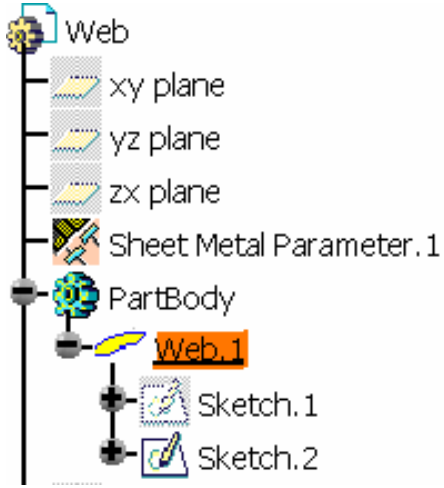
When the contour is defined by a list of geometrical elements, the following operations are performed:

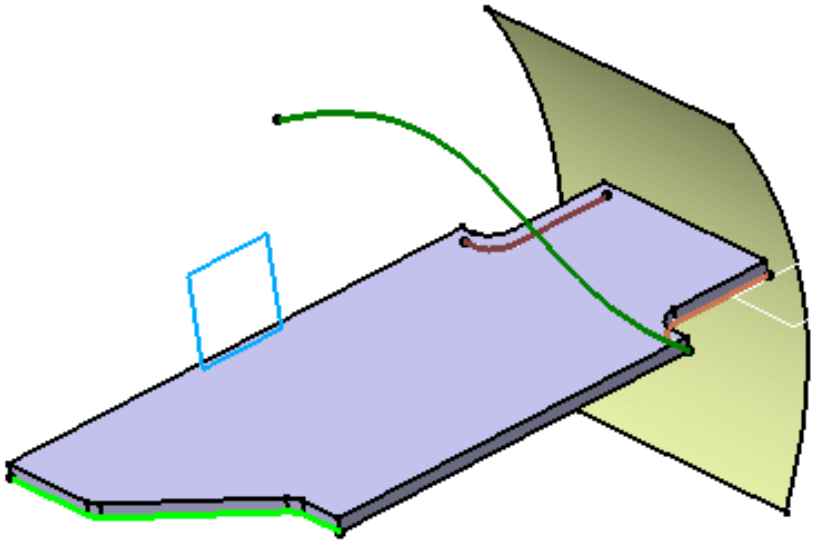
- the curves are projected on the web geometrical support
- the surfaces are intersected with the web geometrical support

4. Click OK.

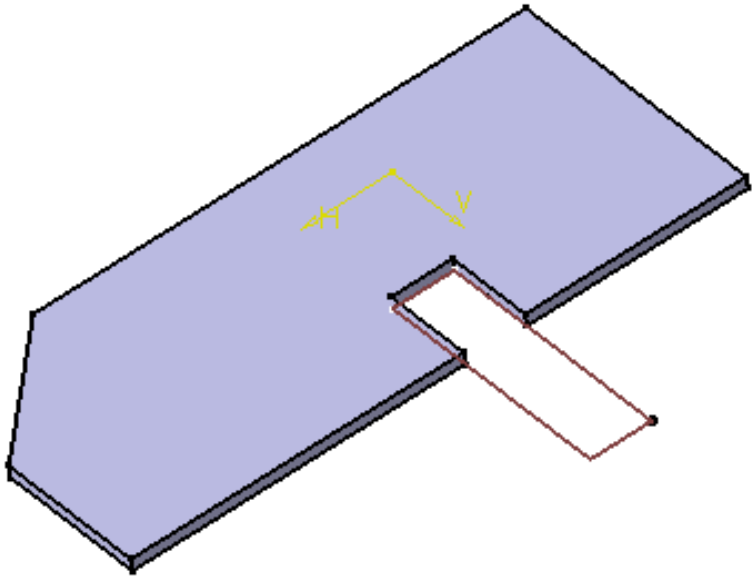
The web (identified as Web.xxx) is created and the specification tree is updated accordingly.







Features are aggregated under the web (identified as Web.xxx) so that they can be selected for a later use.



The sketches are aggregated under the web (identified as Web.xxx). Sketch.1 is displayed in No Show mode as it was only used to create the web.

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Creating Swept Walls

This section explains and illustrates how to create and use various kinds of swept walls, i.e. walls based on a given contour that is swept along a spine.


Create a flange: select a spine, and set the radius, length, and angle values.


Create a hem: select a spine, and set the radius, and length values.


Create a tear drop: select a spine, and set the radius, and length values.

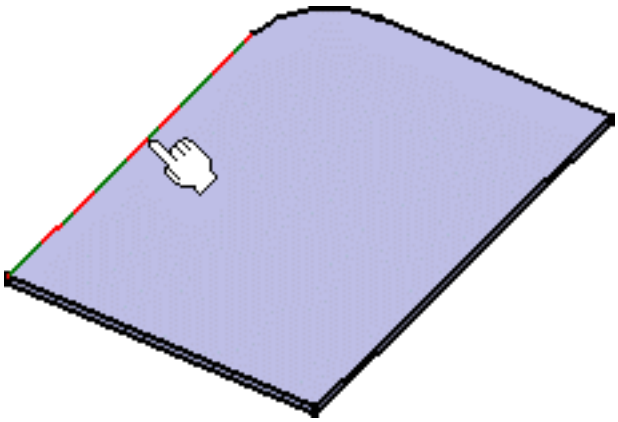

Create a swept flange: select a spine, and a user-defined profile

Redefine swept walls limits: choose the **Relimited** type, and select a point lying on the spine or a plane normal to the spine and intersecting it as limits

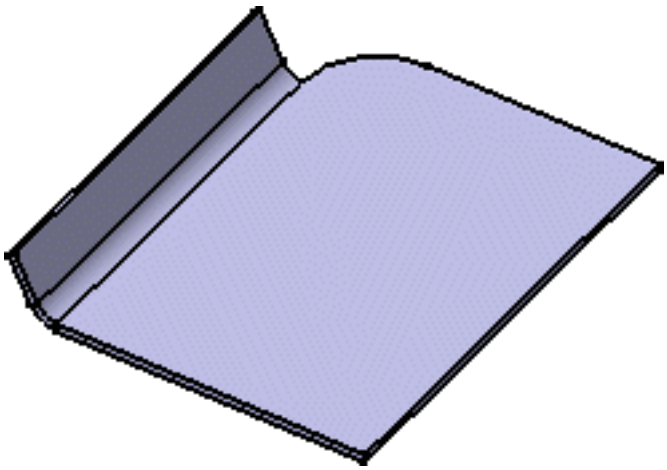
Selecting the Spine

Whatever the type of the swept wall you wish to create, you first need to select one or more contiguous edges to make up the spine along which the contour, either pre- or user-defined, is to be swept. You can:

- manually select one, or more, edge(s)

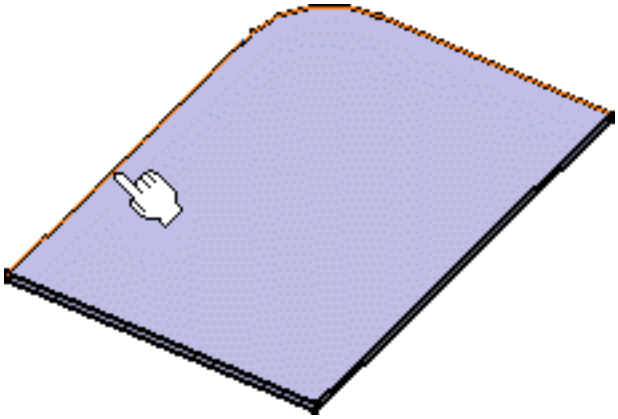


Selection without propagation

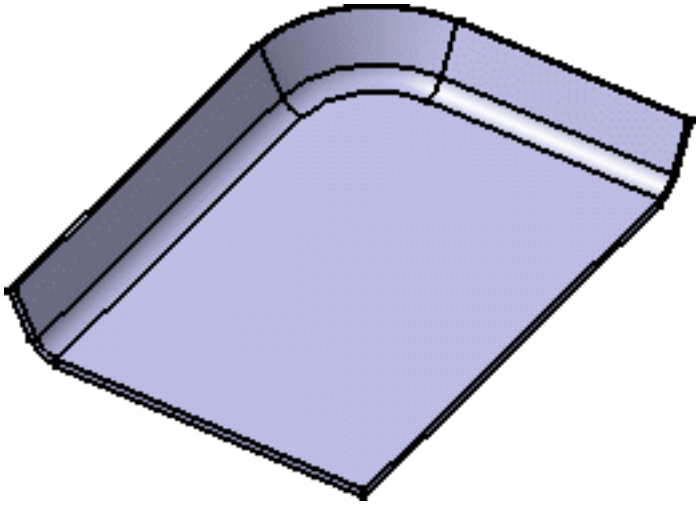


Resulting flange without propagation

- select one edge and click the **Tangency Propagation** button: all contiguous and tangent edges are selected. In this case, would you need to remove one edge, you need to manually select it. Remember that only extremity edges can be removed without breaking the continuity between edges.



Selection with propagation



Resulting flange with propagation

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Unfolding

Unfolded Aerospace Sheet Metal parts can be displayed in two ways:

Folded/Unfolded View Access

Concurrent Access



Each Aerospace Sheet Metal feature is created in a given view: folded, or unfolded. Editing a feature must be done in its definition view. If not, a message is automatically issued, prompting you to change views, before editing the feature.

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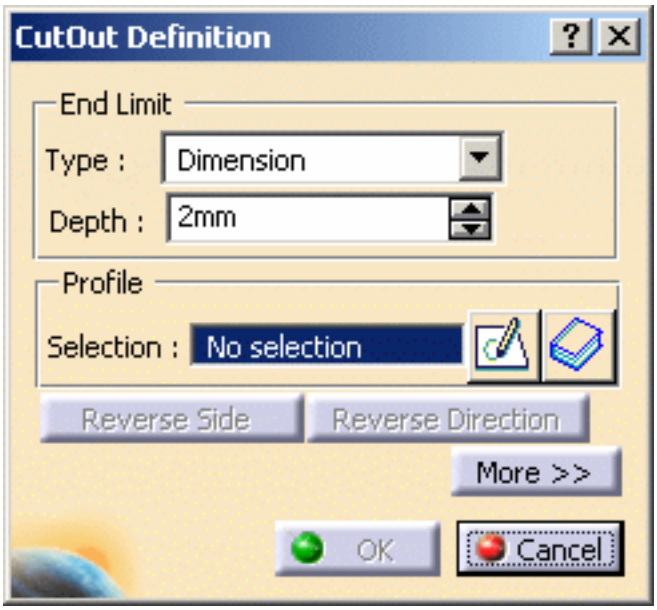
Creating a Cutout

This task explains how to create a cutout.
Creating a cutout consists in extruding a profile and removing the material resulting from the extrusion.

You can create a cutout defined either by a sketch or an open geometry.

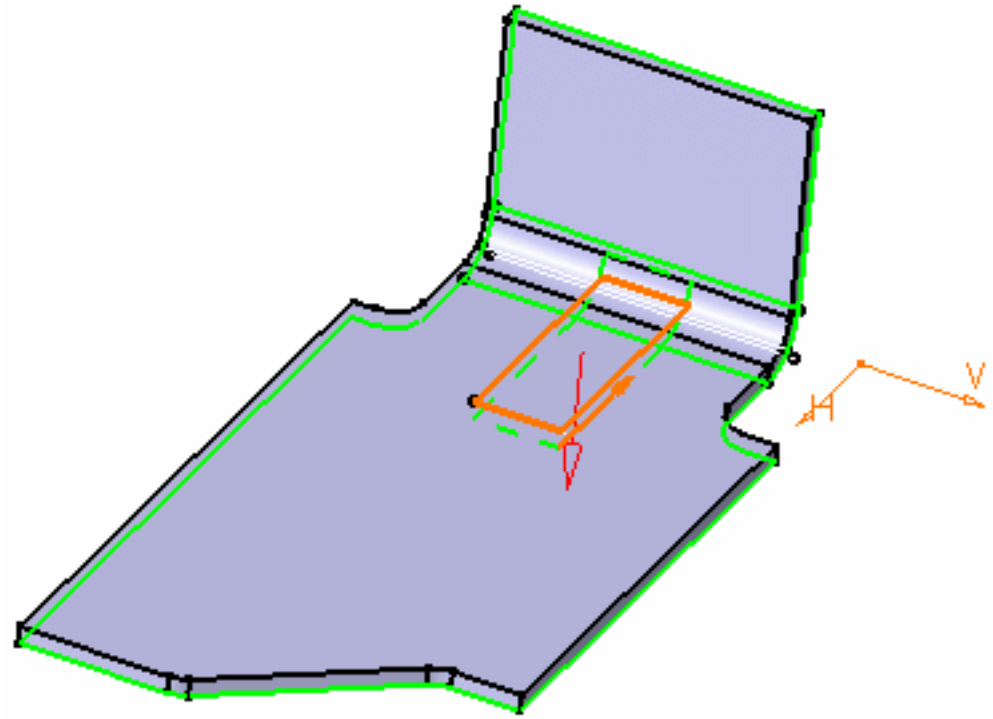
Open the Cutout1.CATPart document.

1. Click the **CutOut** icon .
2. Select a profile.



The CutOut Definition dialog box is displayed and a cutout is previewed with default parameters.

The vectors show the side and the direction of the cutout.



Once the sketch is selected, you can modify it by clicking the Sketcher icon .

3. Select the type.

Several limit types are available:

- **Dimension**: the cutout depth is defined by the value measured along the direction.

The depth corresponds to the web thickness.
Please refer to Editing the Sheet and Tool Parameters.

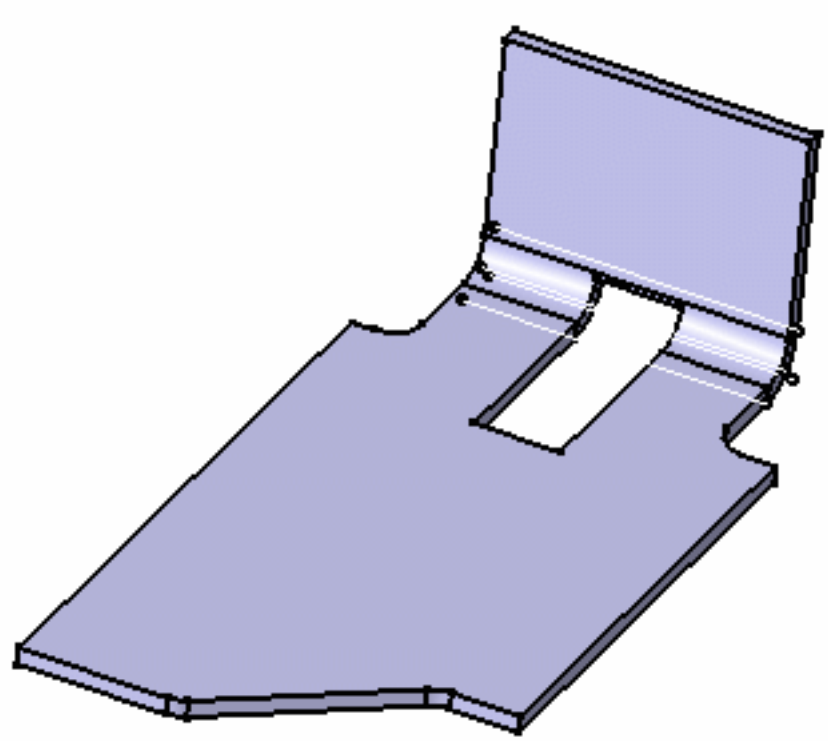
- **Up to next**: the limit is the first face the application detects while extruding the profile. This face must stops the whole extrusion, not only a portion of it, and the hole goes through material.
- **Up to last**: the application will limit the cutout onto the last possible face encountered by the extrusion.

9. Select the profile you created using the sketcher (here Sketch.2).
It can be either a sketch containing one or more shapes, a wire, or a part.

- The Reverse Side option lets you choose between removing the material defined within the profile, which is the application's default behavior, or the material surrounding the profile.
- The Reverse Direction option allows you to invert the direction of the extrusion pointed by the arrow.

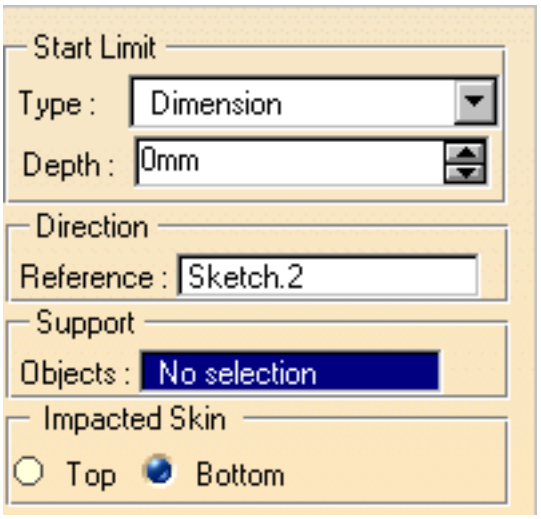
10. Click OK.

The cutout (identified as Cutout.xxx) is created and the specification tree is updated accordingly.



11. Click **More>>** to display the maximum information.

    The Direction is already selected (Sketch.2). If not, it must be perpendicular to the web.



12. Select the **Support** (here we chose the web).

Once the **Reference Direction** and the **Objects Support** fields are filled in, the selection can be modified but cannot be cleared.

Cutouts can be created directly on the unfolded view of the part.

You can use the Catalog icon  to open the Catalog Browser.

May you want to create a cutout on an overlapping element or a bend with radius=0, either choose the top skin of the element (as shown in the picture above), or unfold the part to create the cutout.

- Refer to *Component Catalog Editor* documentation to have further information on how to use catalogs.
- Refer to the Create a Pocket task in the *Part Design User's Guide* for further details on how to create cutouts.



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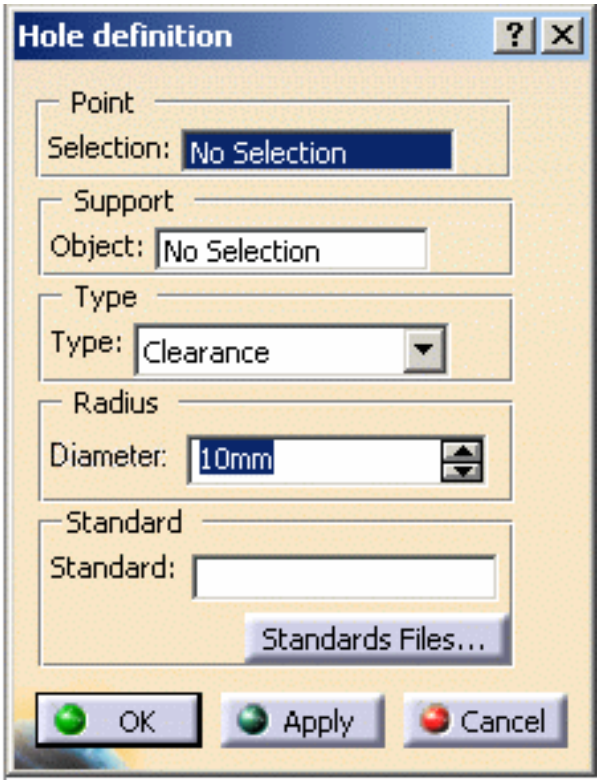
Creating a Hole

This task shows you how to create a hole, that consists in removing material from a body.

Open the Hole1.CATPart document.

1. Click the **Hole** icon .

   The Hole definition dialog box opens.



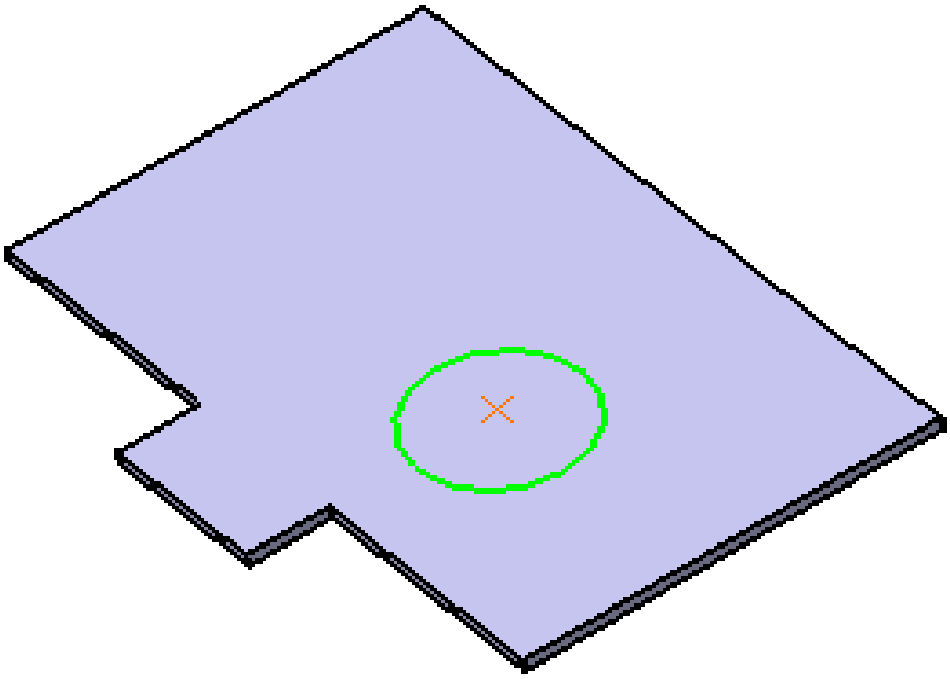
2. Select the **Point** that will be the center of the hole.
   It can be either a sketch containing one or more points, or a point, or several points.

- The point can be selected anywhere in the geometry, not necessarily on a surface. In that case, an orthogonal projection will be performed.
- You can also directly click the surface: a point will be created under the pointer.
- To deselect a point, click it in the specification tree.

3. Select the **Support** object where the hole will be positioned.

- The support can be different from the support where the point lies. In that case, an orthogonal projection will be performed.

The hole is previewed with default parameters.



4. Select hole type:

   - Clearance: defined with a center (point) and a radius
   - Index: used to measure and validate parts
   - Manufacturing: used for manufacturing (for example to fasten a part on an equiment
   - Fastener: used as a rivet

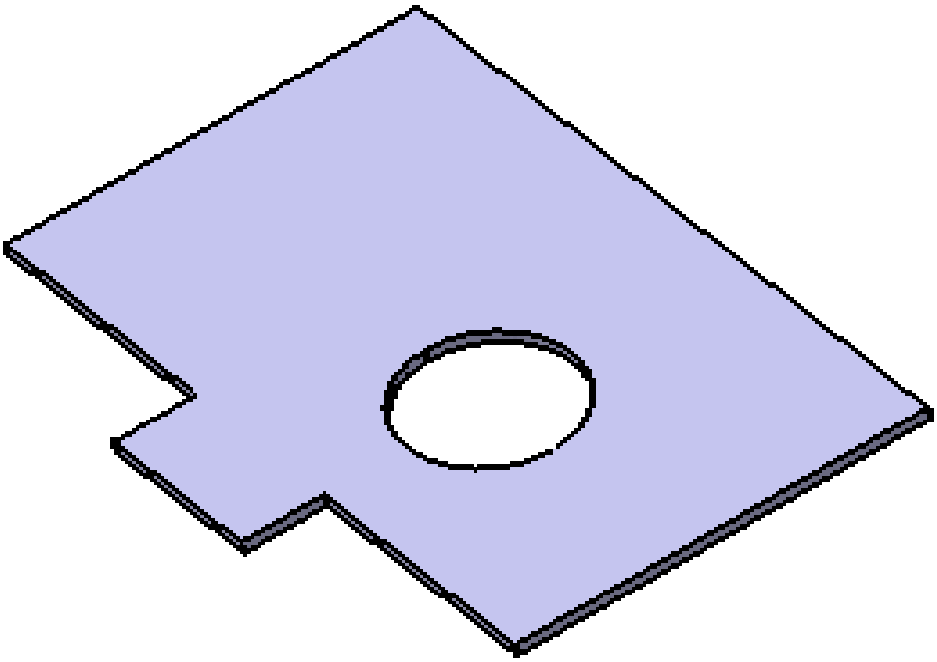
Hole types do not affect the hole geometry.

5. Define the value for the diameter of the hole in the **Diameter** field.

If you change the **Radius** value using the spinners, the preview of the hole automatically updates. However, if you enter a value directly in the field, you need to click the **Apply** button to update the preview.

6. Click OK to validate.

The hole (identified as Hole.xxx) is created and the specification tree is updated accordingly.



- To have further information on **Standard Files...**, please refer to the Customizing section.

Holes can be created on the flattened part and on the bend in case of a flange.

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Creating Stamping Features

This section explains and illustrates how to create and use various kinds of stamps.
The table below lists the information you will find.


Create a flanged hole: select a point on a face, and set the stamping parameters.


Create a bead: select a profile, and set the stamping parameters.


Create a circular stamp: select a point on a face, and set the stamping parameters.


Create a surface stamp: select a sketch, and set the stamping parameters.


Create a flanged cutout: select a profile, and set the stamping parameters.


Create a stiffening rib: select the external surface of the bend, and set the stamping parameters.


Create a curve stamp: select a sketch, and set the stamping parameters.

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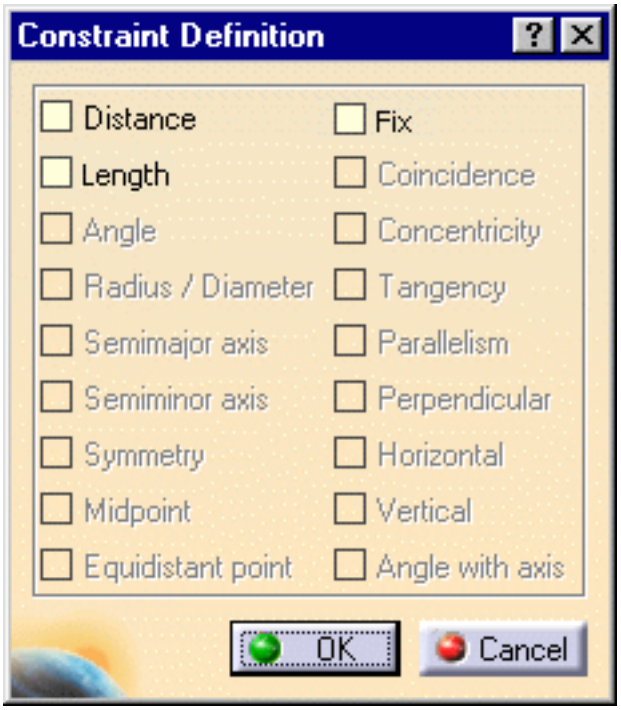
Creating Constraints

This task shows how to set geometric constraints on geometric elements.

Such a constraint forces a limitation. For example, a geometric constraint might require that two lines be parallel.

To set a constraint between elements:

1. Multi-select two or three elements to be constrained.
2. Click the **Constraint defined in dialog box** icon .



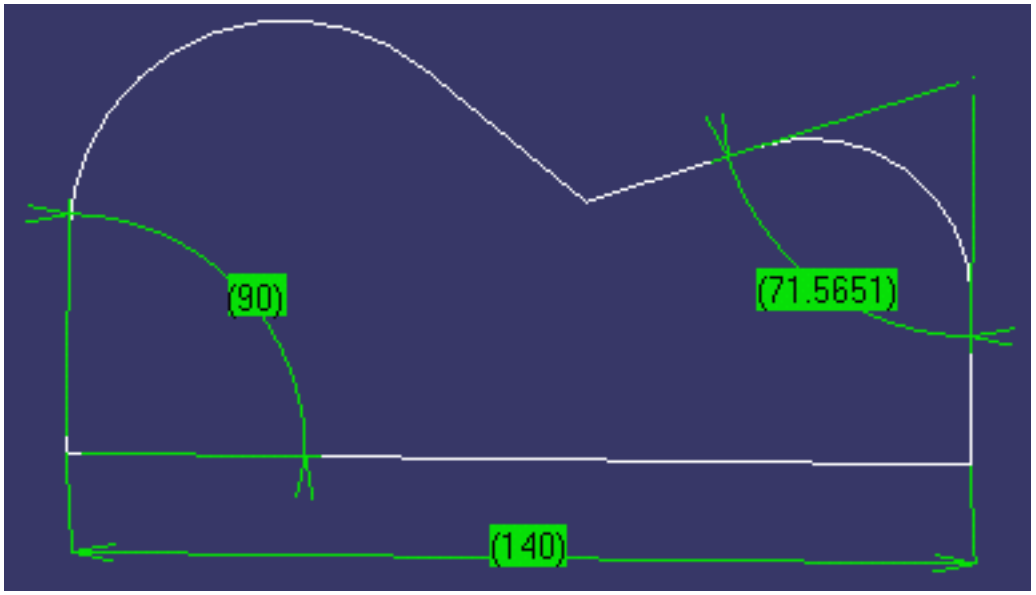
The Constraint Definition dialog box appears indicating the types of constraint you can set between the selected elements.

3. Select one of the available options to specify that the corresponding constraint should be made.
4. Click OK.

The corresponding constraint symbol appears on the geometry.

To set a constraint on a single element:

1. Select the element to be constrained.
2. Click the **Constraint** icon .



The corresponding constraint symbol appears on the geometry.



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Reference Elements

You can create wireframe elements within the Aerospace Sheetmetal Design workbench:

 Create points: click this icon, choose the point creation type, and specify parameters

 Create lines: click this icon, choose the line creation type, and specify parameters

 Create planes: click this icon, choose the plane creation type, and specify parameters

Back Up Next

Displaying Characteristic Curves

This task shows you how to manage characteristic curves.
Characteristic curves are displayed in the folded view of the part, as well as in the flattened view. They can be selected, though not edited, and be used as a support (to create points for example),

A surfacic flange or a stamp with a fillet must be created.

The following curves can be computed:

- **OML**: Outer Mold Line
  Intersection between external surfaces of the feature (before filleting) and the part
- **IML**: Inner Mold Line
  Intersection between internal surfaces of the feature (before filleting) and the part
  Computed only on a hydropressed flange
- **BTL**: Bent Tangent Line
  Limits of the fillet

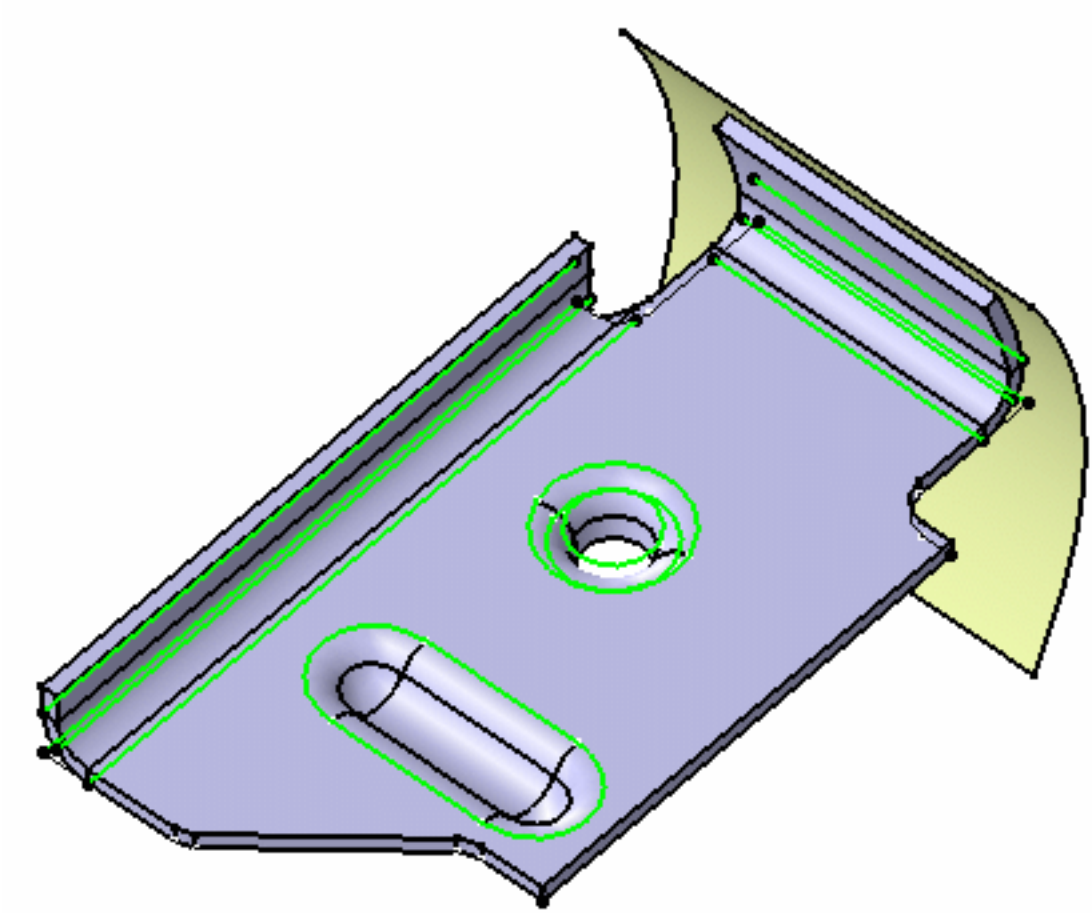
They must be computed on the following features:

- **Surfacic Flange**: the following curves are computed on both folded and flattened view (as defined in the Process tab of the Surfacic Flange Definition dialog box)
  - BTL (two BTL for each limit of the fillet)
  - OML
  - IML
- **Stamps with an internal cutout** (flanged hole, flanged cutout, etc): the following curves are computed on both folded and flattened view
  - BTL on the base feature
  - IML (in no show in the folded view)
  - OML (in no show in the folded view)
- **Stamps without an internal cutout** (bead, curve stamp, etc): the following curve is computed on both folded and flattened view
  - BTL on the base feature

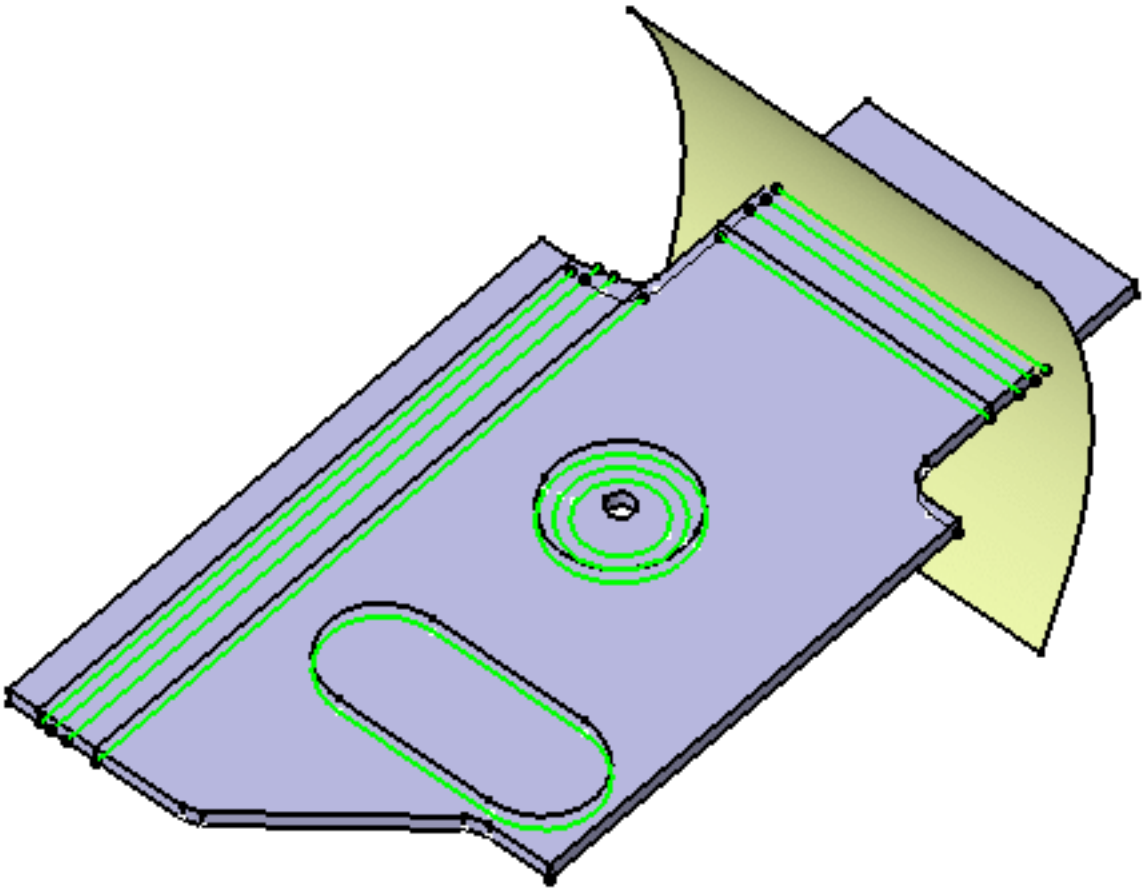
Open the CharacteristicCurves1.CATPart document.

Here is an example with two surfacic flanges, a bead and a flanged hole.
All characteristic curves are put in show in the following images.

Characteristic curves computed on the folded view



Characteristic curves computed on the flattened view



If you want to edit and modify a curve color, select it in the 3D geometry, right-click and choose the **Properties** contextual command. In the Properties dialog box, select the Graphic tab to access the graphic properties of the curve.

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Looking For Aerospace SheetMetal Features

This task shows how to use the Search capabilities on Aerospace SheetMetal Features, in order to detect any specific kind of feature.

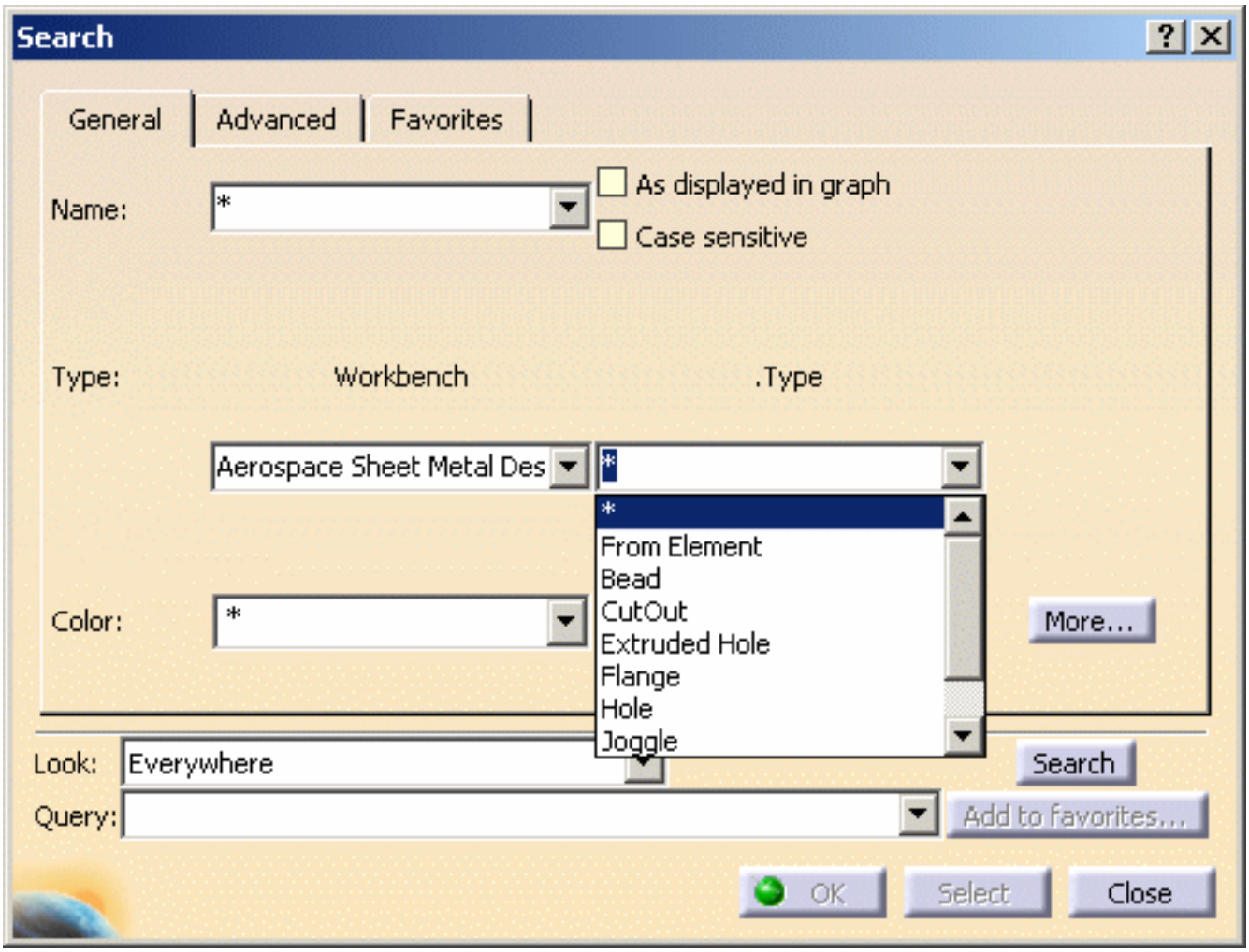
Open the PowerCopyStart.CATPart document.

1. Select the **Edit -> Search** menu item.

   The Search dialog box is displayed.

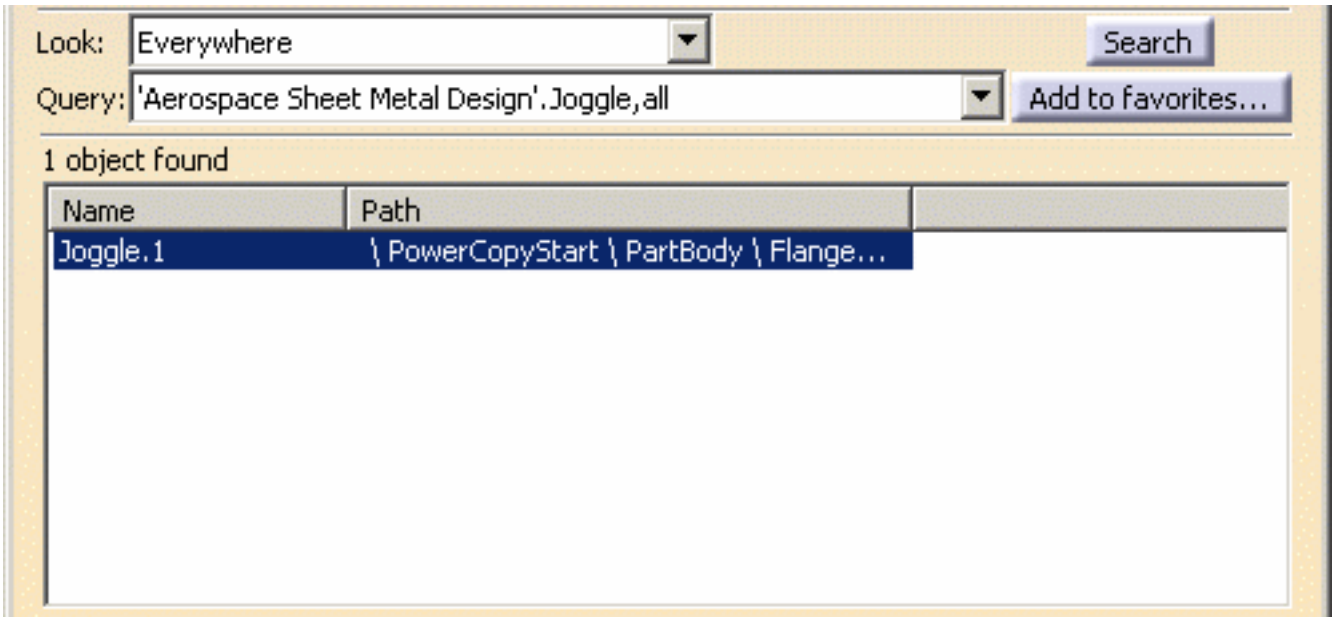
2. From the Type Workbench list choose Sheet Metal.

   You can then display the list of Aerospace Sheet Metal Design features from the **Type** list:



3. Select the type of feature you wish to find within the open .CATPart document. Here we chose *Joggle*.
4. Click **Search**.

   The list of all elements of the selected type is displayed in the Objects found field:



You can select an element from the list, it will be highlighted in the geometry area.

To find out more on the search capabilities, refer to Selecting Using the Search... Command (General Mode) and Selecting Using the Search... Command (Favorites and Advanced Modes) from the *Infrastructure User's Guide*.

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Browsing the SheetMetal Catalog

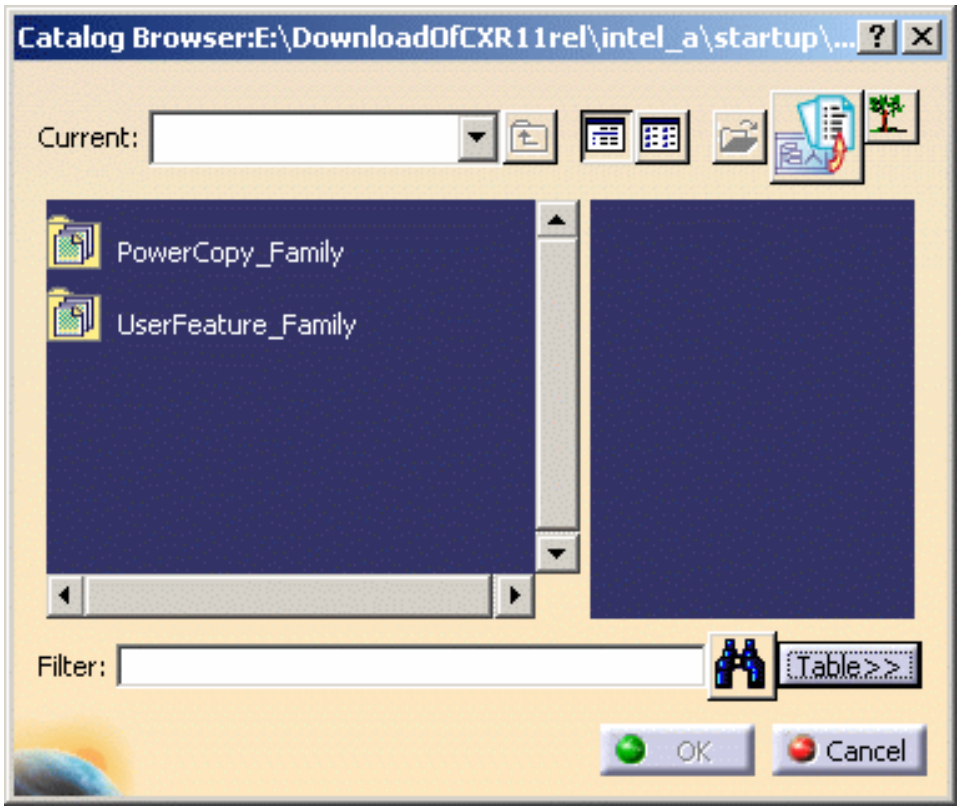
This task explains how to browse the SheetMetal catalog and instantiate its components. The catalog enables to store the available profiles, therefore providing a method to position the profile in the part.
This command is available with the CutOut and the Corner Relief functionalities.

Let's take an example with the CutOut functionality.
For more information on catalogs, please refer to the Using Catalogs chapter in the *CATIA Infrastructure User Guide*.

Open the CutOut1.CATPart document.

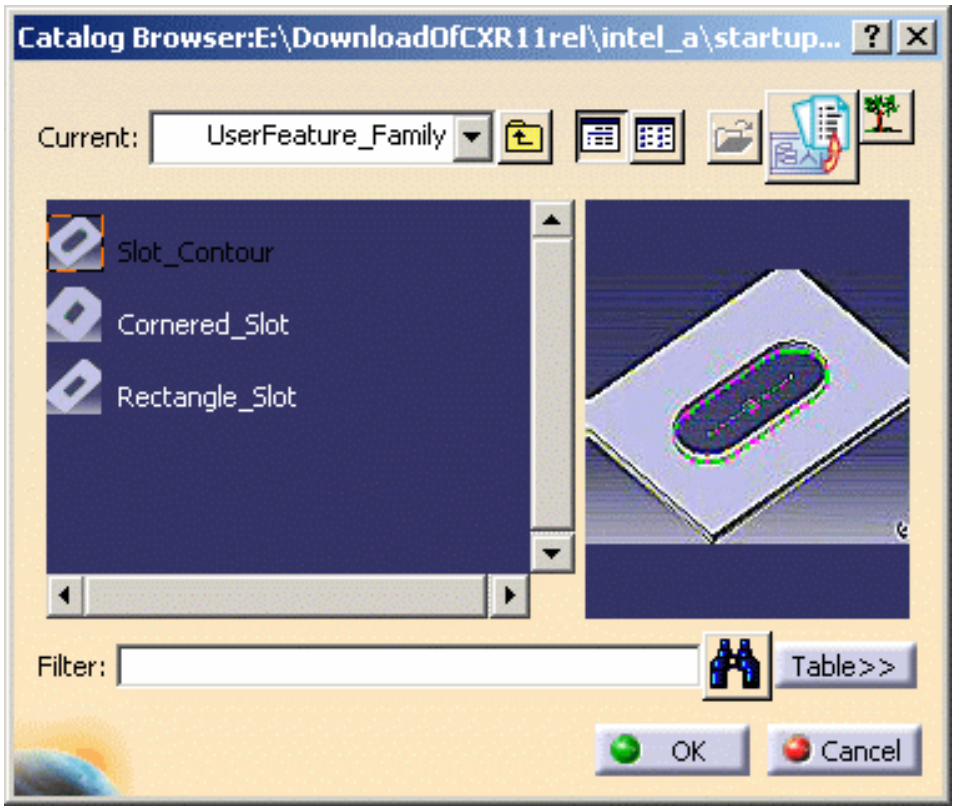
1. Once in the CutOut Definition dialog box, click the **Catalog** icon .

   The Catalog Browser is displayed.



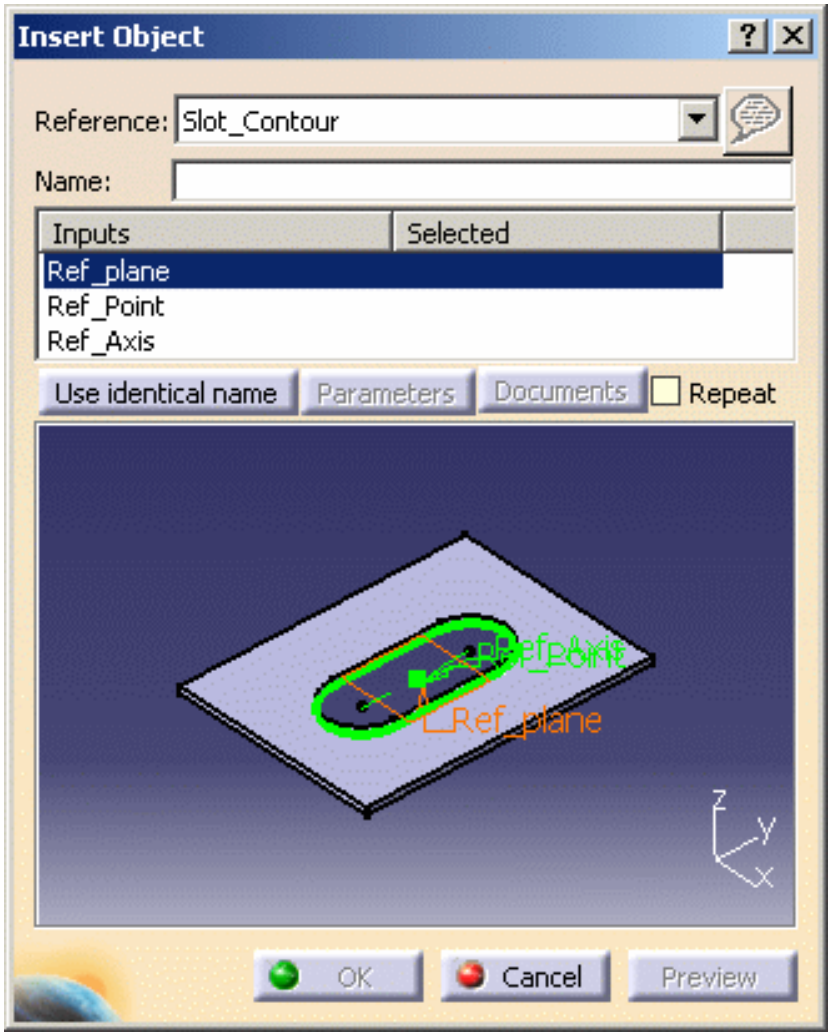
2. Double-click a family from the list to display its components.
   Here we chose the UserFeature_Family.

3. Click a component to see its preview. Here we chose Slot_Contour.



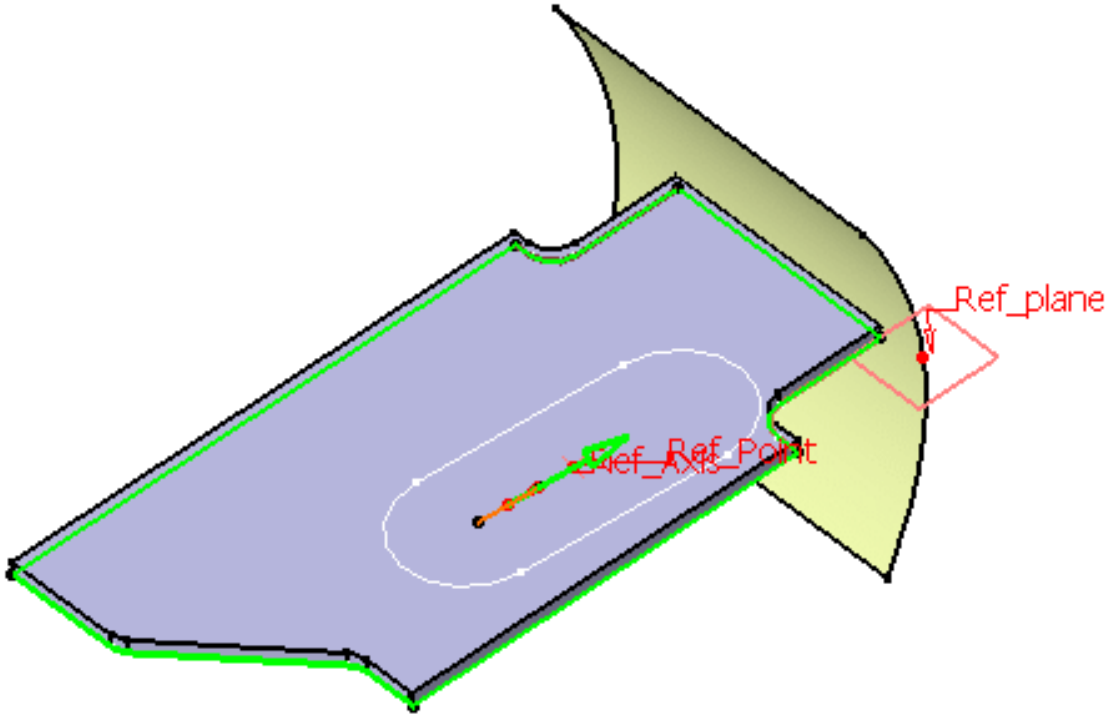
4. Click the **Table>>** button to show/hide the catalog descriptions and keywords. By default, the table is hidden.

5. Instantiate the component by double-clicking it.

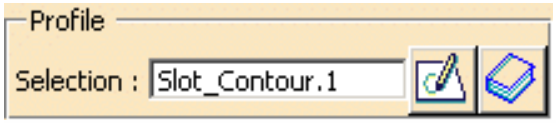


The Insert Object dialog box is displayed.

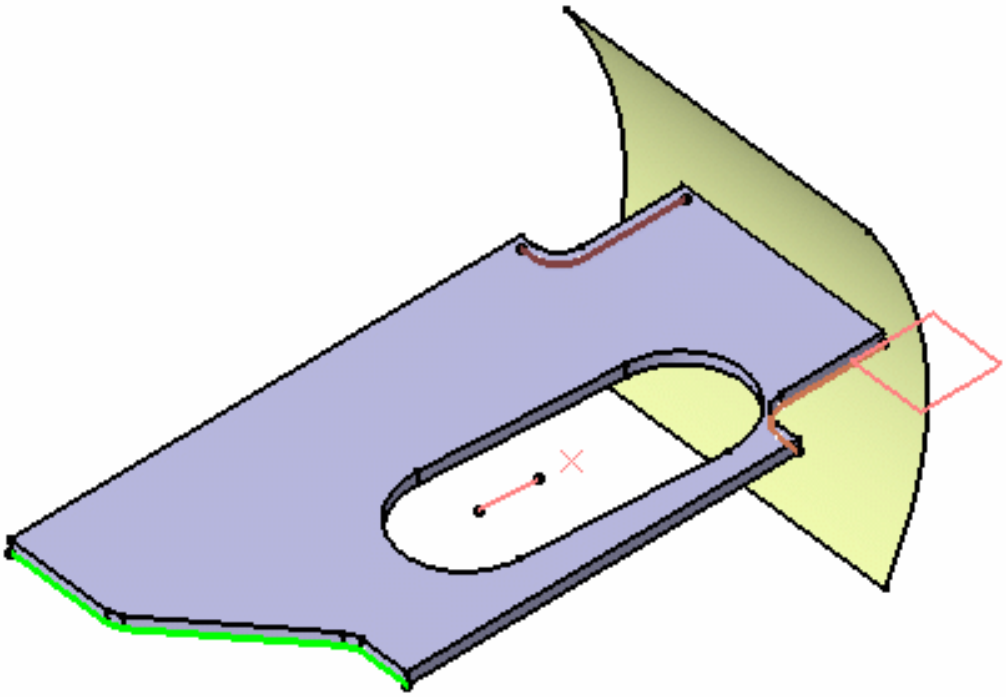
6. Select the required inputs: plane, point, and axis.
7. Click Preview to see the resulting profile in the 3D geometry.
8. Click OK in the Insert dialog box



The selected profile appears in the Selection field.



6. Click OK.



The created element (identified as Cutout.xxx) is added to the specification tree.

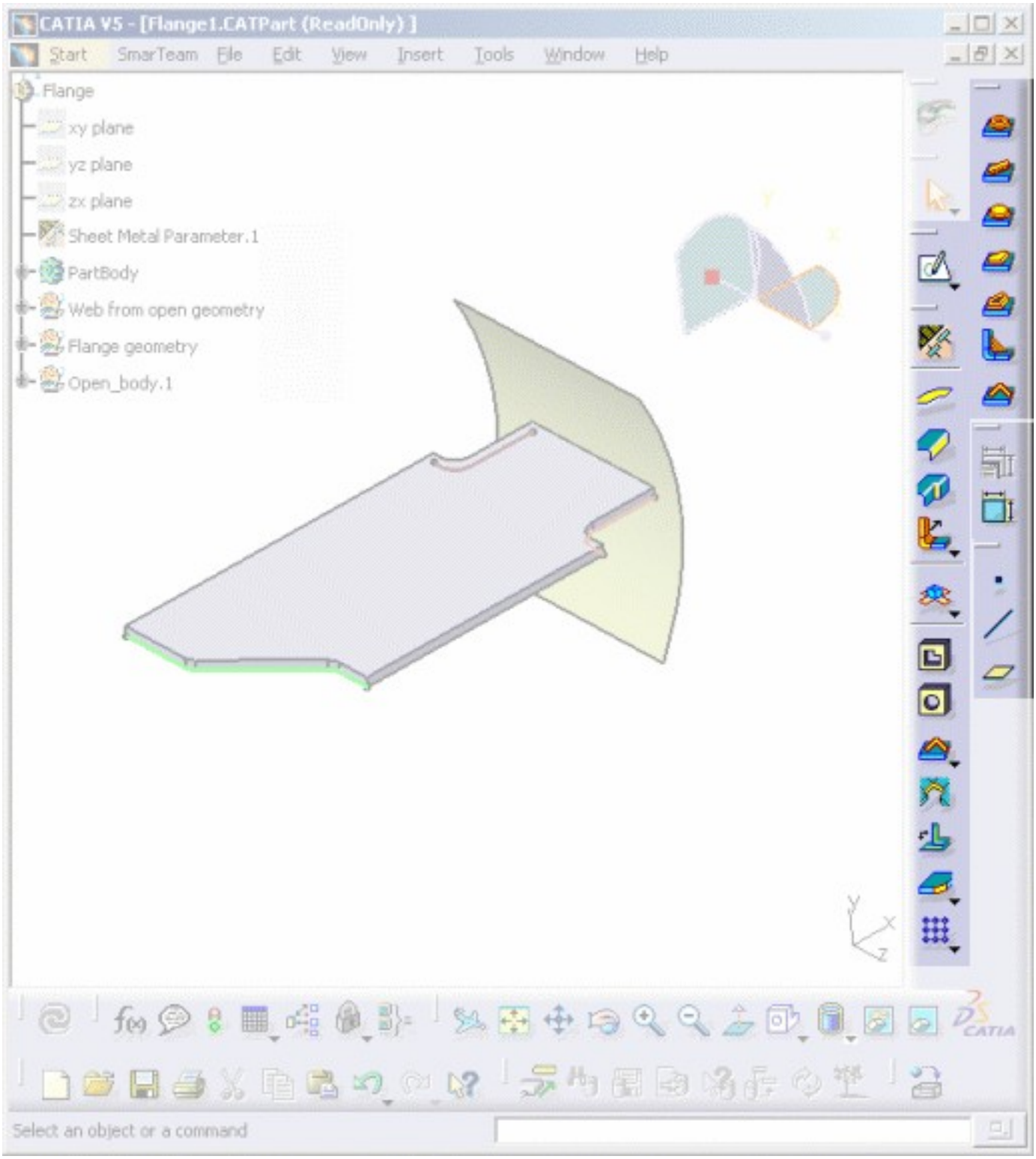
You may need to reverse the direction of the cutout to create it.

Once the profile is instantiated in the default catalog, its path is automatically set in the Standard Profiles Catalog Files field.
See Customizing General Settings.

A new panel now allows you to select alternate document access methods.
See Opening Existing Documents Using the Browse Panel in *CATIA Infrastructure User Guide*.

Workbench Description

The Aerospace Sheet M etal Design application window looks like this:
Click the hotspots to display the related documentation.



Menu Bar
Aerospace SheetMetal Toolbar
Stampings Toolbar
Constraints Toolbar
Reference Elements Toolbar
Specification Tree

Menu Bar

The various menus and menu commands that are specific to Aerospace Sheetmetal Design are described below.

Start	File	Edit	View	Insert	Tools	Windows	Help

Tasks corresponding to general menu commands are described in the *Infrastructure User's Guide*. Refer to the Menu Bar section.

Edit

The Edit menu lets you manipulate selected objects. Refer to the *Infrastructure User's Guide* and *Part Design User's Guide*.

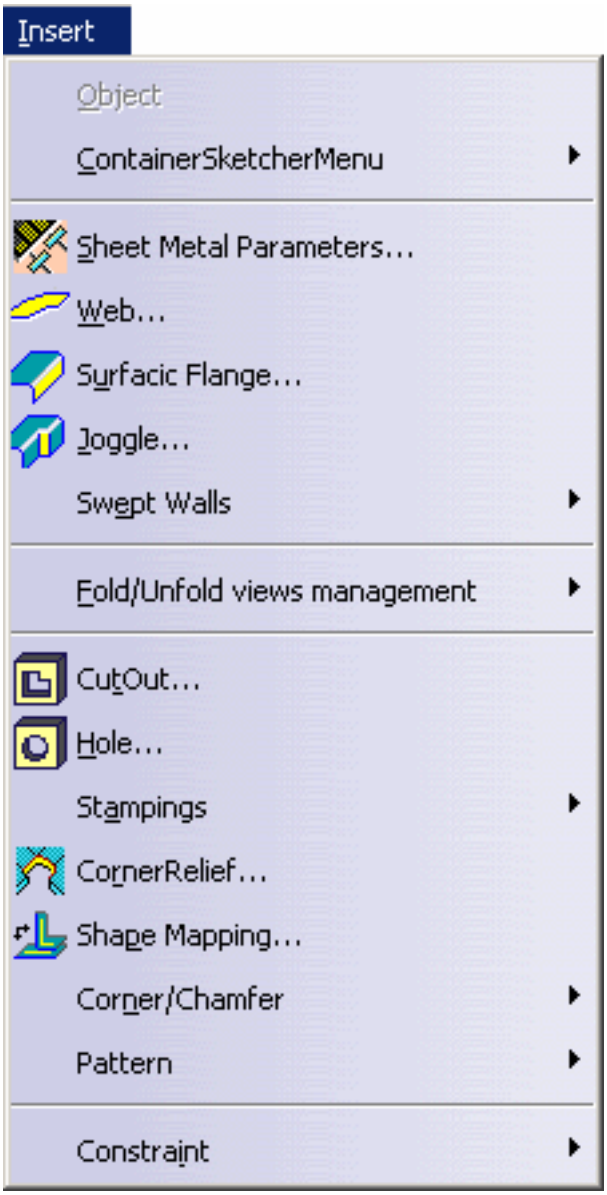
View

The View menu lets you view document contents Refer to the *Infrastructure User's Guide*.

The **Search** capability is available.

Insert

For...	See...
ContainerSketcherMenu	Refer to the *Sketcher User's Guide*.
Sheet Metal Parameters...	Managing the Default Parameters
Web...	Creating a Web
Surfacic Flange...	Creating a Surfacic Flange
Joggle...	Creating a Joggle
Swept Walls	Insert -> Swept Walls
Fold/Unfold views management	Insert -> Unfold



CutOut...	Creating a Cutout
Hole...	Creating a Hole
Stampings	Insert -> Stampings
Corner Relief...	Creating a Local Corner Relief
Shape Mapping	Mapping Elements
Corner/Chamfer	Insert -> Corner/Chamfer
Pattern	Insert -> Pattern
Constraint	Setting Constraints in the *Part Design User's Guide*

Insert -> Swept Walls



For...	See...
Flange	Creating a Flange
Hem	Creating a Hem
Tear Drop	Creating a Tear Drop
User Flange	Creating a Swept Flange

Insert -> Unfold



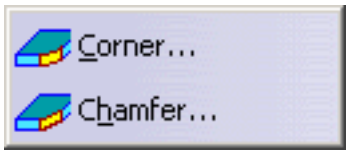
For...	See...
Unfold...	Folded/Unfolded View Access
Multi Viewer...	Concurrent Access

Insert -> Stampings



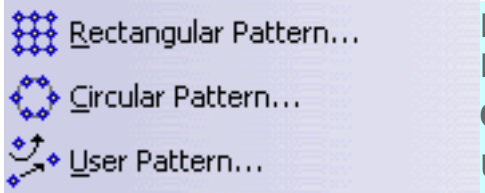
For...	See...
Flanged Hole...	Creating a Flanged Hole
Bead...	Creating a Bead
Circular Stamp...	Creating a Circular Stamp
Surface Stamp...	Creating a Surface Stamp
Flanged CutOut...	Creating a Flanged Cutout
Curve Stamp...	Creating a Curve Stamp
Stiffening Rib...	Creating a Stiffening Rib

Insert -> Corner/Chamfer



For...	See...
Corner...	Creating Corners
Chamfer...	Creating Chamfers

Insert -> Pattern



For...	See...
Rectangular Pattern...	Creating Rectangular Patterns
Circular Pattern...	Creating Circular Patterns
User Pattern...	Creating User-Defined Patterns

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Aerospace SheetMetal Toolbar

The Aerospace Sheet Metal Toolbar contains the following tools:



 See Managing the Default Parameters

 See Creating a Web

 See Creating a Flange

 See Creating a Joggle

 See Creating Swept Walls



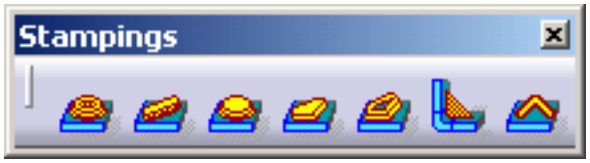
 See Unfolding the Part



 See Creating a Cutout

 See Creating a Hole

 See Creating Stamping Features



 See Creating a Corner Relief

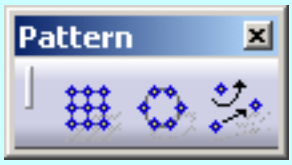
 See Mapping Elements

 See Creating Corners
See Creating Chamfers





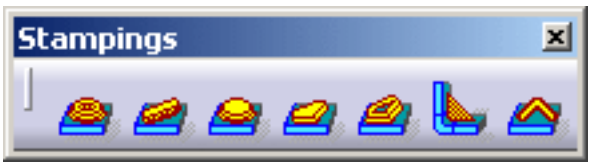
See Creating Patterns



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Stampings Toolbar

The Stampings toolbar contains the following tools:




See Creating a Flanged Hole


See Creating a Bead


See Creating a Circular Stamp


See Creating a Surface Stamp


See Creating a Flanged Cutout


See Creating a Stiffening Rib


See Creating a Curve Stamp

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Constraints Toolbar

The Constraints Toolbar contains the following tools:




See Setting Constraints from the *Part Design User's Guide*



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Reference Elements Toolbar

The Reference Elements Toolbar contains the following tools:




See Creating Points


See Creating Lines


See Creating Planes

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Specification Tree

Within the Aerospace Sheetmetal Design workbench, you can generate a number of features that are identified in the specification tree by the following icons.

Further information on general symbols in the specification tree are available in Symbols Used in the Specification Tree.

 Sketch

 Sheet Metal Parameters

 Web

 Flange

 Joggle

 Flange

 Hem

 Tear Drop

 Swept Flange

 Cutout

 Hole

 Flanged Hole

 Bead

 Circular Stamp

 Surface Stamp

 Flanged Cutout

 Corner Relief

 Mapping

 Corner

 Chamfer

 Rectangular Pattern

 Circular Pattern

 User-Defined Pattern

 Point

 Line

 Plane


Curve Stamp


Stiffening Rib

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Customizing For Aerospace Sheet Metal Design

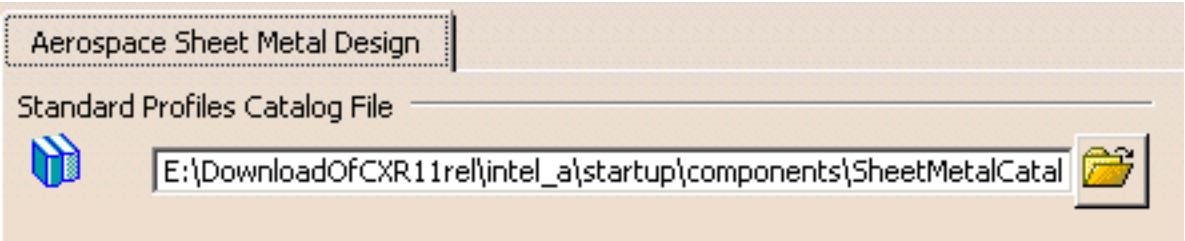
This section describes how to customize different settings specific to the Aerospace Sheet Metal Design workbench. The settings described here deal with permanent setting customization.

Customizing General Settings
Customizing Standard Files To Define Design Tables
Customizing Standards Files To Define Methods for Compensations

Customizing General Settings

This task explains how to customize the Aerospace Sheetmetal Design General Settings.

1. Select the **Tools -> Options** menu item.
   The Options dialog box appears.
2. Click **Mechanical Design category then the Aerospace Sheetmetal Design** subcategory.



The default path is displayed in the Standard Profiles Catalog Files. You can modify it by clicking the **Browse** icon .

If no catalog path has been defined prior to entering the Catalog Browser command, the default catalog is selected and is path automatically added to the Standard Profiles Catalog Files field.

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Customizing Standards Files To Define Design Tables

This section describes how to customize settings.
The task described here deals with permanent setting customizing.

Using Sheet Metal Standards Files

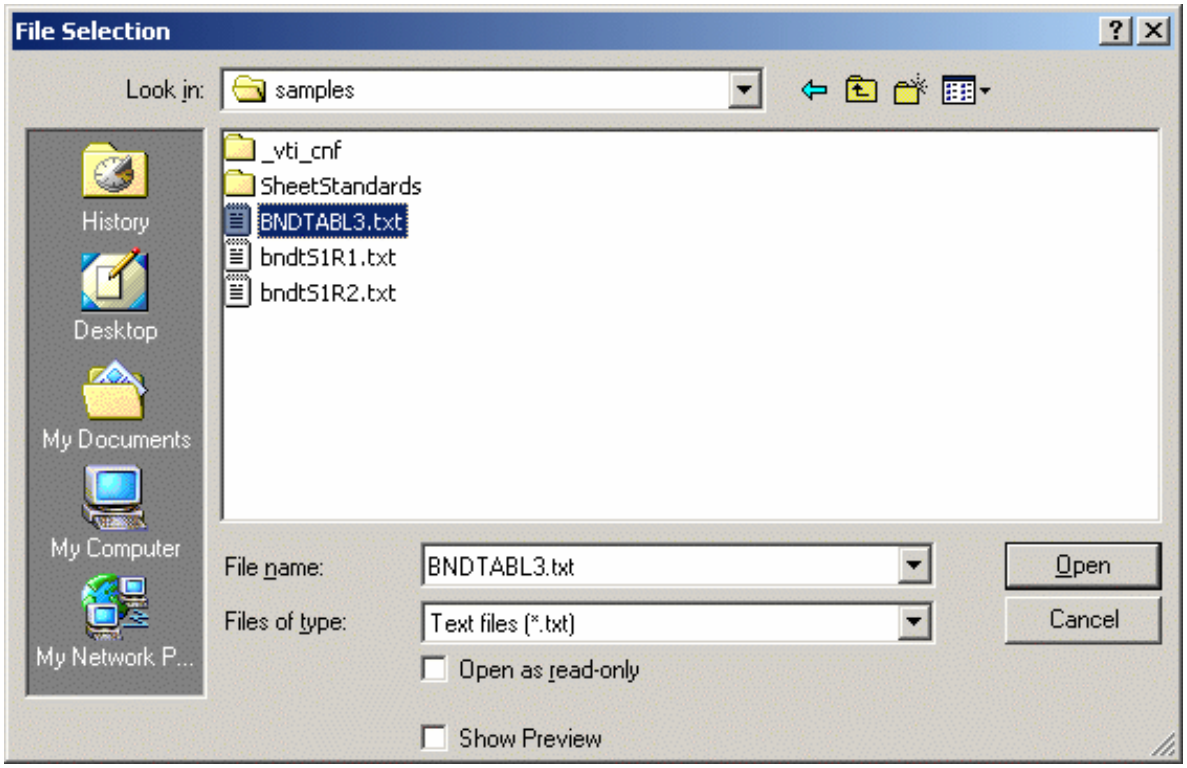
This task explains how to access company standards files in order to access and define design tables.

Open a new document.

1. Click the **Sheet Metal Parameters** icon .
   The Sheet Metal Parameters dialog box opens.

2. Select the **Sheet Standards Files...** button.
   The File Selection window is displayed.

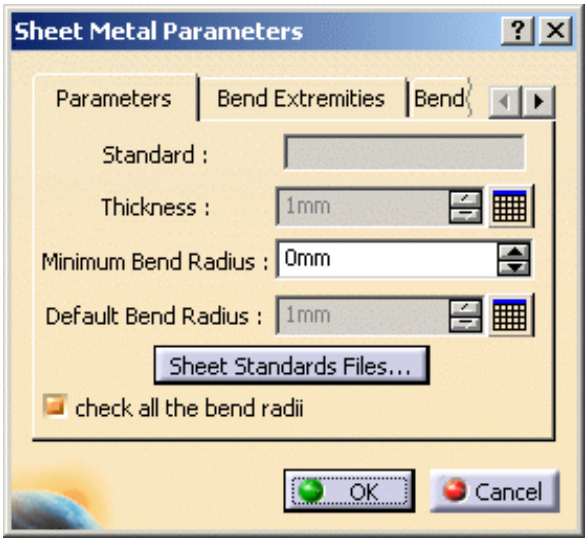
3. Indicate the path to the Sheet Metal table.



These files are available under .txt format or .xls format (only for NT and Windows)

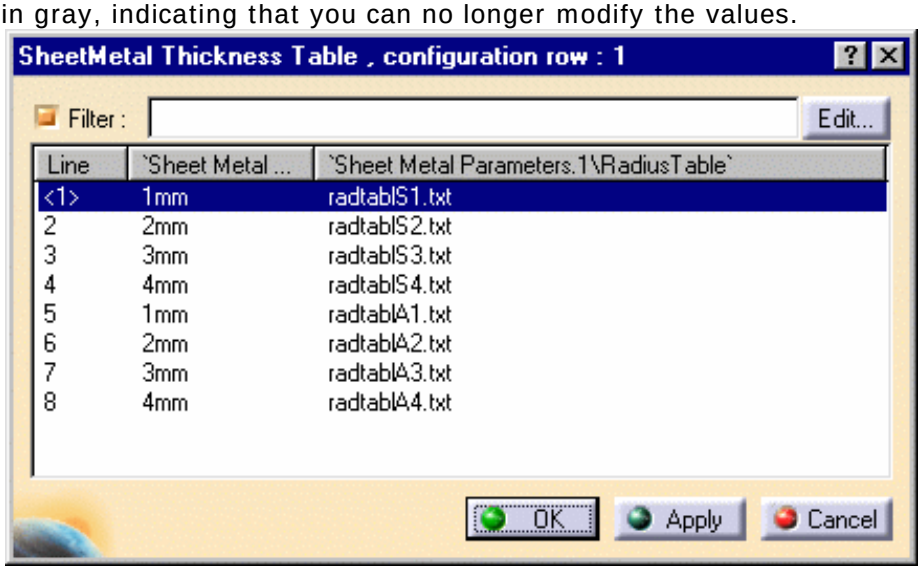
4. Click **Open**.

   In the Sheet Metal Parameters dialog box, the **Design Table** icon appears opposite the Thickness and Bend radius fields.



The parameters are now in gray, indicating that you can no longer modify the values.

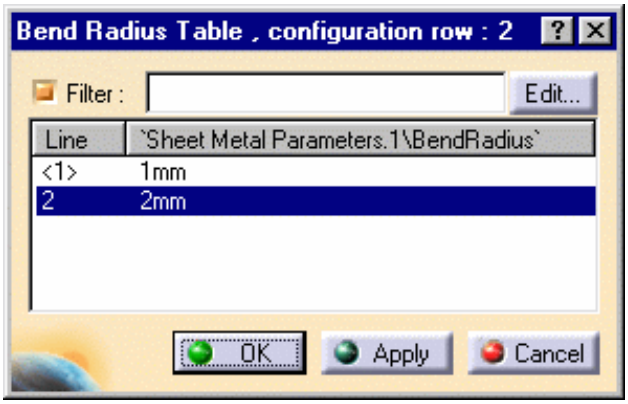
5. Click the Thickness Design Table icon and select the line containing the appropriate parameters (for example Line 1).



Using the **Tools -> Options -> General -> Document** tab, **Other Folders** option, you can specify where the files are located. Refer to Document.
This scenario can work when the .CATPart document and all reference table files (Design & Radius) are located in the same directory. This directory is the current one when the Design table is created, and also when the .CATPart is open.
However, generally speaking, you must reference the complete path indicating where the radius table files are to be found in the RadiusTable column. In this case, regardless of the current directory, the correct tables are located when re-opening the .CATPart document.

6. Click **OK**.
   The parameter values are updated in the Sheet Metal Parameters dialog box.

7. Click the Bend Radius Design Table icon .
8. Select line 2 and click OK. The parameter values are updated in the Sheet Metal Parameters dialog box.

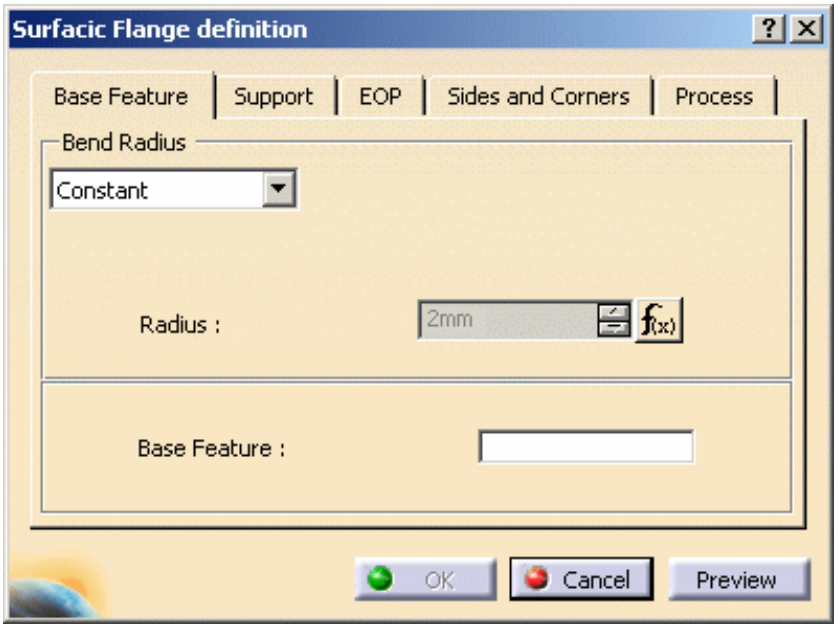


9. Create a flange.

   The Flange definition dialog box reflects the modification for the Radius.

   The default mode, that is to say the formula: **Bend**

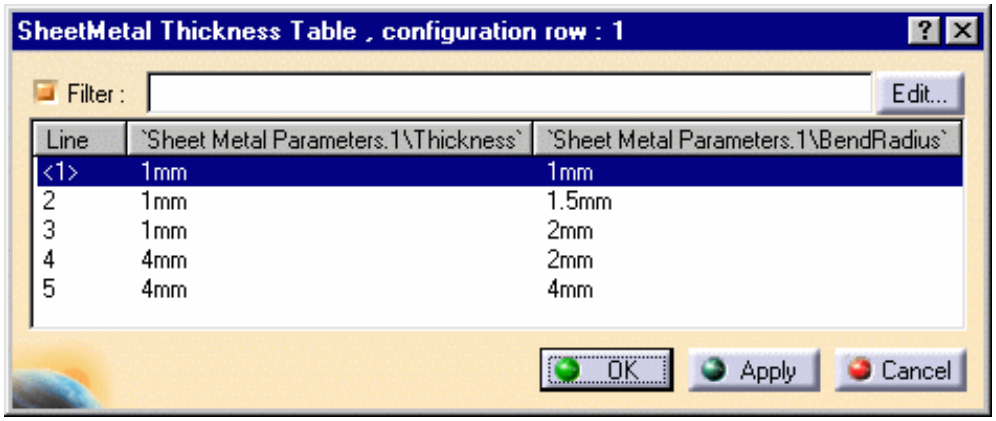
Radius = Part Radius is deactivated.



Using the Sheet Metal Design Tables:

Steps 1 to 4 are identical.

5. Click the Design Table icon and select a line.



6. Click **OK**.
   The parameter values are updated in the Sheet Metal Parameters dialog box.

At that time, the parameters Thickness and Bend radius are driven by the design table.
They are now in gray, indicating that you can no longer modify the values.
Note that if you create a bend, there is no design table: it's the formula which is used.

To disable the access to design tables:

- Select the **Tools -> Options -> Part -> Display** tab and check **Relations:** the **Design Table** icon is displayed in the specification tree.
- Right-click this icon: the contextual menu appears.
- Select **SheetMetal Thickness Table object -> Deactivate**

The relation is no longer used but still exists. It can be activated at any time.

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Glossary

B C D E F J K P W

B

bead A local deformation in the web or a flange.

C

corner relief A feature defined on two flanges, which forms a corner.
It relimits the two flanges and redefines the outer contour of the web between the two flanges.

cutout A feature corresponding to an opening through a feature.
The shape of the opening corresponds to the extrusion of a profile.

D

depth Dimension specifying the geometry of a bead or a joggle.

E

edge of part Element (usually a curve), which defines the length/height of a flange.

F

feature Characteristic form. Features are used to define a part.

flange A feature along the outer section of the web or an existing flange. It is used to connect to another product or to stiffen the part.

J

joggle Feature which causes the flange to be locally deformed.
Usually because the skin which is connected to the web is locally enforced by a strip or stringer (L or T profile).

K

K factor Determines the computation of the unfolded length of flanges.

P

pattern A set of similar features repeated in the same feature or part.

W

web Main constituent of a hydroformed part. Many other features (flanges, holes, etc.) are defined onto this feature.

Index

A B C D E F G H I J L M P R S T U W

A

Aerospace SheetMetal Design features

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    defining
bend extremities
bend radius
    defining
bisecting
    lines
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    catalog

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    command
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unfolding
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Preface

The V5 Sheet Metal Production is a product offering an intuitive and flexible user interface.

Natively integrated, Sheet Metal Production offers the same ease of use and user interface consistency as all V5 applications.

As a scalable product, Sheet Metal Production can be used in cooperation with other current or future companion products in the next CATIA generation such as SheetMetal Design and Generative Drafting. The widest application portfolio in the industry is also accessible through interoperability with CATIA Solutions Version 4 to enable support of the full product development process from initial concept to product in operation.

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Preface

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Using This Product
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Using This Product

This guide is intended for the user who needs to become quickly familiar with the Sheet Metal Production product. The user should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars.

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

The next sections deal with the handling of more detailed capabilities of the product.

Up Next

Where to Find More Information

Prior to reading this book, we recommend that you read the *Infrastructure User's Guide.*

The *Part Design User's Guide*, the *Assembly Design User's Guide*, and the *Sheet Metal Design User's Guide* may also prove useful.

Conventions

Back Up

What's New?

There is no new or improved capabilities in Version 5 Release 12 of the CATIA Sheet Metal Production.

Getting Started

Before getting into the detailed instructions for using Version 5 Sheet Metal Production, the following tutorial provides a step-by-step scenario demonstrating how to use key functionalities.

The main tasks proposed in this section are:

Entering The Workbench
Recognizing Walls
Using the Bending Wizard
Unfolding the Part
Checking Overlapping
Saving As DXF

 All together, these tasks should take about 15 minutes to complete.

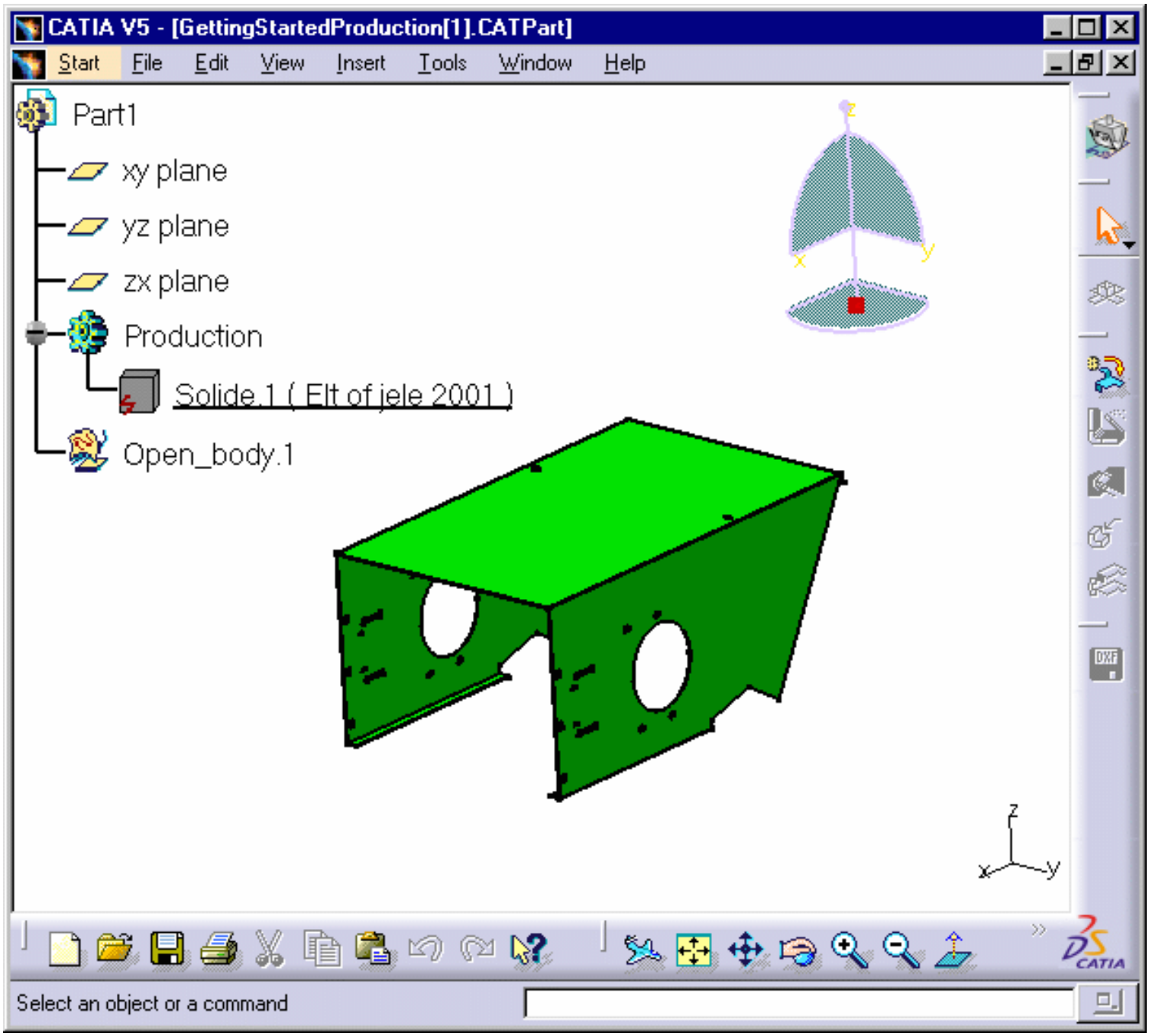
Entering The Workbench

The Sheet Metal Production functions are available when you are in the Part environment.

This task shows how to load a document and enter the Sheet Metal Production workbench.

Open the GettingStartedProduction.CATPart document.

Choose the **Mechanical Design -> Sheet Metal Production** item from the **Start** menu.
The Sheet Metal Production workbench is displayed and ready to use.



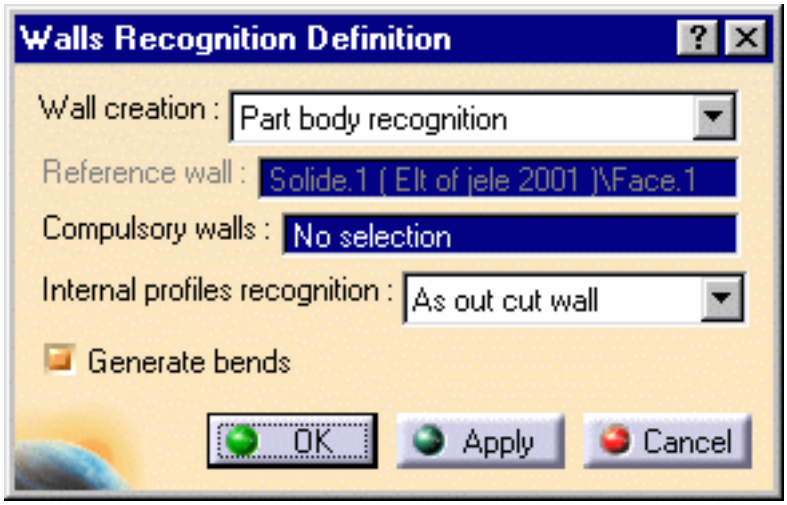
Up Next

Recognizing Walls

This task shows how to automatically create walls from an existing solid.

1. Click the **Walls Recognition** icon 
2. Select a surface as the reference wall.
   We selected the top surface.

   The Wall Recognition Definition dialog box is displayed.



3. Choose the **Wall creation** mode:

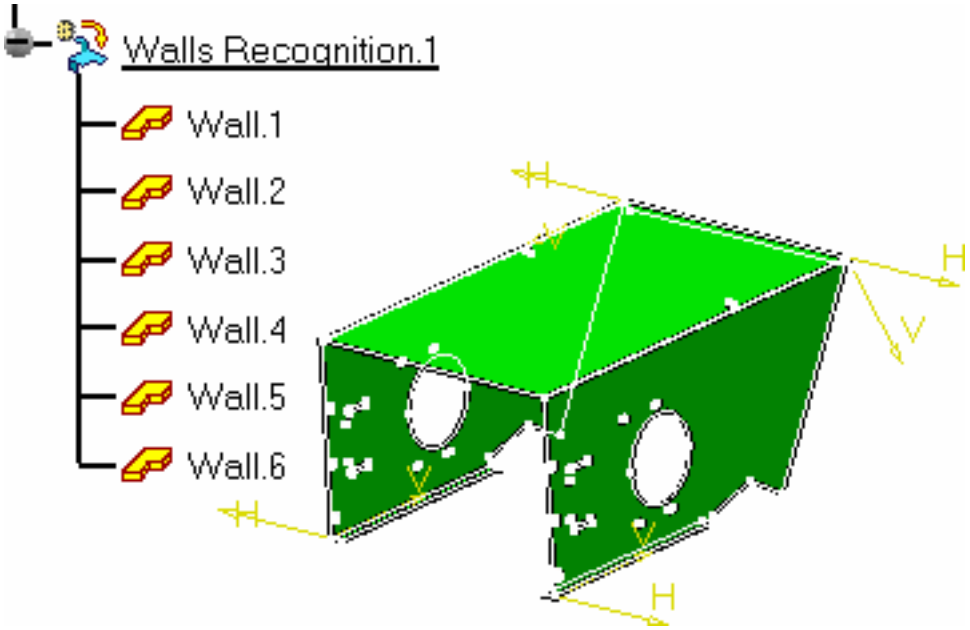
- **Part body recognition**: the whole solid is processed and walls are created wherever possible
- **Only selected faces**: only explicitly selected faces of the solid are processed and the corresponding walls are created.

The **Reference wall** is indicated in the Walls Recognition Definition dialog box for information only (it is grayed out).

4. Do not select any face as the **Compulsory walls**.
5. Choose the **As cut out wall** option as the Internal profiles recognition mode. This means that only walls with inner contours are generated, and not walls and cutout features.
6. Uncheck the **Generate Bends** button to avoid creating bends automatically.
7. Click OK.

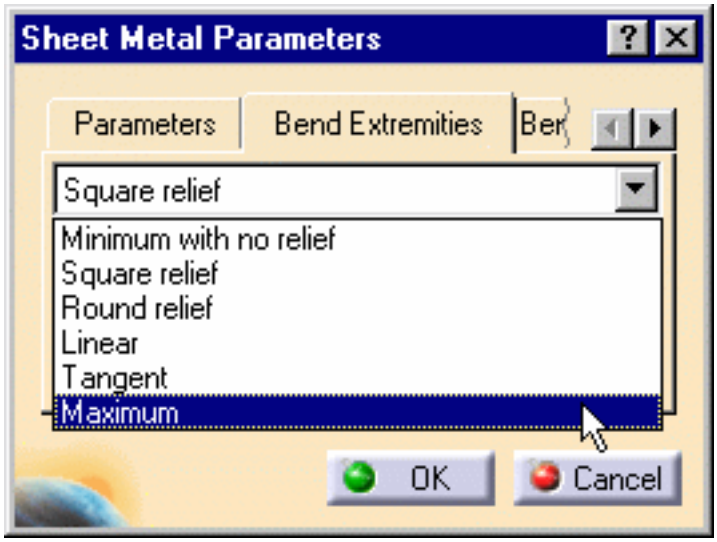
   The walls are generated from the geometry.

   The **Walls Recognition.1** feature is added to the tree view.



At the same time, the Sheet Metal parameters are created, deduced from the Part geometry.

8. Double-click **Sheet Metal Parameters.1** in the Specification Tree.
   The Sheet Metal Parameters dialog box is displayed.
9. Select the **Bend Extremities** tab and choose **Maximum**.
10. Click OK.



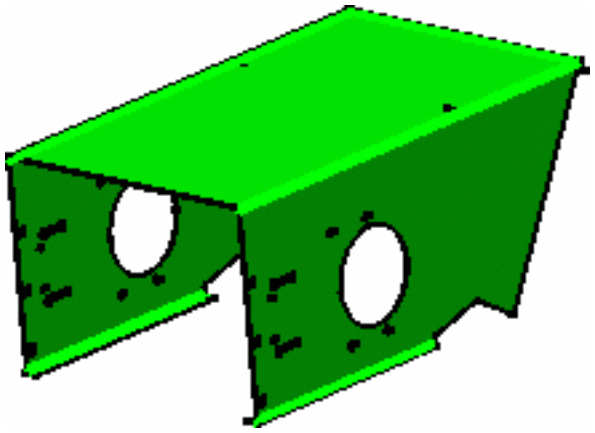
Back Up Next

Using the Bending Wizard

This task shows how to automatically create bends from the generated walls.

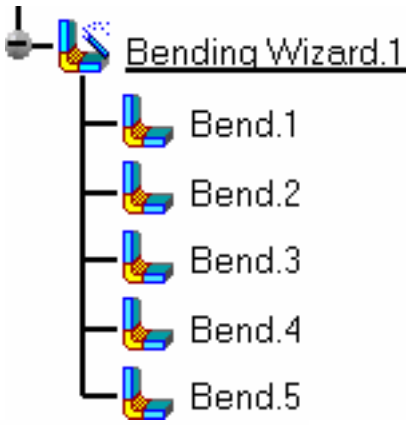
1. Click the **Bending Wizard** icon .
   Bends are previewed.

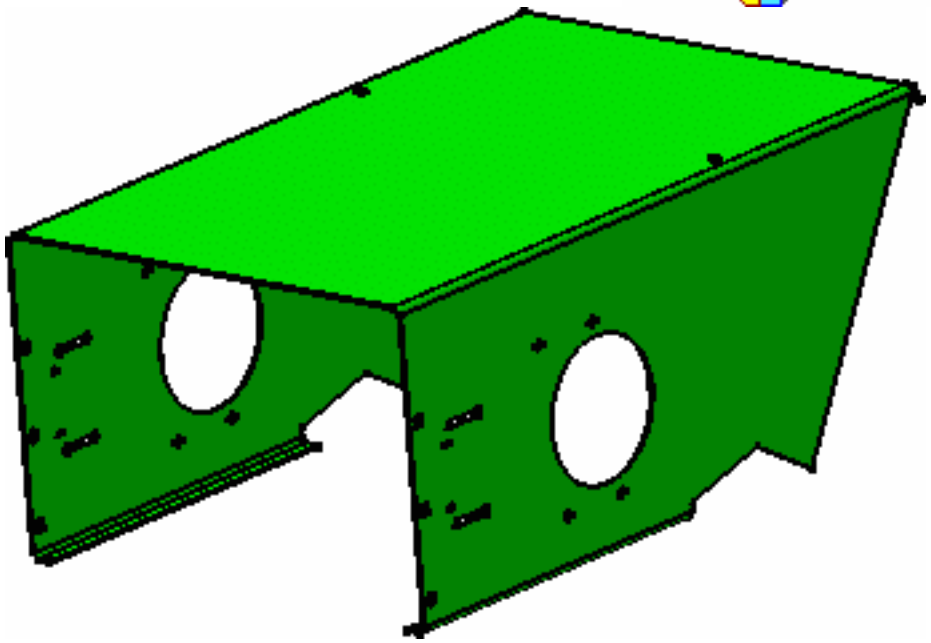
   Bends that can be created are displayed and computed.



In case an ambiguity arise on a bend, it would be displayed in red.

The bends are created and the specification tree updated accordingly.







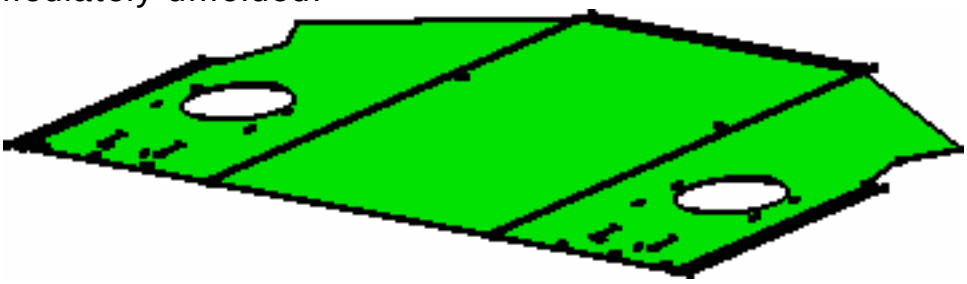
Back Up Next

Unfolding the Part

This task shows how to unfold the part.

1. Click the **Unfold** icon .

The part is immediately unfolded.



To return to the 3D view, simply click the **Unfold** icon again.

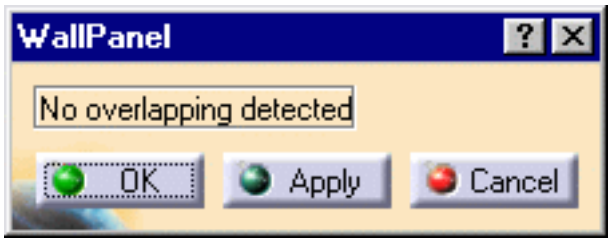
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Checking Overlapping

This task shows how check the overlapping of different areas of the unfolded part.

1. Click the **Check Overlapping** icon .

   In this case, a message is issued indicating that no overlapping is detected.



2. Click OK in the dialog box.

To find out more on overlaps, refer to Checking Overlapping.

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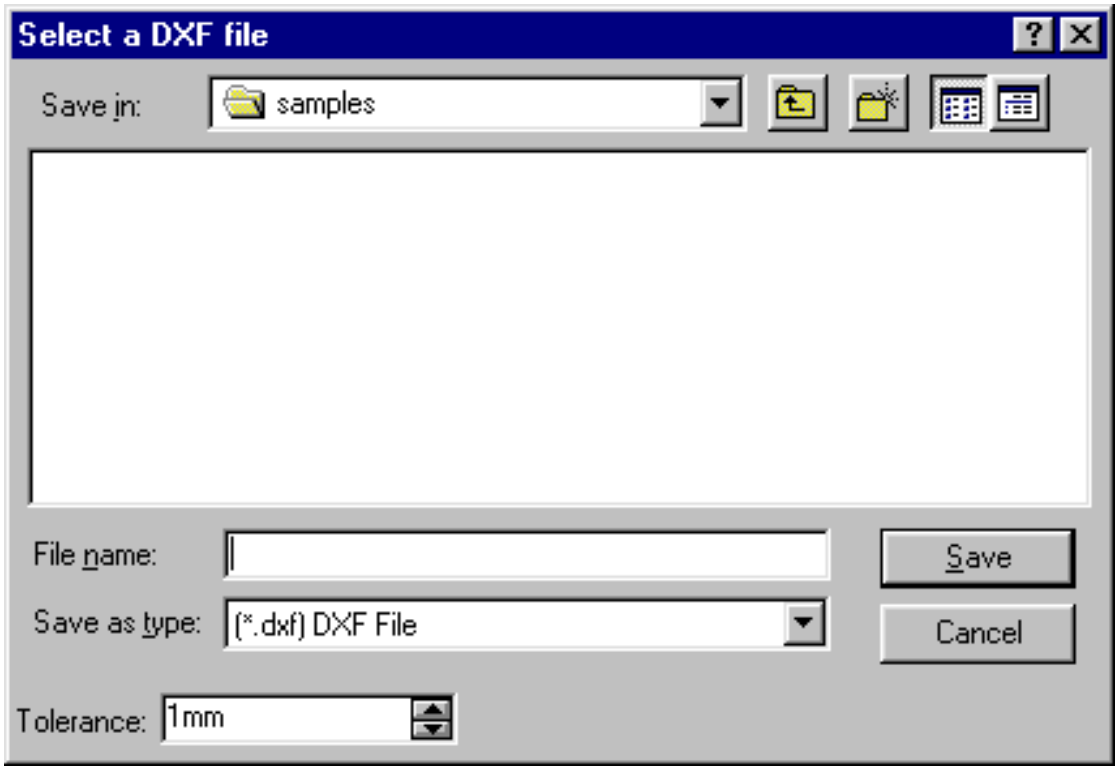
Saving As DXF

This task shows how to save the generated geometry as a DXF document.

Open the SaveAsDXF01.CATPart Document.

1. Click the **Save As DXF** icon .

   The **Select a Dxf file** dialog box is displayed allowing you to navigate to the correct location.



2. Indicate the correct path and file name.
3. Click **Save**.

The geometry has been saved as a DXF type file.

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User Tasks

The User Tasks section explains how to create and modify various kinds of features. The table below lists the information you will find.

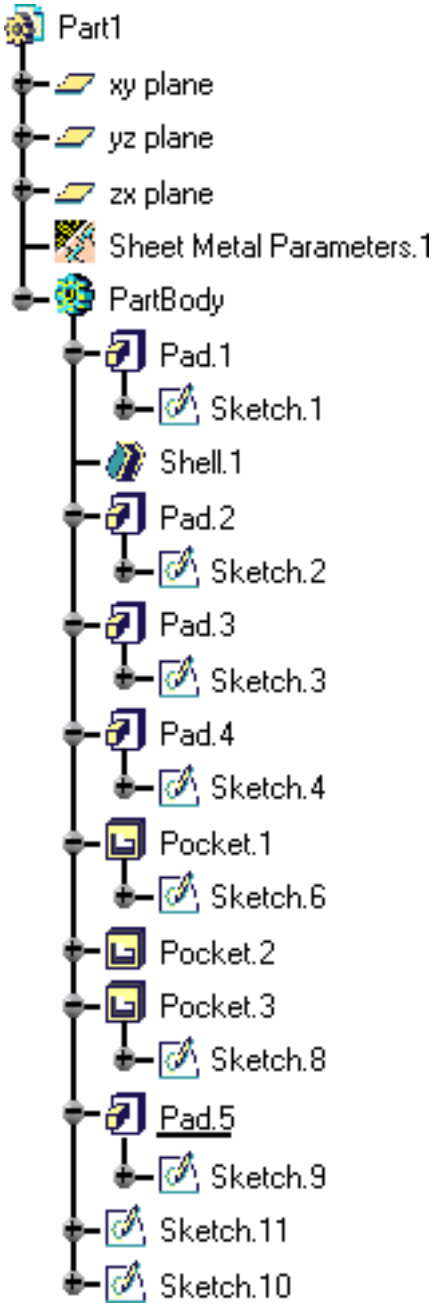
Recognizing Walls
Using the Bending Wizard
Checking Features
Unfolding
Saving As DXF

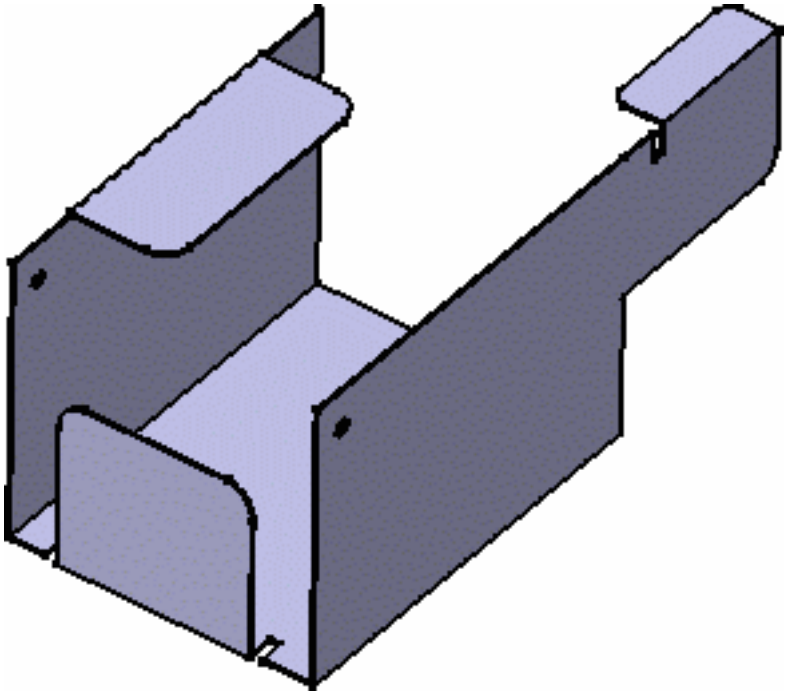
Recognizing Walls

This task shows how to create a Sheet Metal part based on an existing Part, that is recognizing the thin part shapes of the Part as created using the Part Design workbench or from a CATIA Version 4 Solid for example.

Open the RecognizeWalls01.CATPart document from the samples directory.

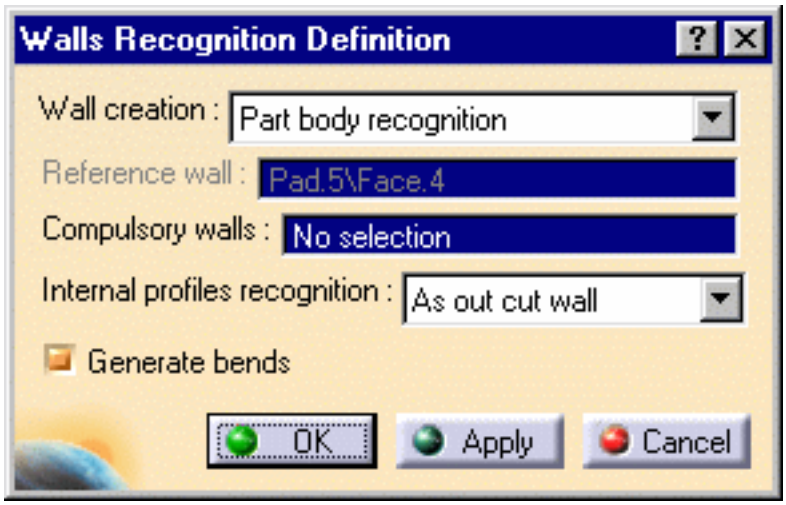
This document contains a part created in the Part Design workbench and looking like this:





1. Click the **Walls Recognition** icon .

The Walls Recognition Definition dialog box is displayed.



3. Choose the **Wall creation** mode:

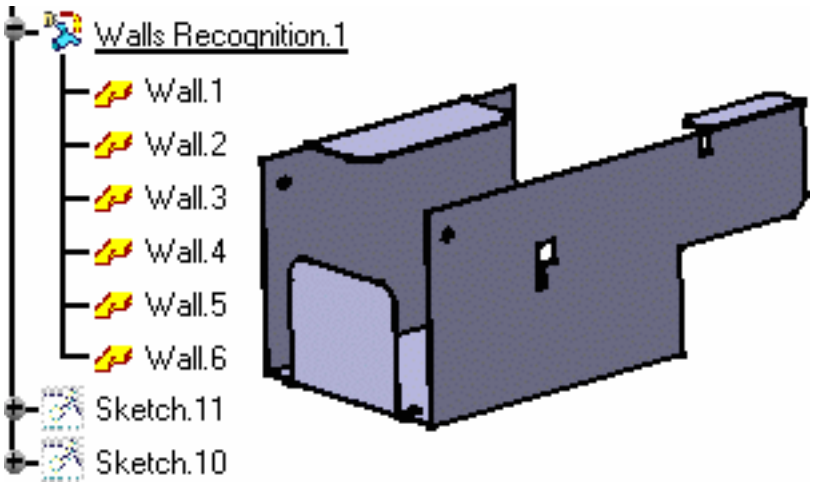
- Part body recognition: the whole solid is processed and walls are created wherever possible
- Only selected faces: only explicitly selected faces of the solid are processed and the corresponding walls are created.

The **Reference wall** is indicated in the Walls Recognition Definition dialog box for information only (it is grayed out).

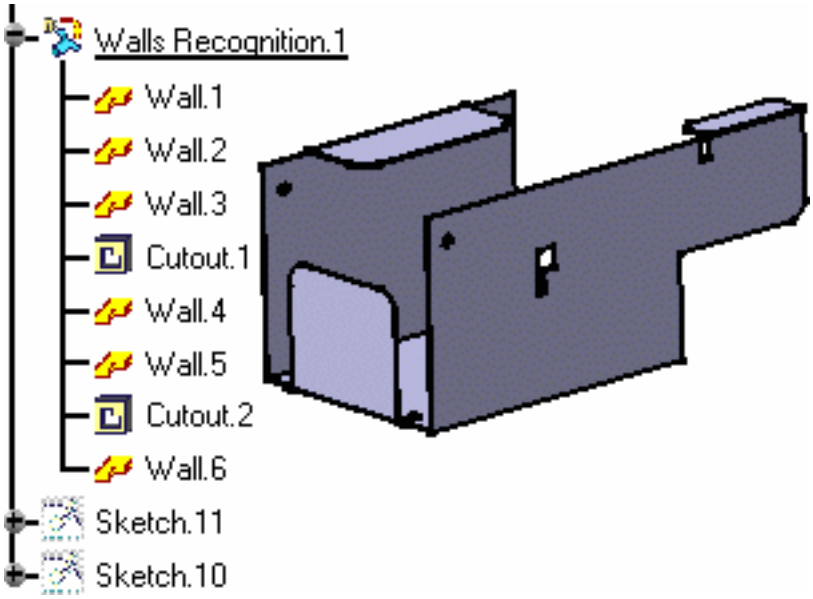
4. Select faces as the **Compulsory walls**.
These are faces from which the walls are to be generated when there might be an ambiguity. For example, if the initial part is a box, you will need to select two opposite inner faces and outer faces on the other two sides of the box, in order to avoid overlapping when generating the walls.

5. Set the Internal profiles recognition mode:

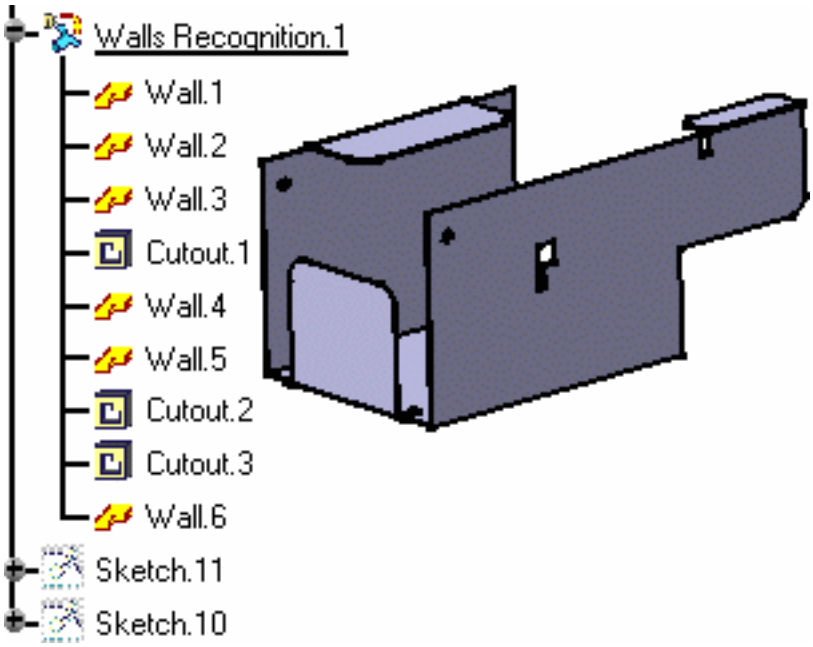
- **As cut out wall**: generates walls with inner contours (no cutout feature is generated)



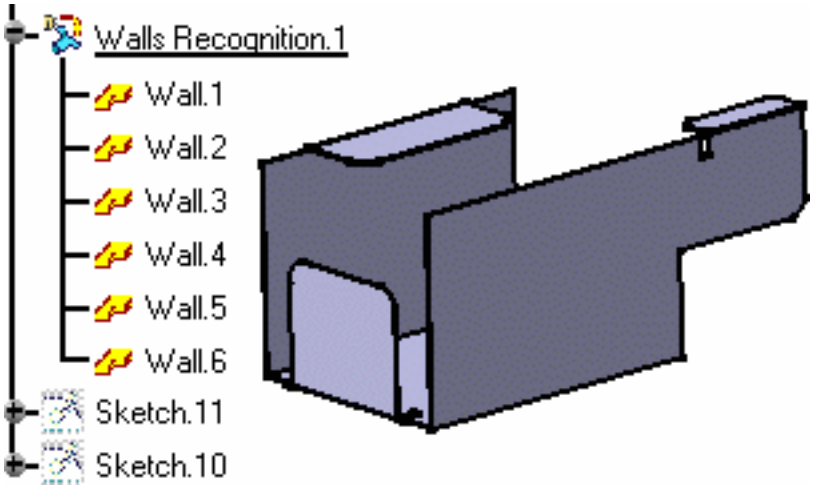
- **One cut out by wall**: regardless of how many pockets there are on a face of the solid, only one cutout feature is generated per wall



- **One cut out by profile**: for each inner contour on the sketch-based solid, a cutout feature is generated



- **None**: whether there are pockets on the solid faces, or not, no cutout feature is created in the resulting SheetMetal features.

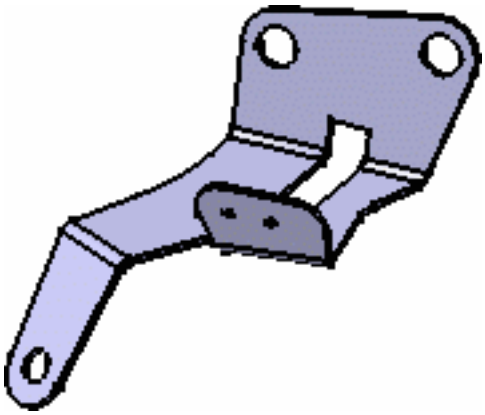


The **Generate Bends** check button allows the automatic creation of bends as the walls are being created, wherever applicable.

6. Click OK to generate the walls, and bends if any.

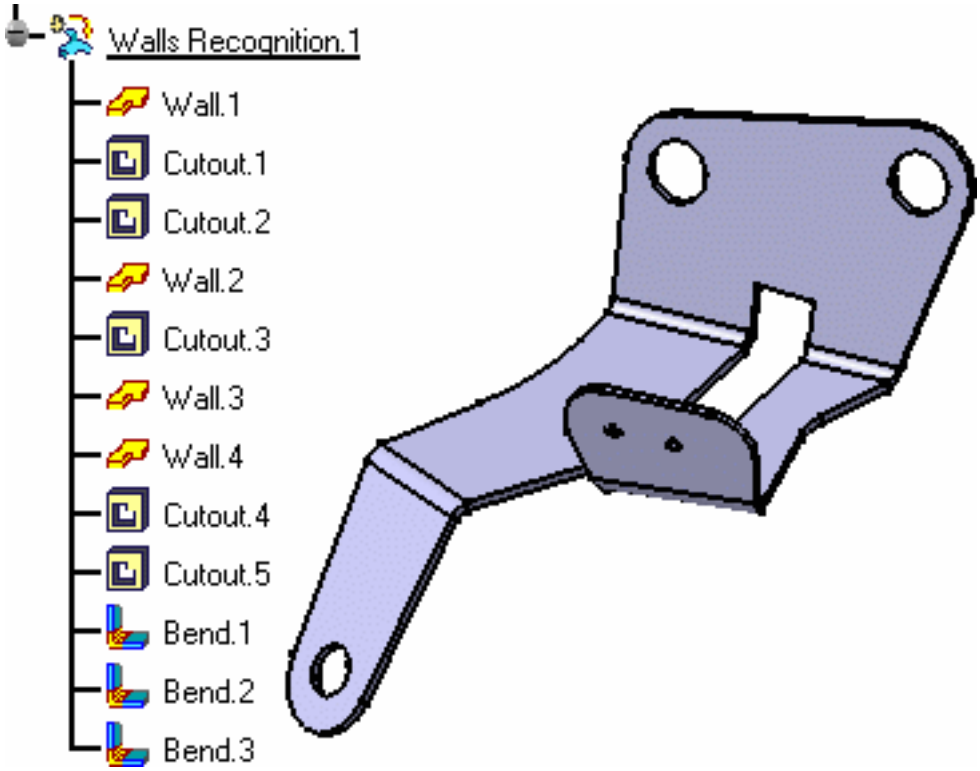
   Walls are automatically generated from the Part Design geometry.

Open the RecognizeWalls02.CATPart document from the samples directory. This part contains a filleted CATIA V4 solid presenting different orientations.



1. Click the **Walls Recognition** icon .
2. Click any face of the part as the reference wall.
   The Wall Recognition Definition dialog box is displayed.
3. Make sure the **Generate Bends** button is checked.
   It allows the automatic creation of bends, wherever applicable, as the walls are being created, .

   Walls and bends, due to the presence of fillets in the initial part, are automatically generated.



Note that once the part has been transformed into Sheet Metal features, the Sheet Metal Parameters are also created in the specification tree.

Double-click the Sheet Metal Parameters entry from the specification tree to see them. The **Thickness** parameter cannot be modified because it is based, like the bend extremities and radius, on the initial solid geometry . However you can modify these parameters (bend radius and bend extremities) to be taken into account for sheet metal features other than the "recognized" ones.
The bend allowance, being used to unfold the part, and the corner relief affect all features, and therefore can be edited even for "recognized" features.

Uncheck the **Generate Bends** button, if you do not wish bends to be created automatically.

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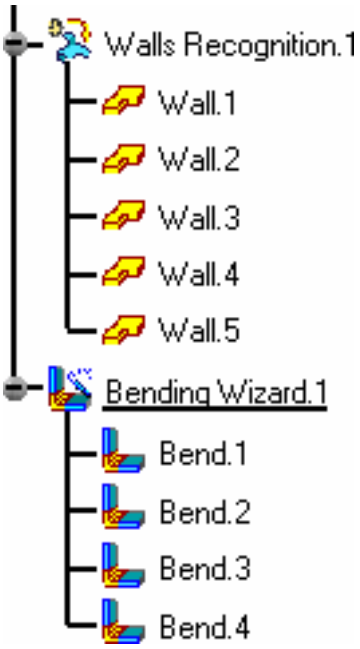
Using the Bending Wizard

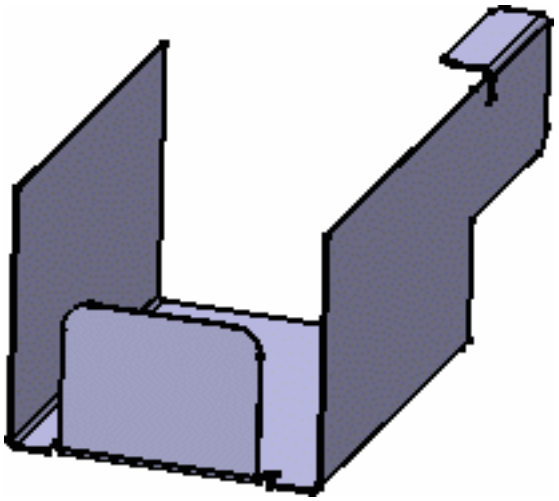
This task shows how to automatically create bends onto existing walls.

Open the BendingWizard01.CATPart document from the samples directory.

1. Click the **Bending Wizard** icon

   The bends are created.

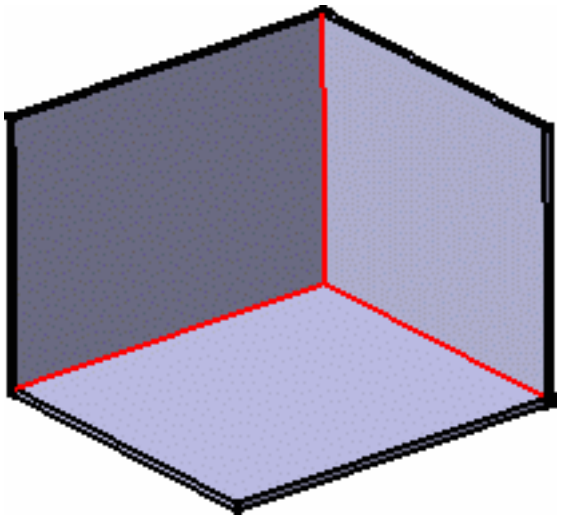




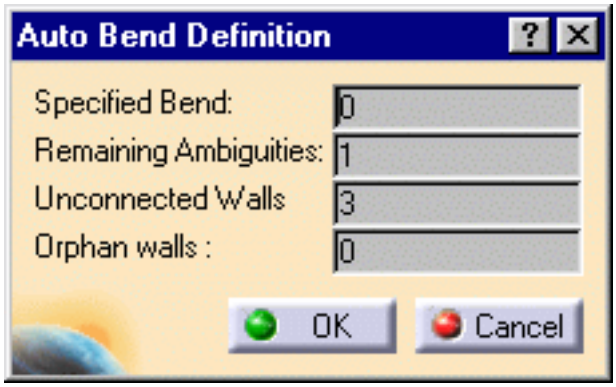
Open the BendingWizard02.CATPart document from the samples directory.

1. Click the **Bending Wizard** icon

   The bends cannot be created as there is an ambiguity because too many bends would overlap.

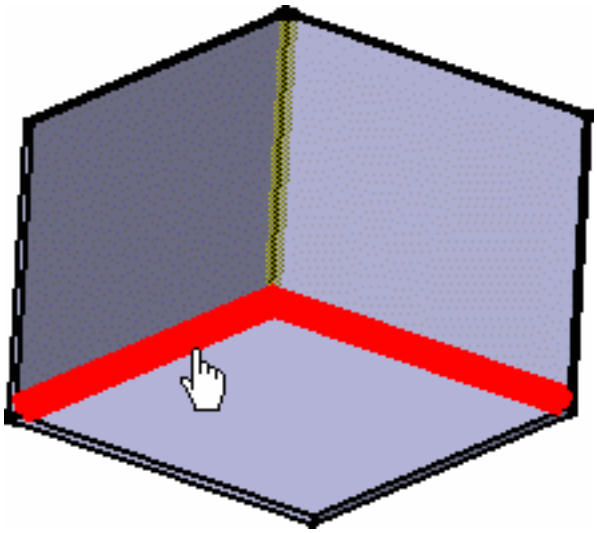


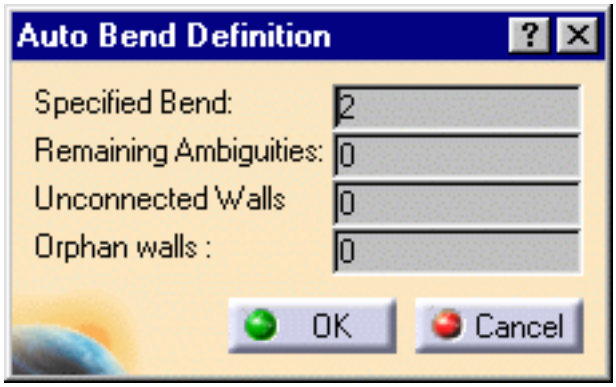
The Auto Bend Definition dialog box is displayed indicating how many inconsistencies have been detected, and the number of walls not yet connected by a bend (**Unconnected walls**).



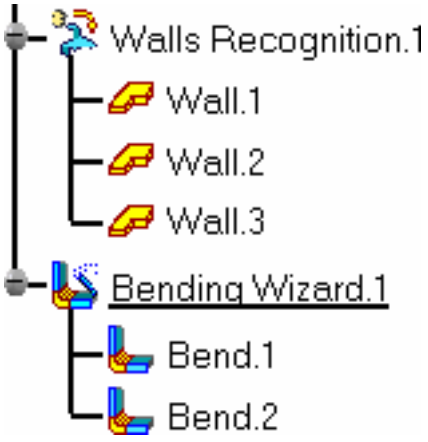
Orphan walls are walls that are not connected to any other wall.

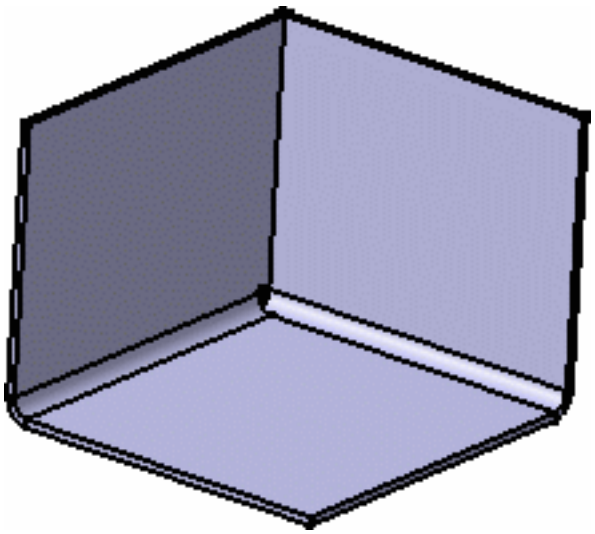
2. Explicitly select the bends that should be created. They are immediately previewed and the dialog box reflects your selection.





3. Click OK in the dialog box to create the bends.







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Checking Features

This section gives you tools to check the Sheet Metal Part prior to production:


Check overlapping: click the icon to highlight the overlapping areas, then click Apply to create a curve.


Check bend radius: click the icon to generate the list of bends not using the default bend radius parameter value.


Check feature fold/unfold visibility: click the icon to generate the list of Sheet Metal features that are not visible in the unfolded view.

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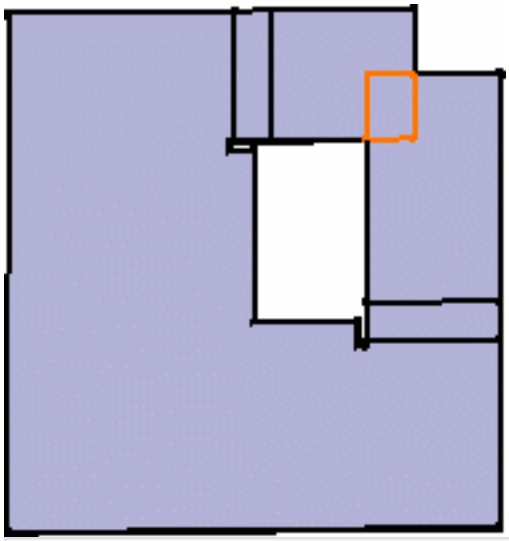
Checking Overlapping

This task shows how check the overlapping of different areas of the part. This is usually done when the part has been designed, bends created and the part is unfolded. Overlapping between areas of the part are highlighted.

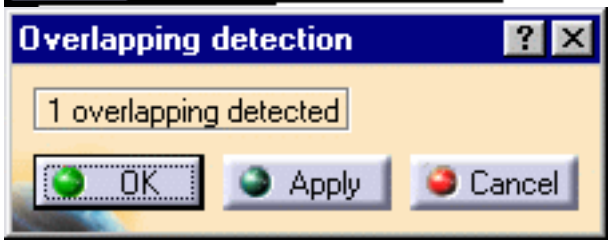
Open the Overlap01.CATPart document from the samples directory.

1. Click the **Unfold** icon to unfold the part.
2. Click the **Check Overlapping** icon .

   The area with the overlapping is highlighted in red.

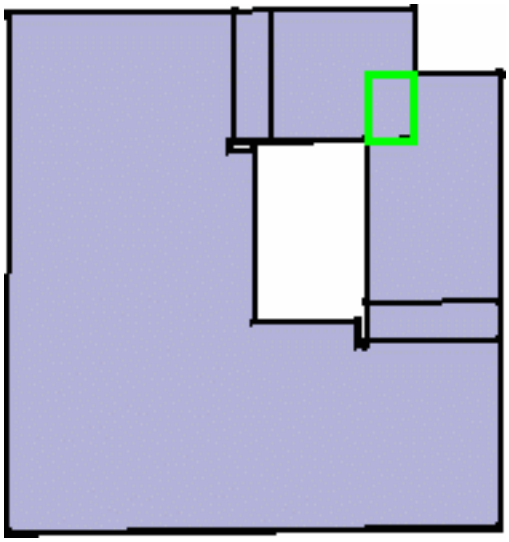


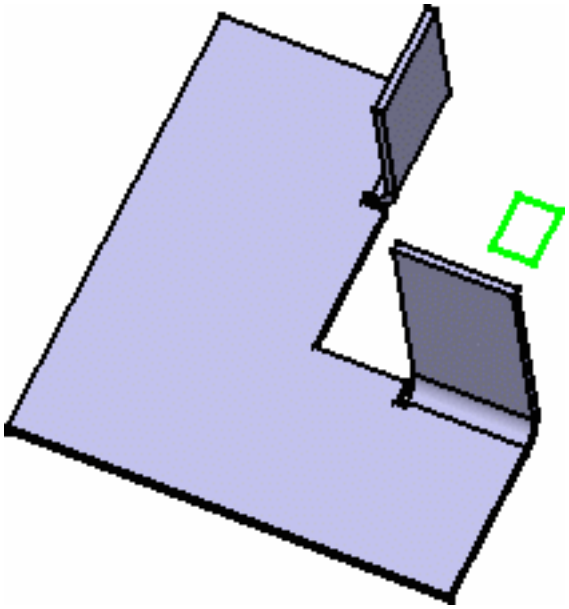
The Overlapping Detection dialog box is displayed indicating how many overlaps are present on the analyzed part.



3. Click Apply.
   A curve is generated. This curve can then be used to solve the overlapping, by redesigning the involved feature's sketches or specifications.

4. Click OK.

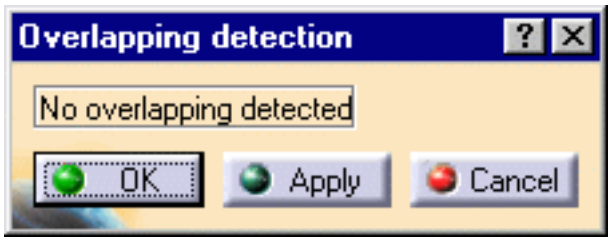




Generated curve on unfolded view

Generated curve on folded view

When no overlapping is detected, the Overlapping Detection dialog box indicates so:



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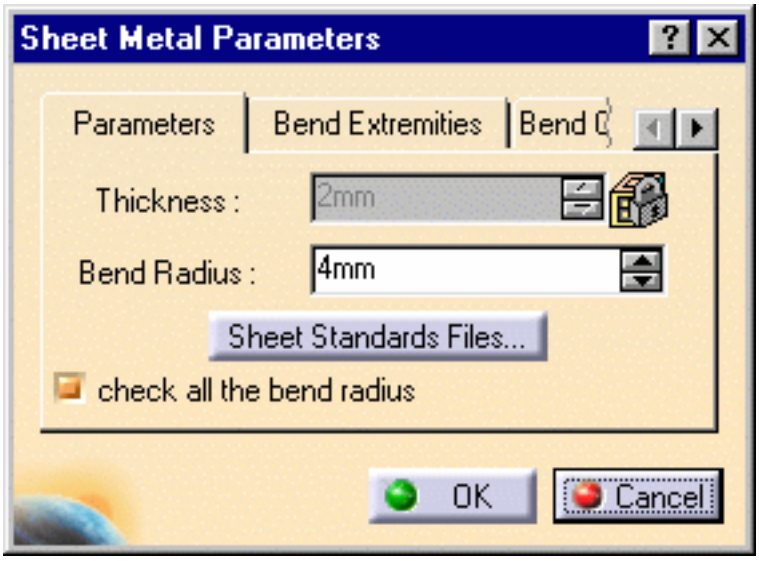
Checking Bend Radius

This task shows how check the bends of a Sheet Metal part, i.e. to detect the bends that do not use the default bend radius parameter value, thus allowing the production teams to choose the correct tools to process the part.

Open the BendRadius01.CATPart document from the samples directory.

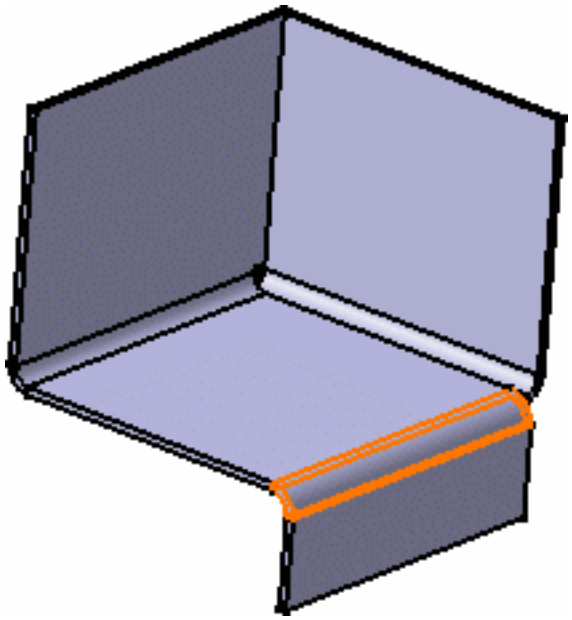
1. Double-click the Sheet Metal Parameters from the specification tree.

   From the Sheet Metal Parameters dialog box you can see that the standard bend radius for the current document is 4mm.

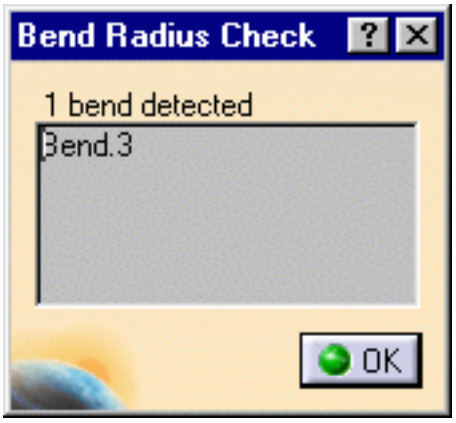


2. Click Cancel in the Sheet Metal Parameters dialog box.
3. Click the **Check Bend Radius** icon .

   The bends whose radius value is different than the standard value as defined in the Sheet Metal Parameters dialog box, Parameters tab, are highlighted.



Furthermore, a list of these bends is displayed in a dialog box.



3. Click OK.

If you display the Sheet Metal Parameters dialog box and that a bend differ from the set value, clicking OK will automatically highlight the bend involved and display the Bend Radius Check dialog box.

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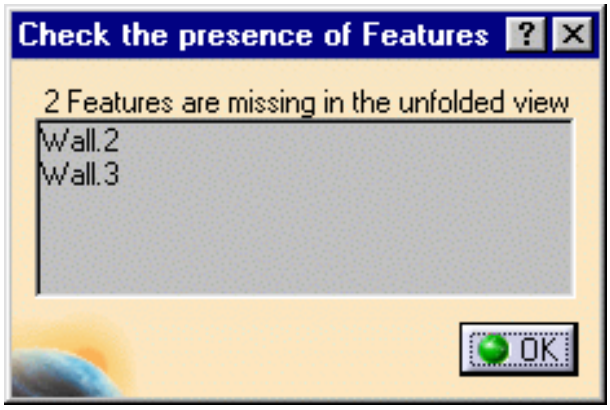
Checking Feature Fold Visibility

This task shows how to check the fold/unfold visibility of a Sheet Metal Part. Indeed in some cases, you can notice that a few features are no longer visible once the part is unfolded.
This capability gives you the list of these elements.

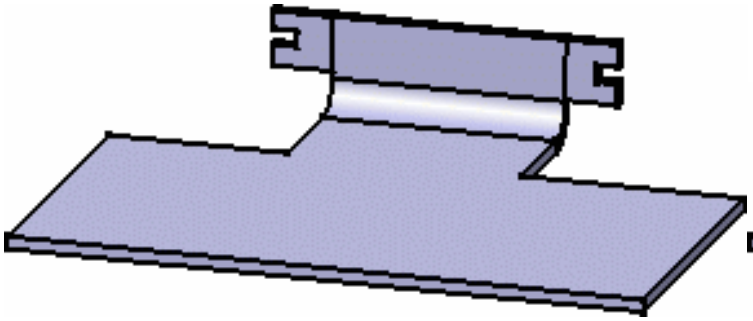
Open the TangentWall1.CATPart document from the samples directory.

1. Click the **Check Feature** icon .

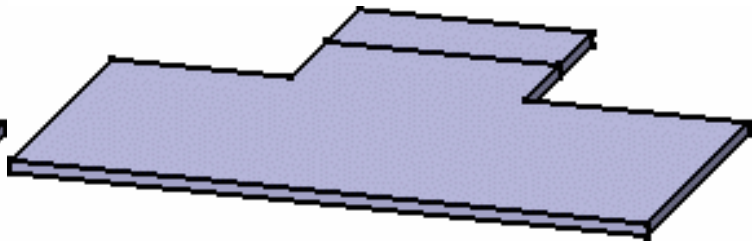
   The list of the features that will not be visible is you unfold the part, using the **Unfold** icon is displayed in the dialog box.



2. Click OK to exit the list.



Folded part with all features visible



Unfolded part with features not visible

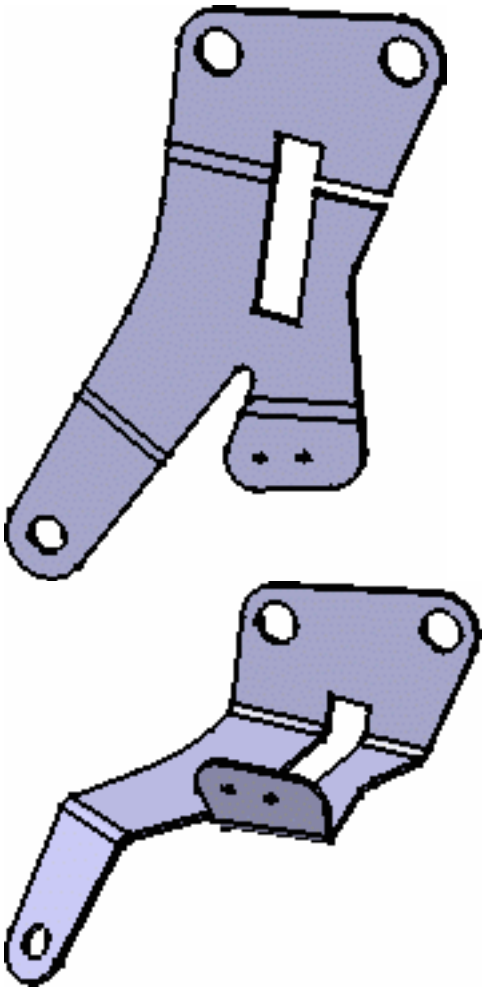
Back Up

Unfolding

This task shows how unfold/fold a part.

Open the Unfold1.CATPart document from the samples directory.

1. Click the **Unfold** icon .



2. Click the **Unfold** icon again to return to the view of the folded part.

Back Up Next

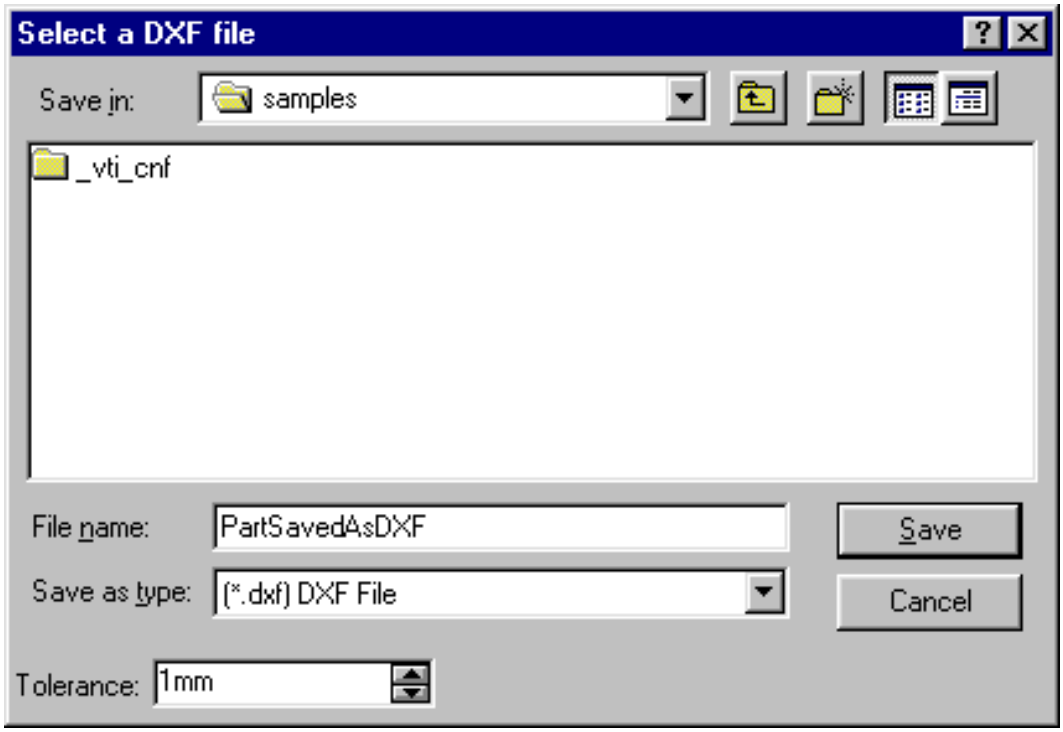
Saving As DXF

This task shows how save the generated geometry as a DXF document.

Open the SaveAsDXF01.CATPart document from the samples directory.

1. Click the **Save As DXF** icon .

   The **Select a Dxf file** dialog box is displayed allowing you to navigate to the correct location.



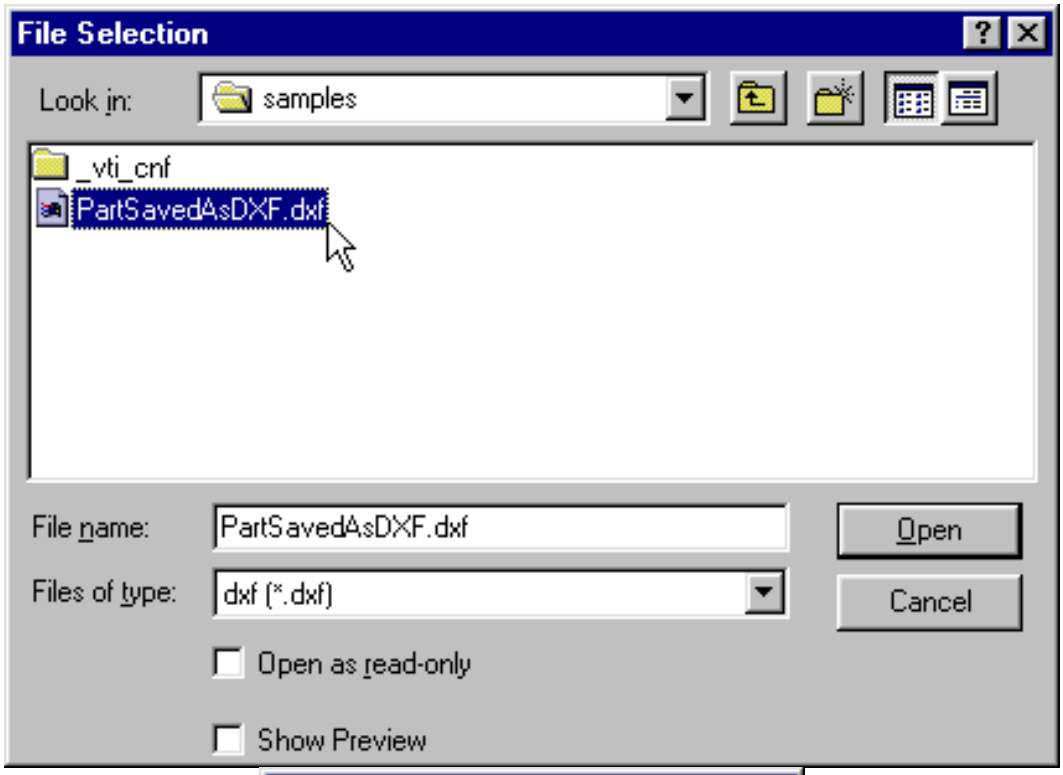
2. Indicate the correct path and file name (we saved it as PartSavedAsDXF.dxf)..
3. Click **Save**.

The geometry has been saved, and can be imported as a DXF file in any system supporting this type.

4. Close the CATPart Document.
5. Click **File -> Open**.

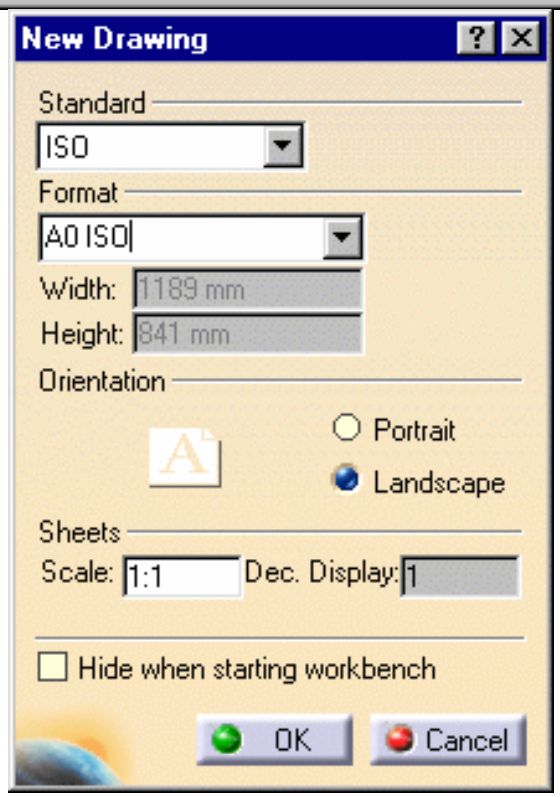
6. From the File Selection dialog box, choose the .dxf file type, then select the saved part (PartSavedAsDXF.dxf).

7. Click Open.

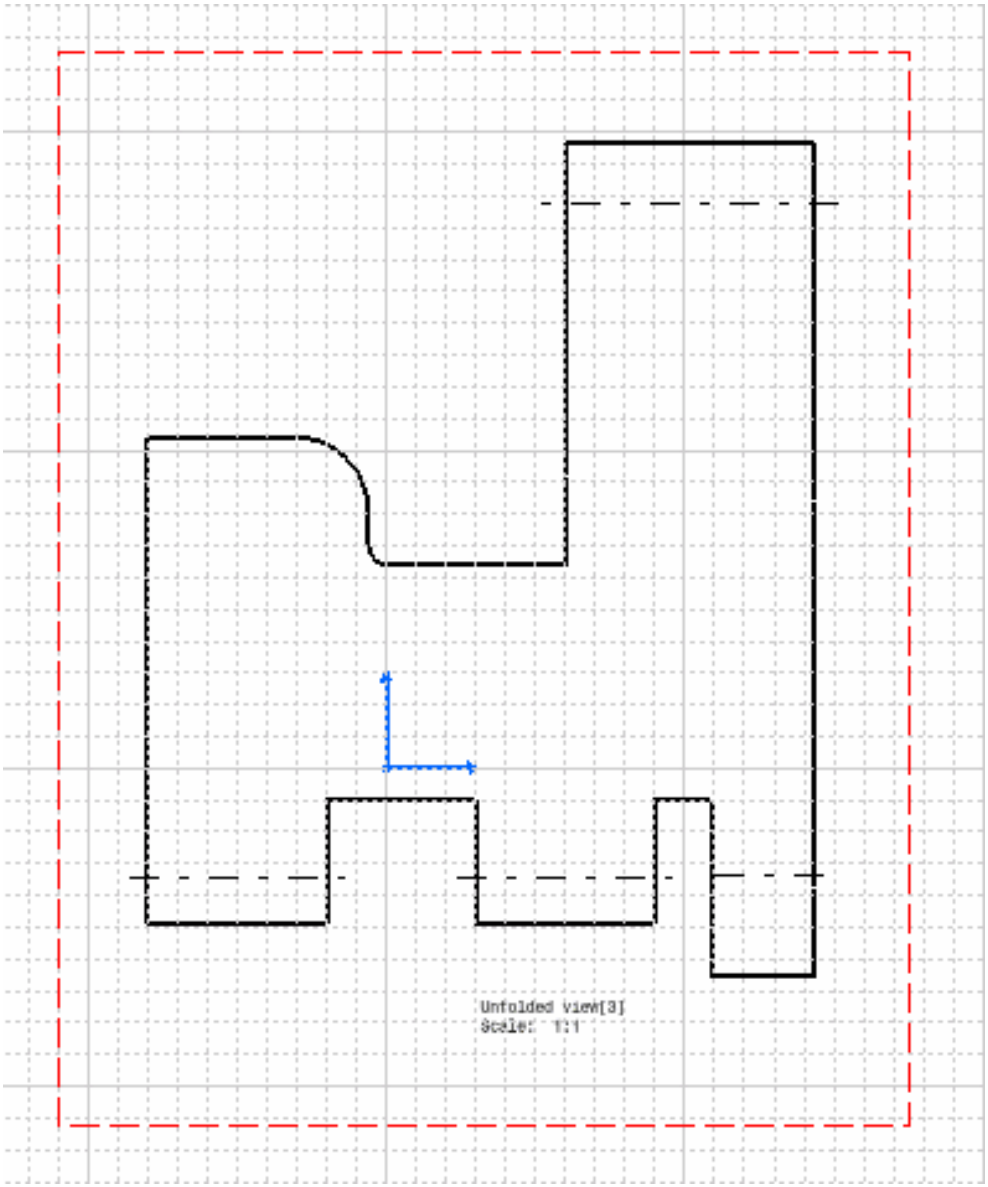


The New Drawing dialog box is automatically displayed because the .dxf type is recognized as being a drafting type.

8. Click OK in the New Drawing dialog box.



The unfolded view of the part is created within the CATDrawing document.



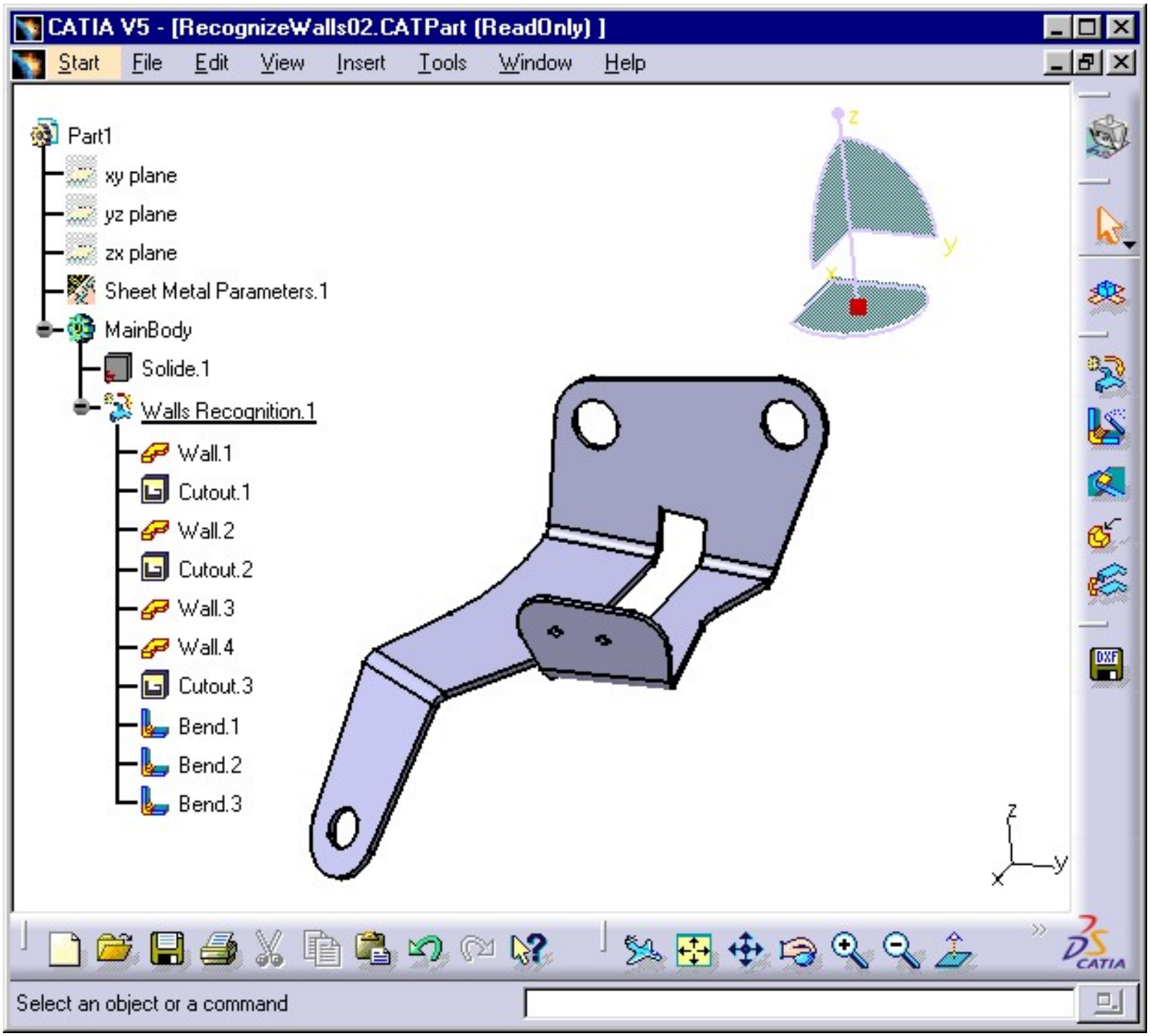
Note that the axes of bends and planar hems, tear drops, or flanges are automatically displayed on the drawing.

Refer also to DXF/DWG Settings.

Back Up

Workbench Description

The Sheet Metal Production Version 5 application window looks like this:
Click the hotspots to display the related documentation.



Menu Bar
Toolbars
Specification Tree

Sheet Metal Production Menu Bar

The various menus and menu commands that are specific to Sheet Metal Production Version 5 are described below.

Start | File | Edit | View | Insert | Tools | Windows | Help

Tasks corresponding to the Start, File, Edit, View, Tools, Windows and Help menu commands are described in the *Infrastructure User's Guide*.

Insert



For...	See...
Unfold...	See Unfolding
Walls Recognition...	See Recognizing Walls
Bending Wizard...	See Using the Bending Wizard
Check Overlapping...	See Checking Overlapping
Check Bends radius	See Checking Bend Radius
Check Feature	See Checking Feature Fold/Unfold Visibility
Save As DXF...	See Saving As DXF

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Toolbars

The Sheet Metal Production Version 5 tool bars look like this:



 See Unfolding the Part

 See Recognizing Walls

 See Using the Bending Wizard

 See Checking Overlapping

 See Checking Bend Radius

 See Checking Feature Fold/Unfold Visibility

 See Saving As Dxf

Back Up Next

Specification Tree

Within the Sheet Metal Production workbench, you can generate a number of Sheet Metal Features that are identified in the specification tree by the following icons.

Further information on general symbols in the specification tree are available in Symbols Used in the Specification Tree.

 Sheet Metal Parameters

 Walls

 Wall On Edge

 Flange

 Hem

 Tear Drop

 User-defined Flange

 Automatic Bends

 Bends from the Wizard

 Bend

 Point Stamp

 Extruded Hole

 Curve Stamp

 Surface Stamp

 Bridge

 Louver

 Stiffening Rib

 User-Defined Stamp

 Cutout

 Rectangular Pattern

 Circular Pattern

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B

bend A feature joining two walls

bend extremity Axial relimitation for a straight bend

C

cutout A feature corresponding to an opening through a feature.
The shape of the opening corresponds to the extrusion of a profile.

E

extrusion A feature created by extruding a profile and adding thickness

F

flange A feature created by sweeping a profile along a spine.
The different flanges or swept walls available are:
simple and swept flange, hem and tear drop.

P

pattern A set of similar features repeated in the same feature or part

profile An open or closed contour including arcs and lines created by the **Profile** command in the Sketcher workbench

R

reference wall The first created wall; when unfolding the part, it is the fixed wall.

S

stamping A feature created by embossing sheet metal.
The different stampings available are:
point, extruded hole, curve, surface, bridge, louver and stiffening rib.

W

wall A feature created by adding thickness to a profile

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analyzing
parts
automatic bends

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bend radius
checking
Bending Wizard
command
bends
creating

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Version 5 Release 12
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3D Functional Tolerancing & Annotation



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- Where to Find More Information
- Conventions

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Preface

3D Functional Tolerancing & Annotation is a new-generation product which 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 3D 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.

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) or 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.

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Preface

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Using this Guide
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Using this Guide

This guide is intended for the user who needs to become quickly familiar with the Functional Tolerancing and Annotation, Product Functional Tolerancing and Annotation and Process Tolerancing and Annotation workbenches.

Note that the functionalities available are exactly the same from one workbench to another, even though the scenarios provided in this guide use parts (CATPart documents) as examples.

Before reading this guide, the user should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars.

To get the most out of this guide, we suggest you start reading and performing the step-by-step tutorial Getting Started. Once you have finished, you should move on to the next sections, where you will find detailed explanations regarding the handling of the product. You can also take a look at the Workbench Description section describing the workbench.

To perform the scenarios, you will use sample documents contained in the C:\Program Files\Dassault Systemes\Bxxdoc\English\online\fdtug\samples folder (where xx in Bxxdoc stands for the current release number). For more information on accessing sample documents, refer to Accessing Sample Documents in the *Infrastructure User's Guide.*

Up Next

Where to Find More Information

Prior to reading this book, we recommend that you read Infrastructure User's Guide.

Part Design User's Guide as well as Generative Drafting User's Guide may prove useful too. Certain conventions are used in V5 documentation to help you recognize and understand important concepts and specifications.

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Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications. The following text conventions may be used:

- The titles of CATIA documents *appear in this manner* throughout the text.
- **File** -> **New** identifies the commands to be used.

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)

Graphic conventions are denoted as follows:

 indicates the estimated time to accomplish a task.

 indicates a target of a task.

 indicates the prerequisites.

 indicates the scenario of a task.

 indicates tips

 indicates a warning.

 indicates information.

 indicates basic concepts.

 indicates methodological information.

 indicates reference information.

 indicates information regarding settings, customization, etc.

 indicates the end of a task.

 indicates functionalities that are new or enhanced with this Release. Enhancements can also be identified by a blue-colored background in the left-hand margin or on the text itself.

 indicates functionalities that are P1-specific.

 indicates functionalities that are P2-specific.

 indicates functionalities that are P3-specific.

 allows you to switch back the full-window viewing mode.

These icons in the table of contents correspond to the entries or mode.

 "Site Map".

 "Split View" mode.

 "What's New".

 "Preface".

 "Getting Started".

 "Basic Tasks".

 "User Tasks" or the "Advanced Tasks".

 "Workbench Description".

 "Customizing".

 "Reference".



"Methodology".



"Glossary".



"Index".

What's New?

New Functionalities

Managing view/annotation plane associativity
: You can now make views associative to the geometry so that any modification applied to the geometry or to the axis system will be reflected in the view definition.

Aligned section view and section cut support
: You can now define aligned section views and section cuts ready for 2D extraction.

Offset section view and section cut support
: You can now define offset section views and section cuts ready for 2D extraction.

Thread dimensioning and tolerancing
: You can now tolerance thread features.

Coordinate dimensions
: You can now create coordinate dimensions based on the coordinates of selected points.

Stacked dimensions
: You can now create stacked dimensions by selecting several geometrical elements.

Cumulated dimensions
: You can now create cumulated dimensions by selecting several geometrical elements.

Curvilinear length dimensions
: You can now create and manage curvilinear length dimensions in 3D. A curvilinear length dimension measures the overall length of a curve.

Isolated annotations
: You can now create and manage isolated annotations, i.e. annotations that are not linked to any geometry. Isolated annotations are non-semantic.

Enhanced Functionalities

Query, filter and search using Part Design feature or Generative Shape Design feature
: You can now query, filter and search for 3D annotations by selecting any Part Design or Generative Shape Design feature in the specification tree.
For more information on using search, refer to the *Infrastructure User's Guide*.

Creating annotations by selecting Part Design feature or Generative Shape Design feature
: You can now select the geometry on which a 3D annotation has to be created by selecting any Part Design or Generative Shape Design feature in the specification tree.
All 3D annotations support this capability:
- Texts
- Flag notes
- Datum
- Datum targets
- Geometrical tolerances
- Roughness symbols

- Note Object Attributes

Generative Dimension enhancements

The Generative Dimension command now supports Part Design chamfer features, thread features and threaded hole dimensions.

New text frame

A new text frame is now available: parallelogram.

Customizing Settings

Section cut and section view associativity to 3D geometry

A new setting now lets you create views that will be associative to the geometry selected during their creation. In this case, any modification applied to the geometry or to the axis system will be reflected in the view definition.

View/Annotation Plane Display settings moved to a new tab

View/Annotation Plane Display settings have been moved from the Display tab to the new View/Annotation Plane tab.

Annotations in specification tree

A new setting now lets you specify where 3D annotation features should be displayed in the specification tree: under the annotation set node (as is the case today), under the Part Design and Generative Shape Design feature nodes, or under the view (annotation plane) node.

Annotation parameters in specification tree

A new setting now lets you specify that you want to display the annotation parameters under the annotation feature node. In this case, the annotation will be modifiable by directly editing its parameters.

Getting Started

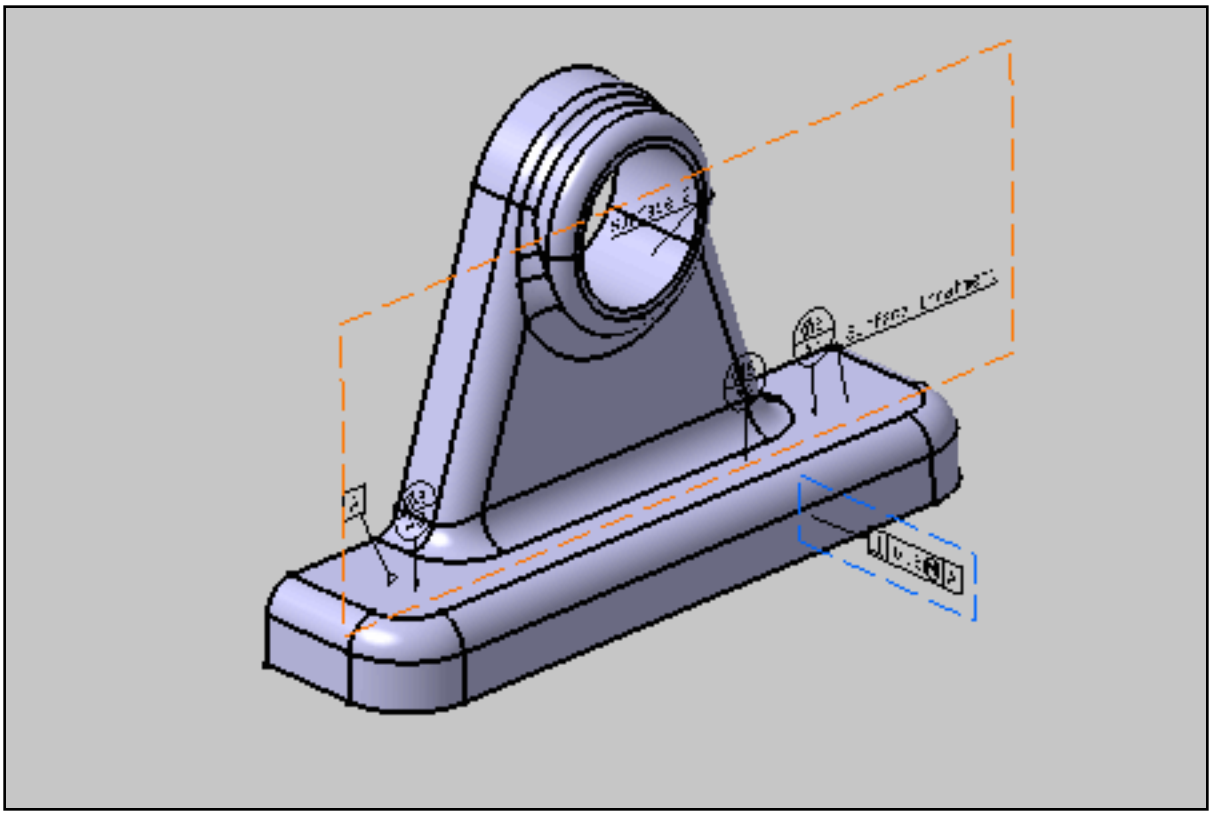
Before we discuss the detailed instructions for using the Functional Tolerancing & Annotations or the Product Functional Tolerancing & Annotations workbench, the following scenario aims at giving you a feel for what you can do.
You just need to follow the instructions as you progress.

The Getting Started section is composed of the following tasks:

Entering the 3D Functional Tolerancing & Annotation Workbench
Choosing the Standard
Creating Annotation Texts
Creating Simple Datum
Creating Datum Targets
Creating a Geometrical Tolerance
Filtering Annotations
Disabling 3D Annotations
Accessing the Set Properties

This scenario should take about 15 minutes to complete.

Eventually, the toleranced part will look like this:



Entering the 3D Functional Tolerancing & Annotation Workbench

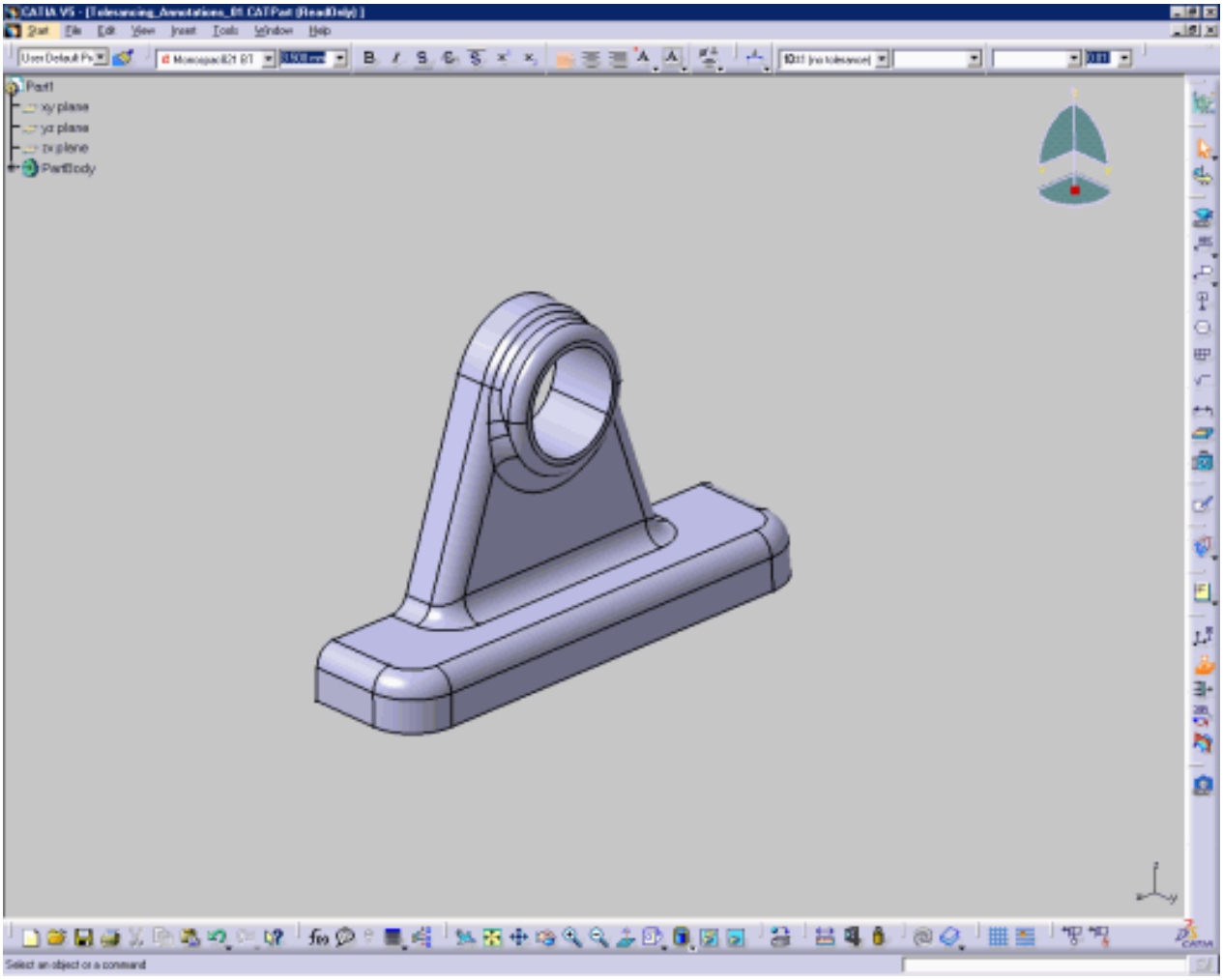
This task shows you how to enter the workbench and open the document you need for performing this tutorial.

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*.



Up Next

Choosing the Standard

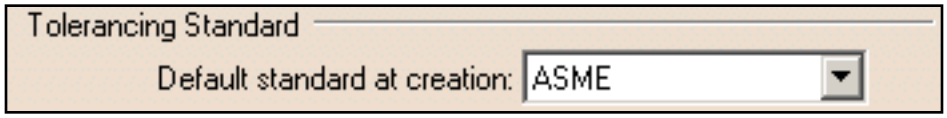
This task shows you how to set the standard you need for tolerancing your part.

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.

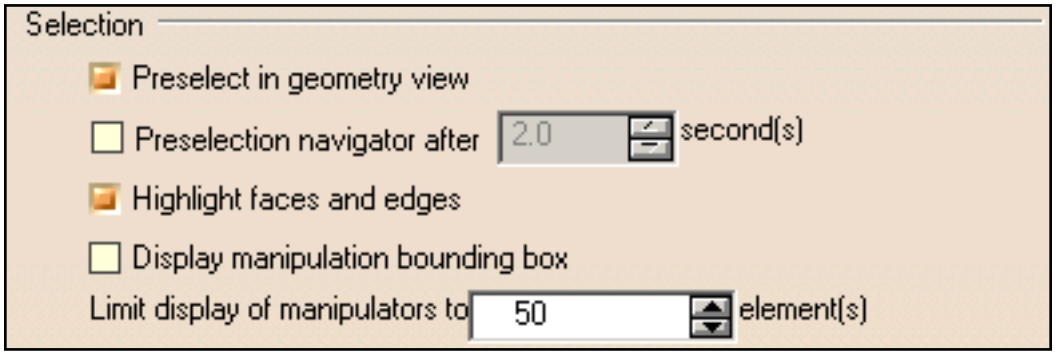
Note that this choice of standard must be expressed prior to specifying any tolerance. After any creation in the workbench, the standard may be modified but the corresponding syntax and semantic variation will not be taken into consideration.

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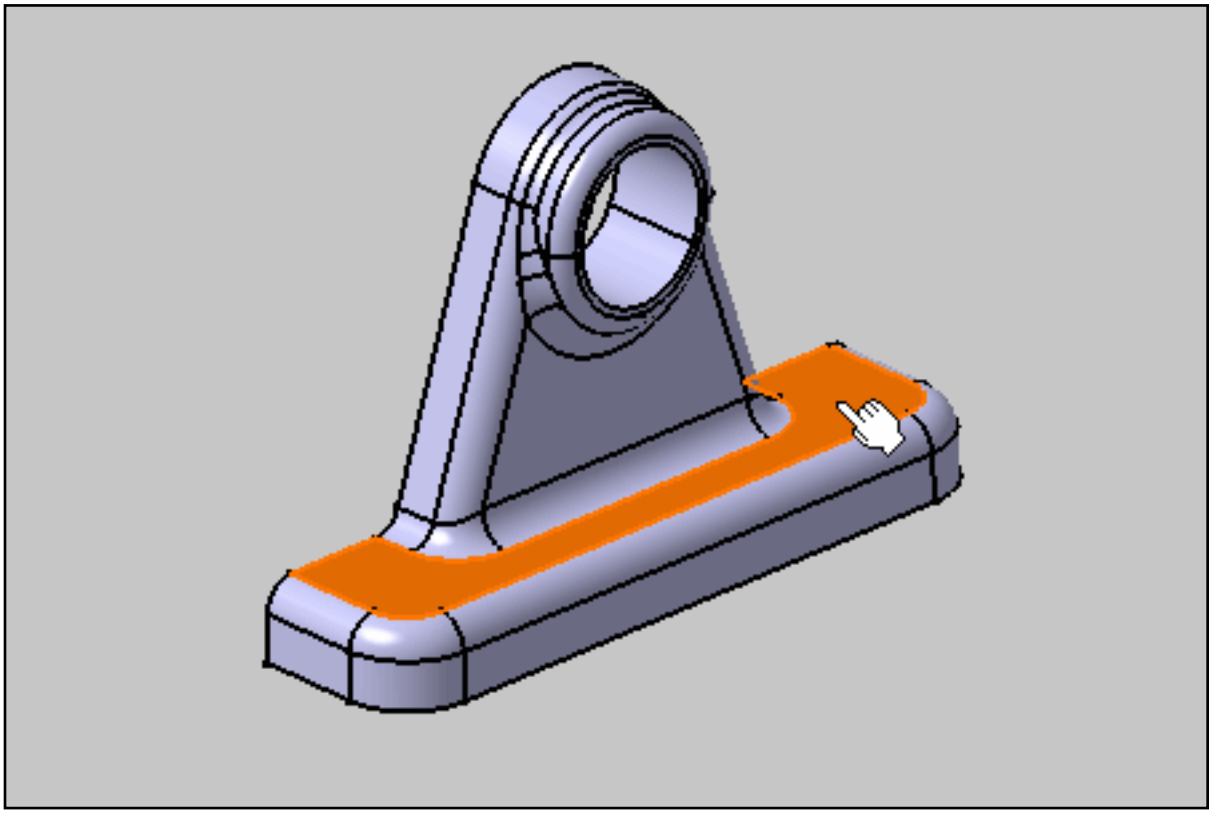
Creating Annotation Texts

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 geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

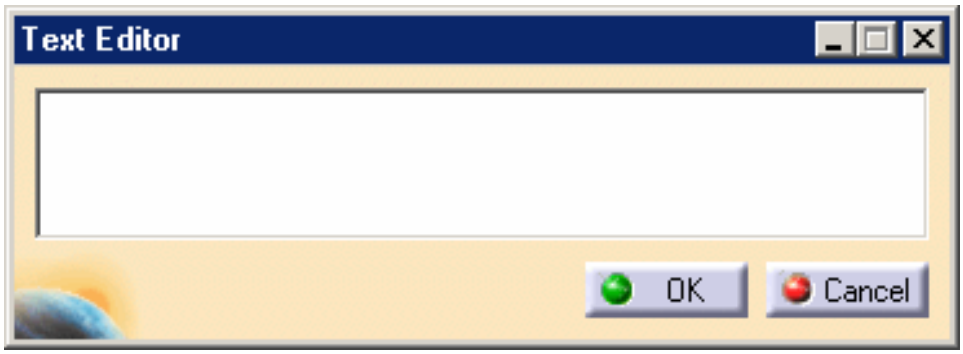


1. Click the face as shown to define the surface and the location for the arrow head of the leader line.



2. Click the **Text** icon: 

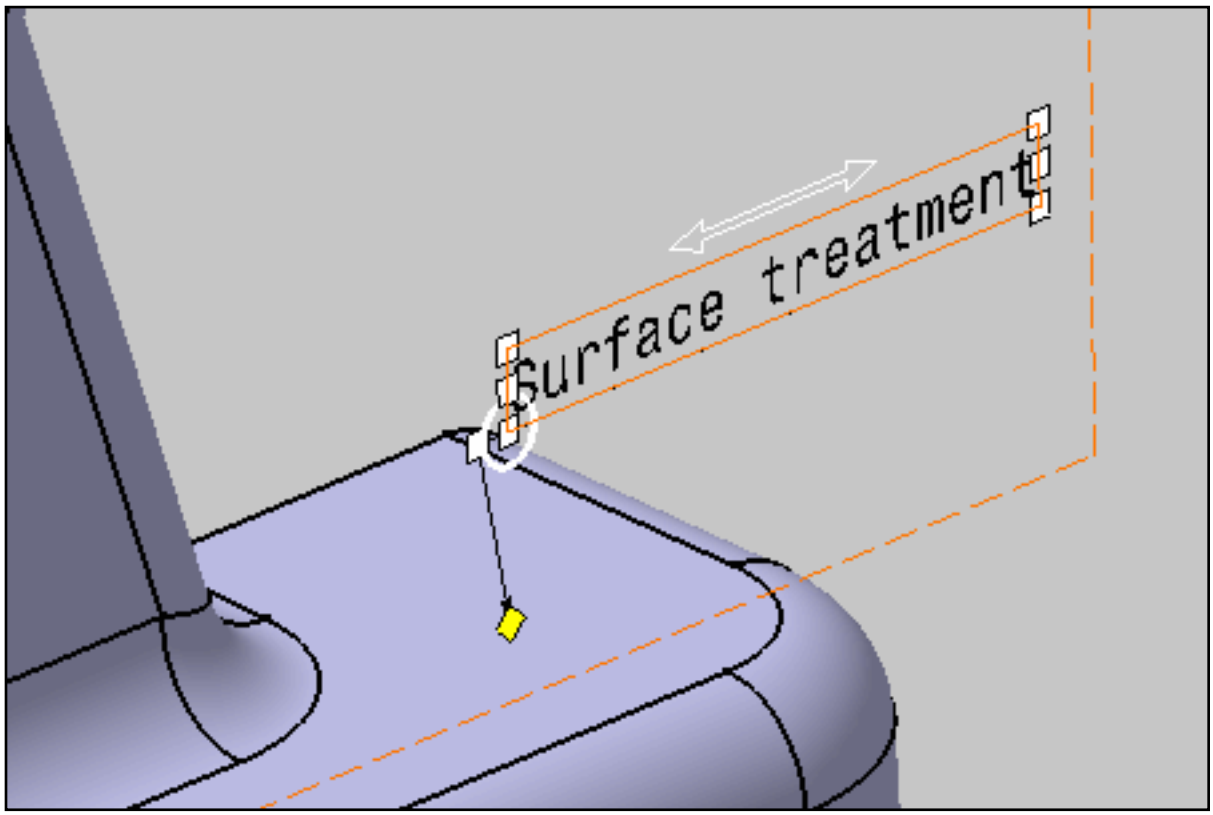
   The **Text Editor** dialog box is now displayed.



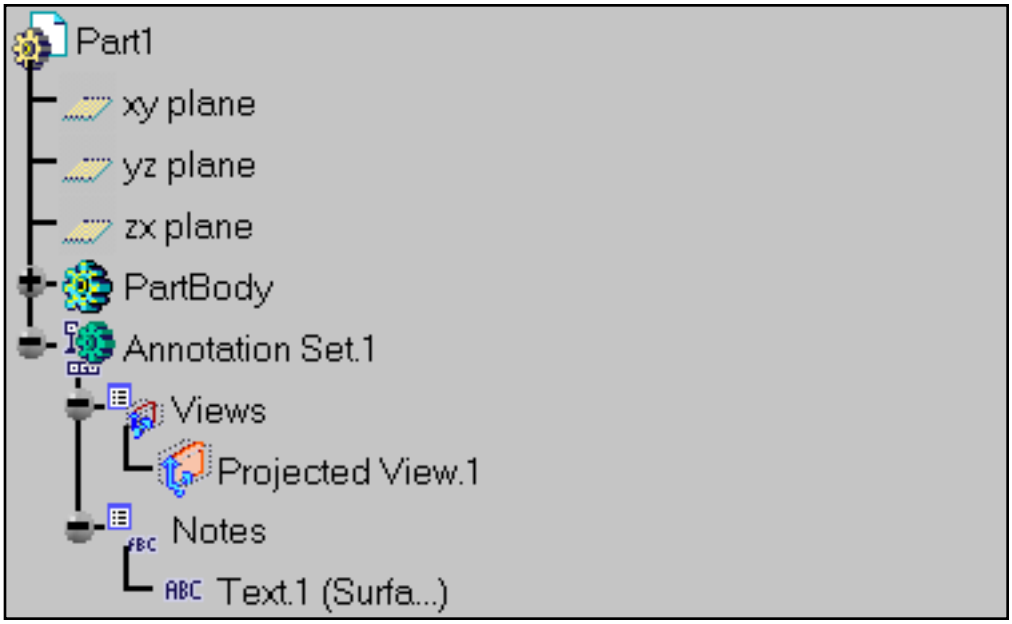
3. Enter "Surface treatment" in the dialog box.

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

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

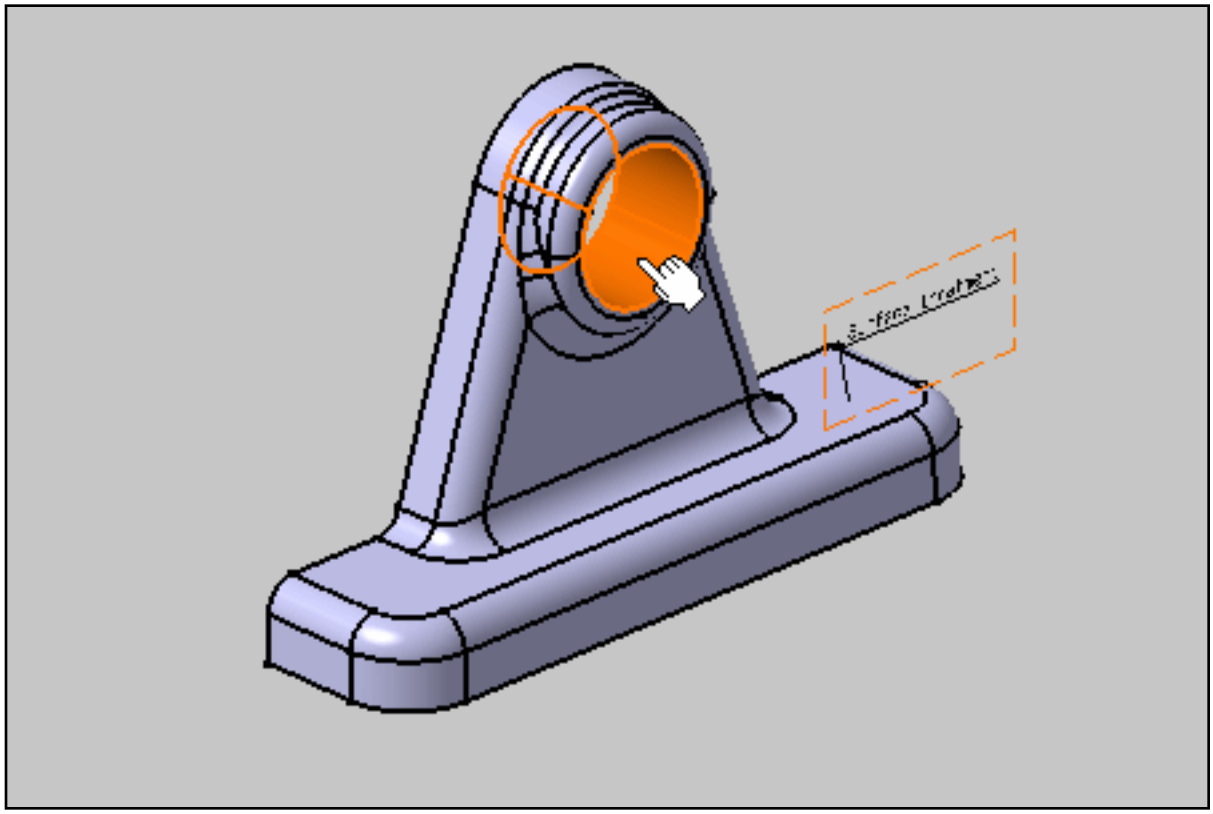


The textual annotation identified as Text.1 is added to the specification tree. Note also that the application creates a projected view as shown in the tree. This view is automatically created when any view have been created yet or any existing view cannot be re-use.



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.

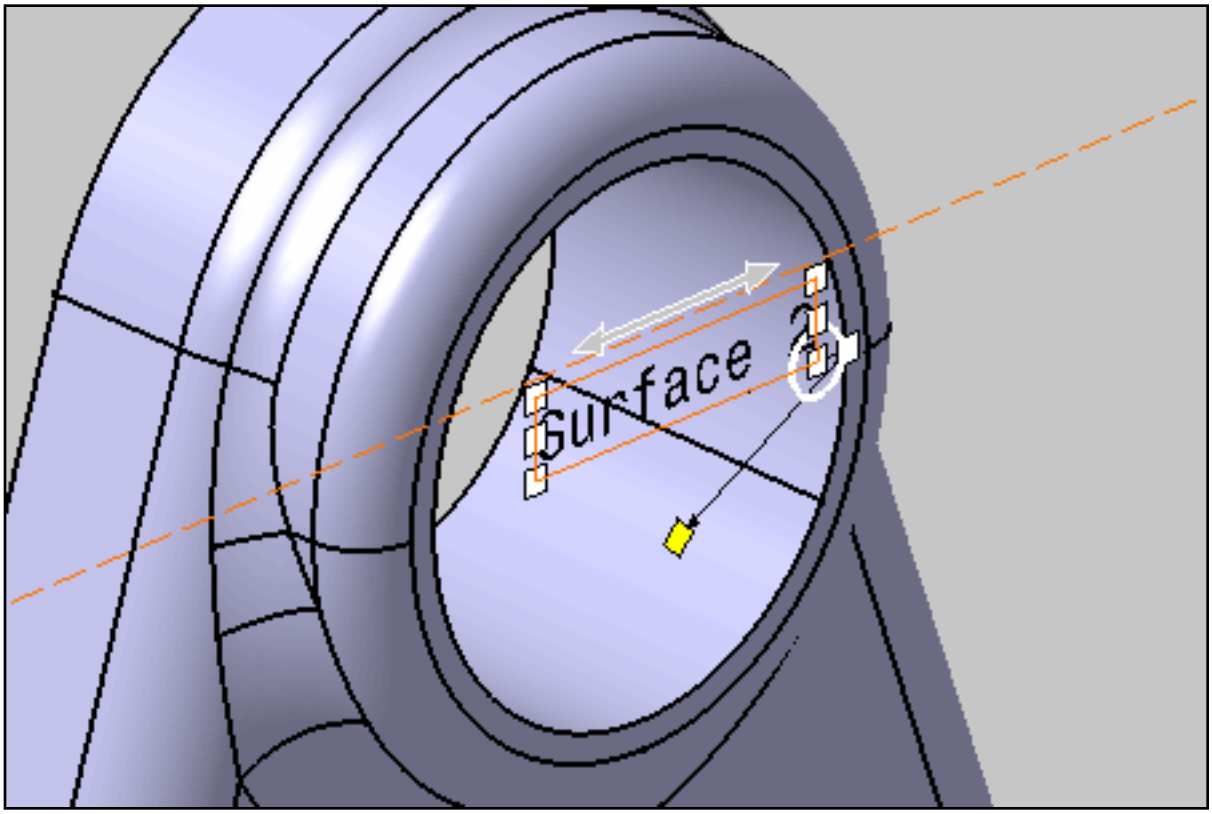
5. To create another text annotation, select the inner cylindrical face:



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.

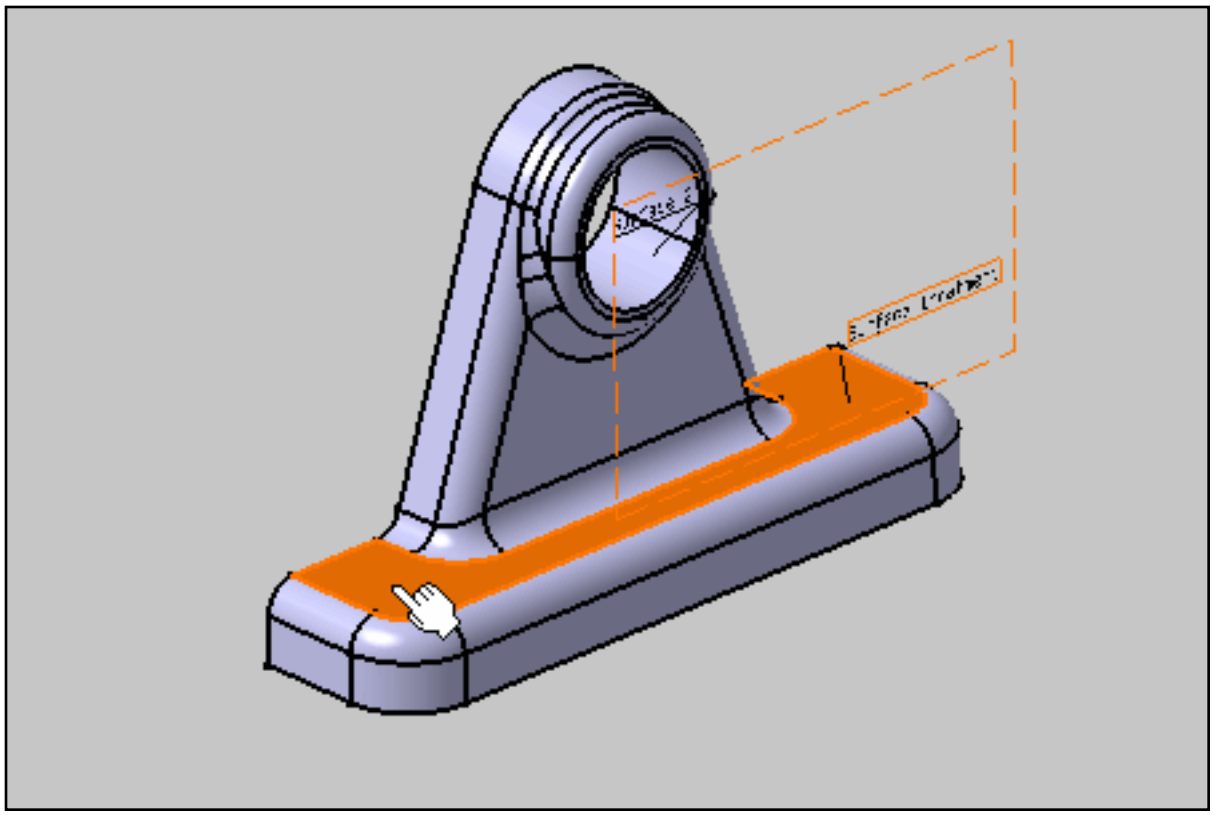
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Creating Simple Datum

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

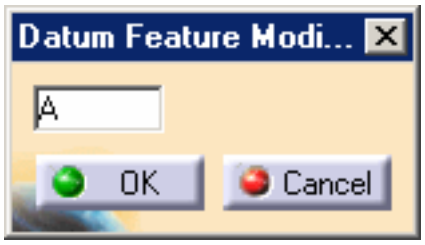
Datum elements are involved in geometric tolerancing specifications. For instance, when specifying an orientation or a position tolerance, you need to refer to datum elements for the specification.

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



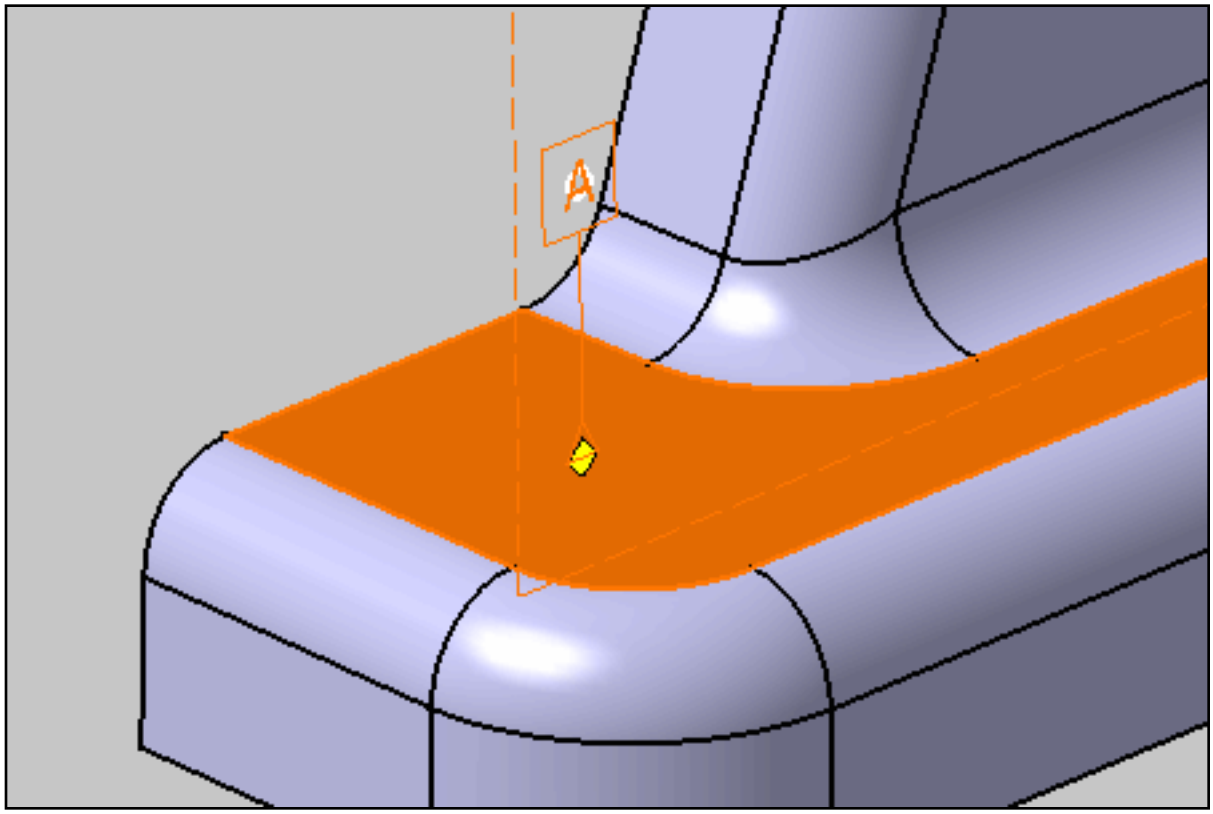
2. Click the **Datum Feature** icon: 

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



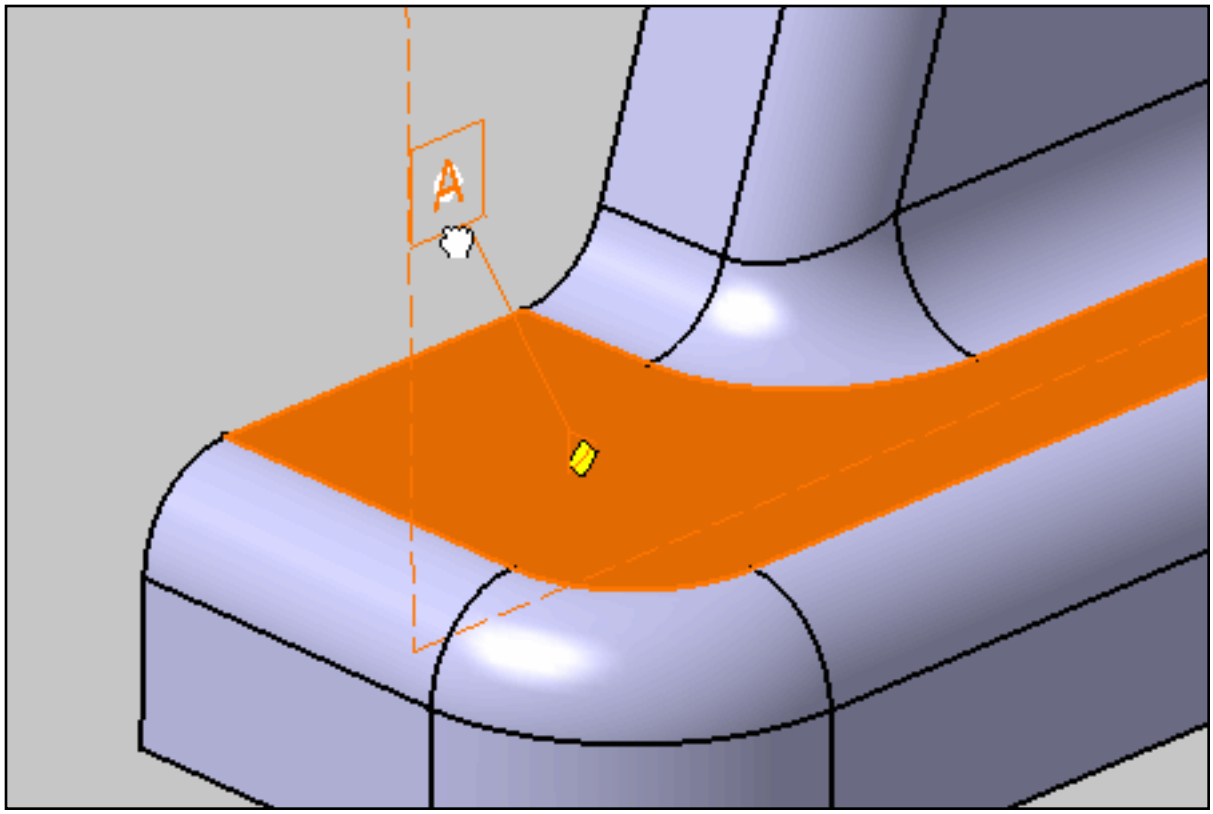
3. Click **OK** to create the datum if the identifier corresponds to your choice.

   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.

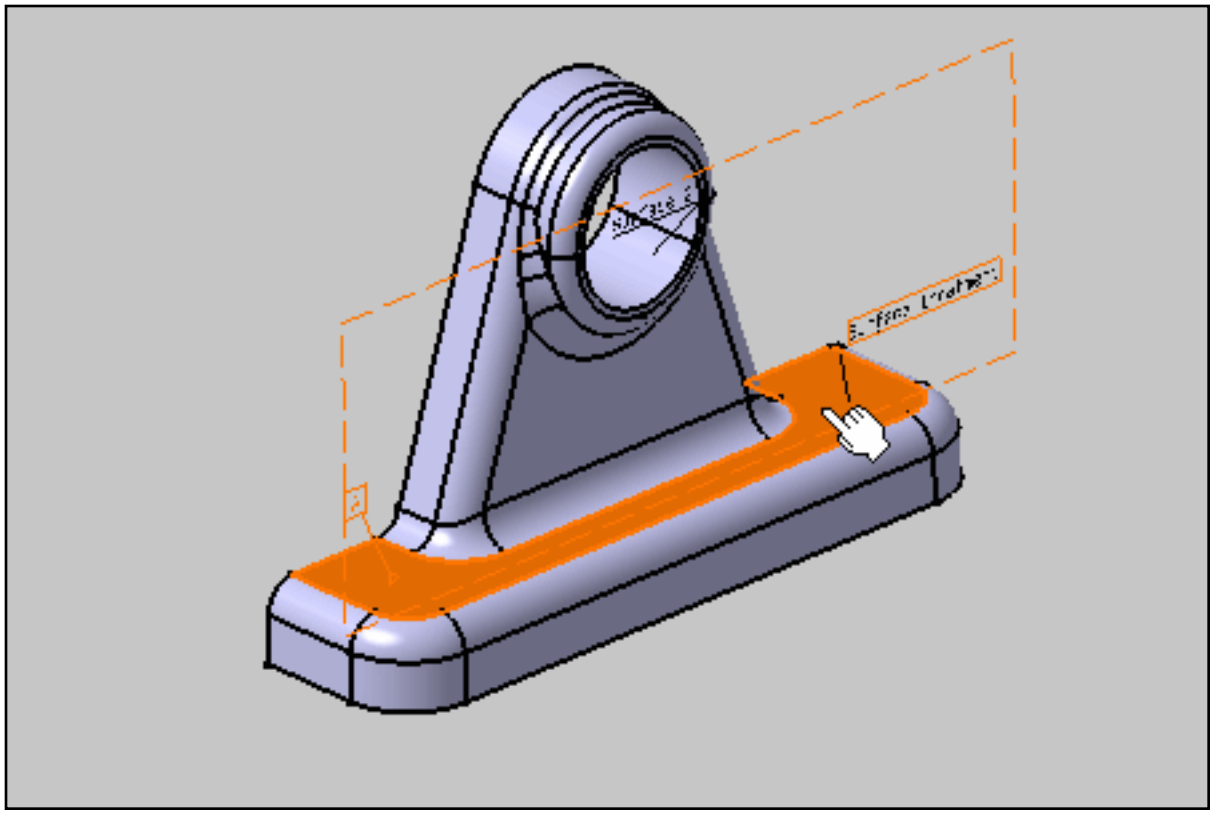


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Creating Datum Targets

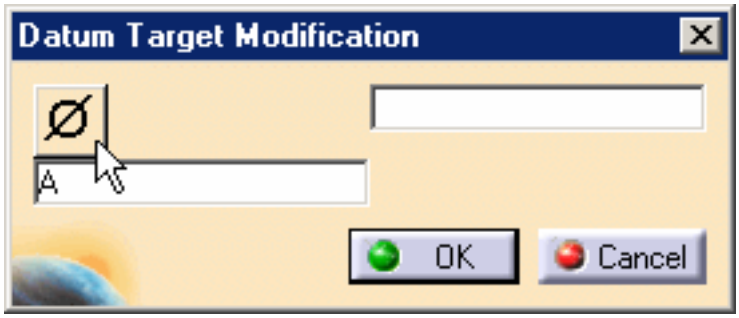
This task shows you how to create three datum targets.

1. Select the face as shown.

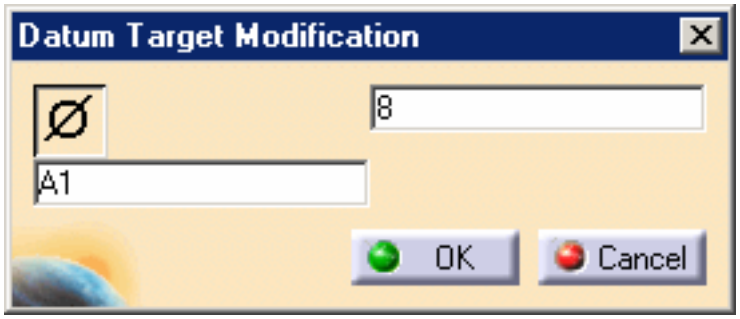


2. Click the **Datum Target** icon: 

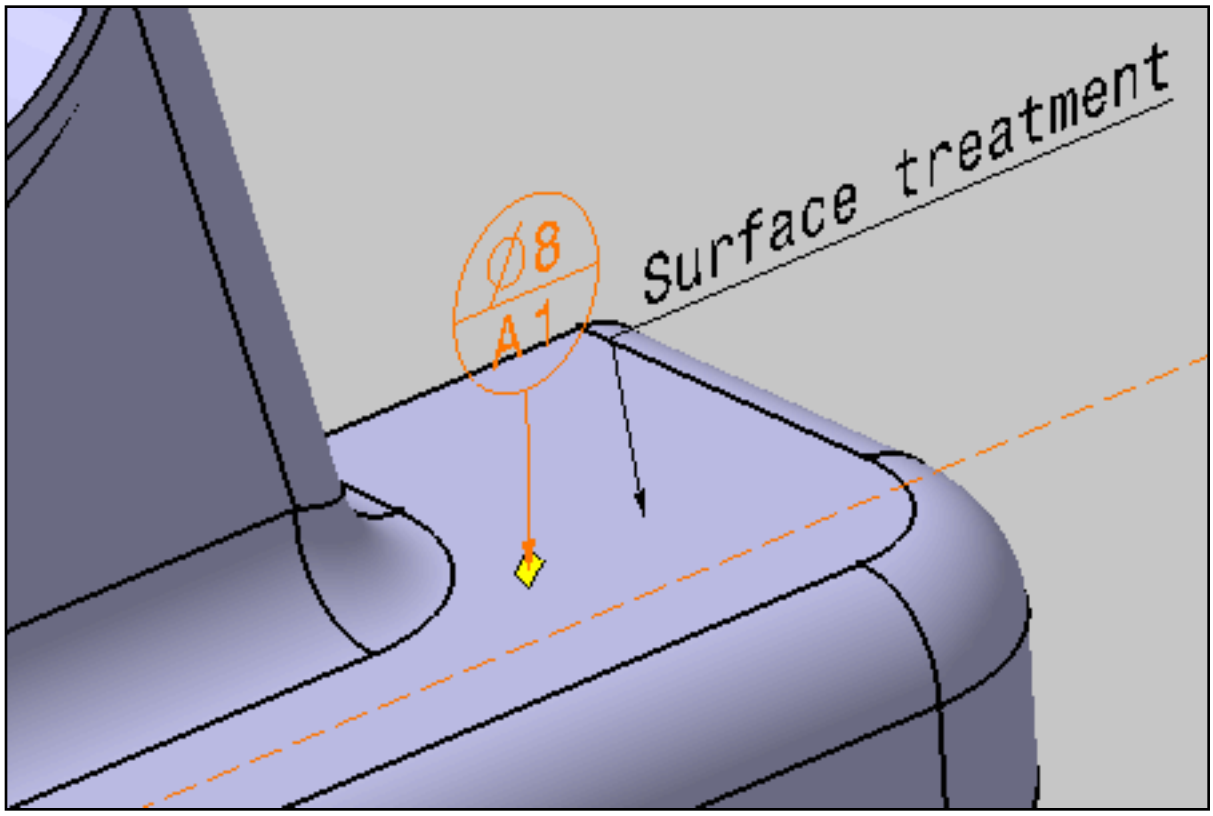
3. In the dialog box that appears, click the diameter icon.



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

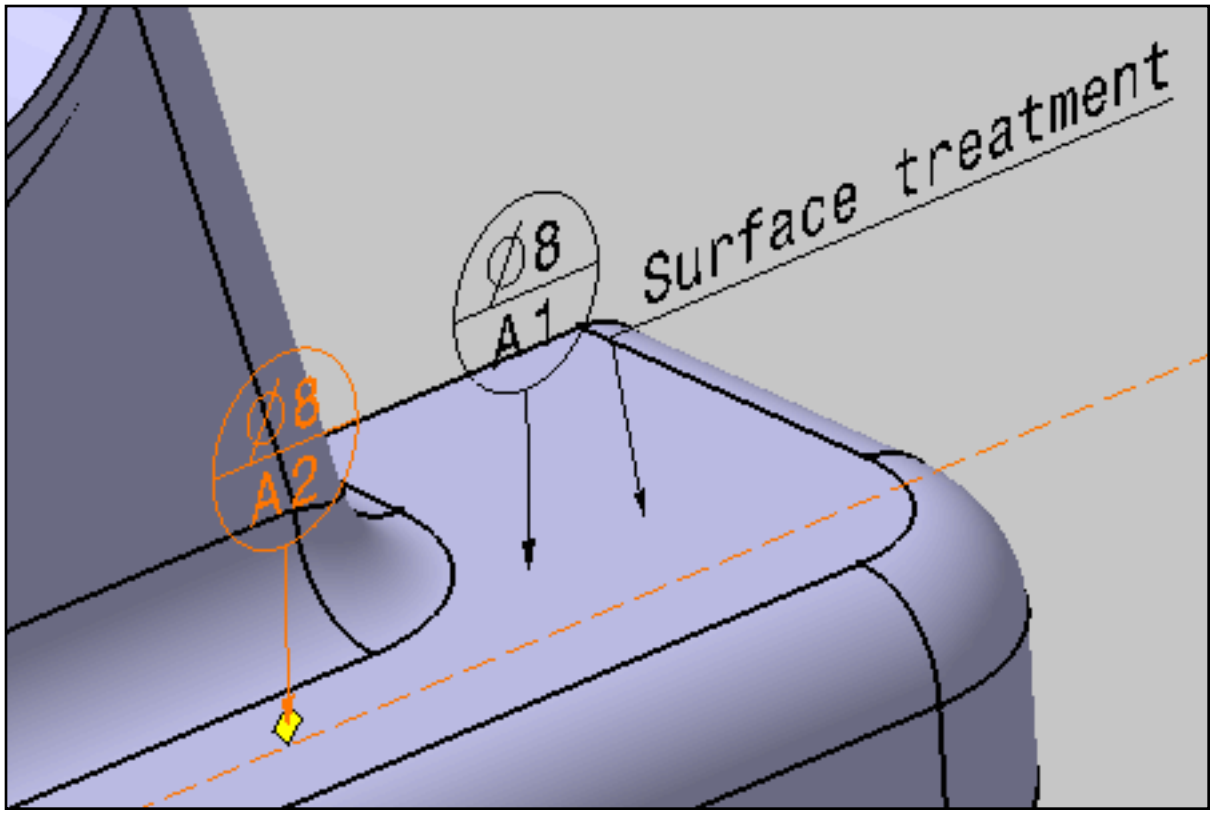


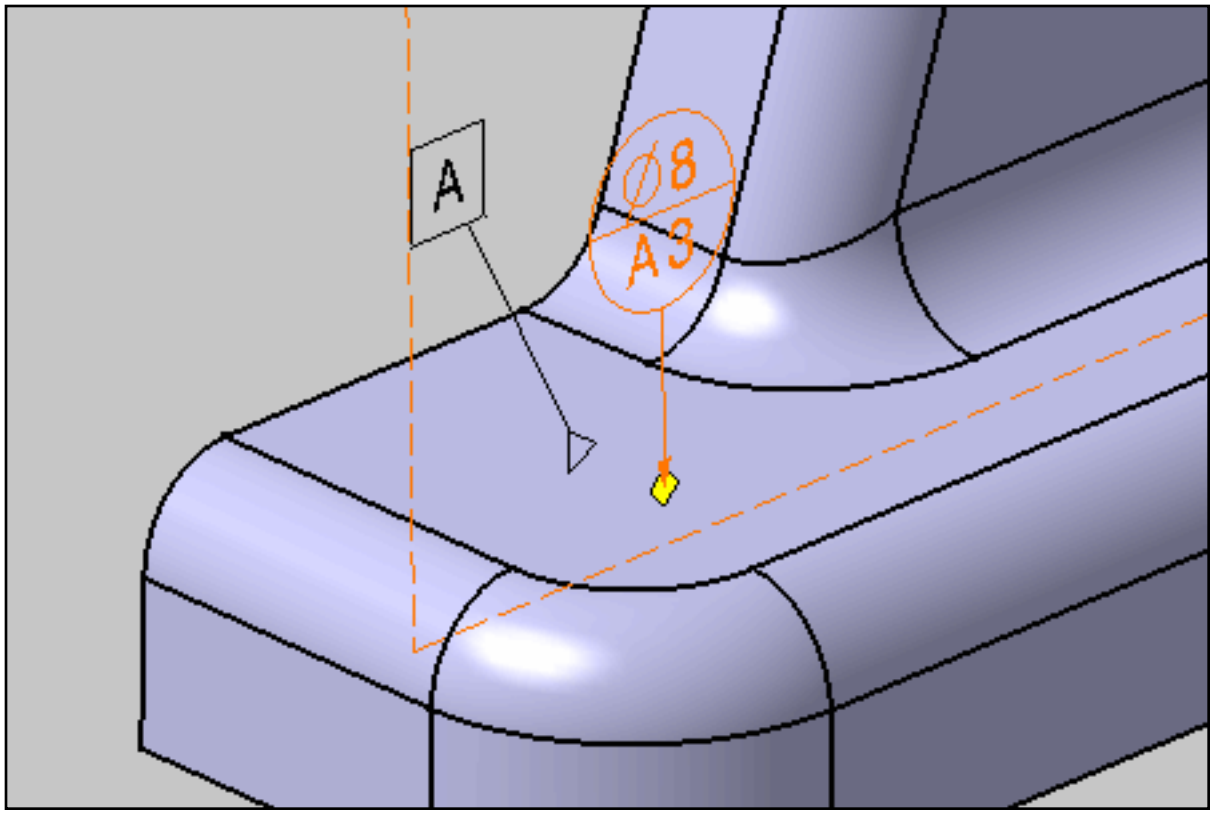
5. Click OK to validate.



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.



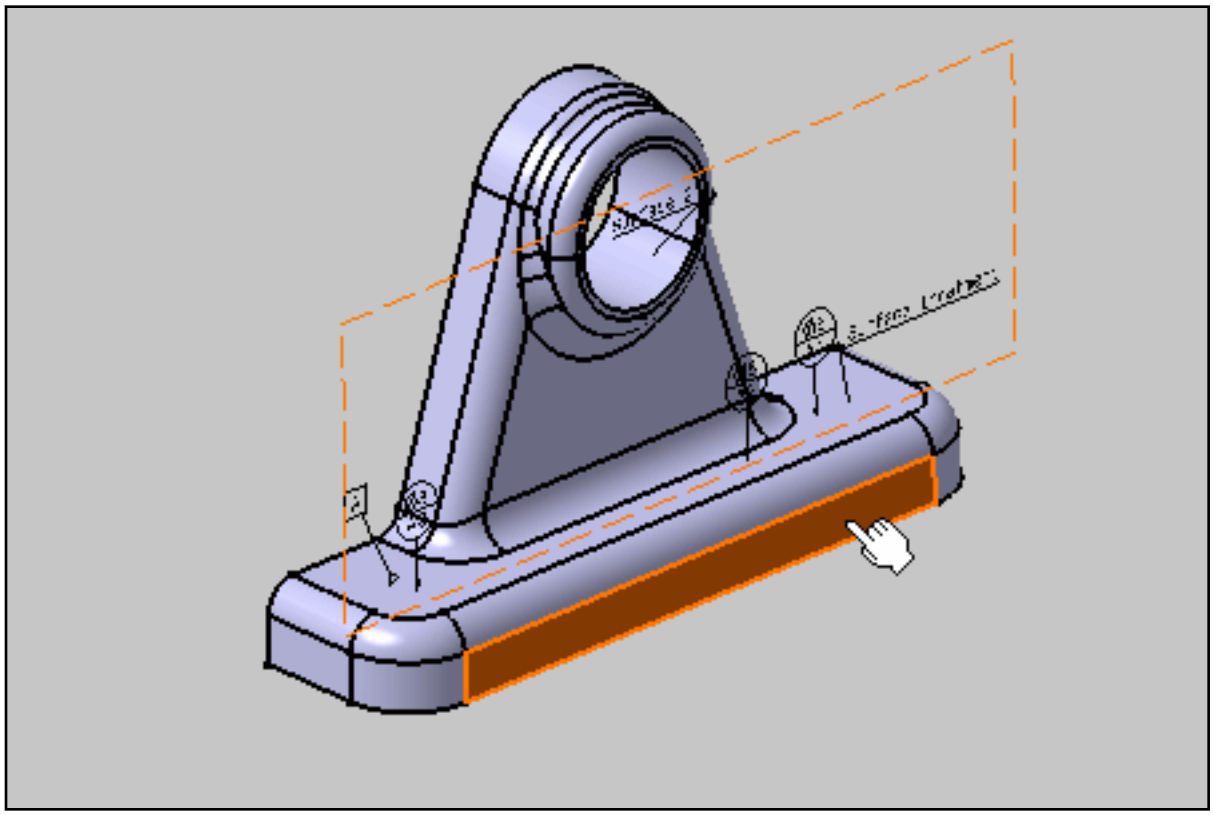


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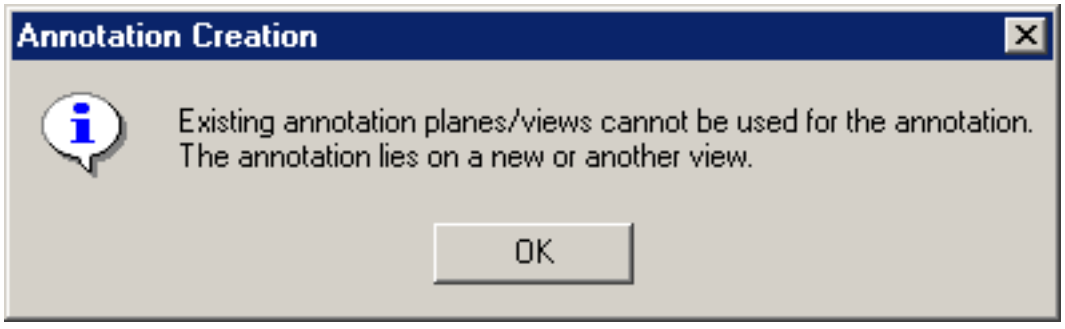
Creating a Geometrical Tolerance

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:

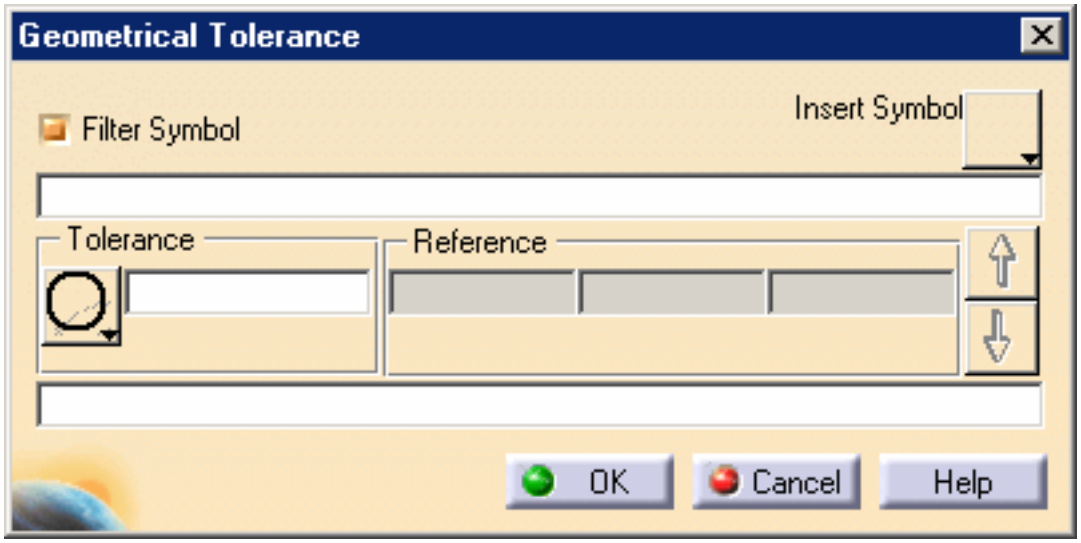


2. Click the **Geometrical Tolerance** icon: 



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.

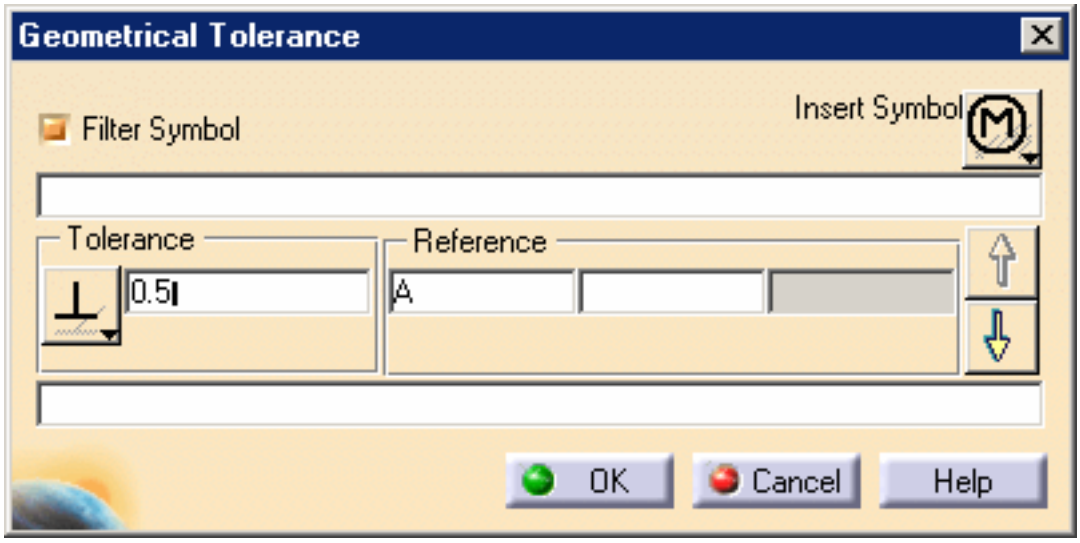
To create a geometrical tolerance you need to specify one geometric tolerancing symbol and fill in the tolerance value field.

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.

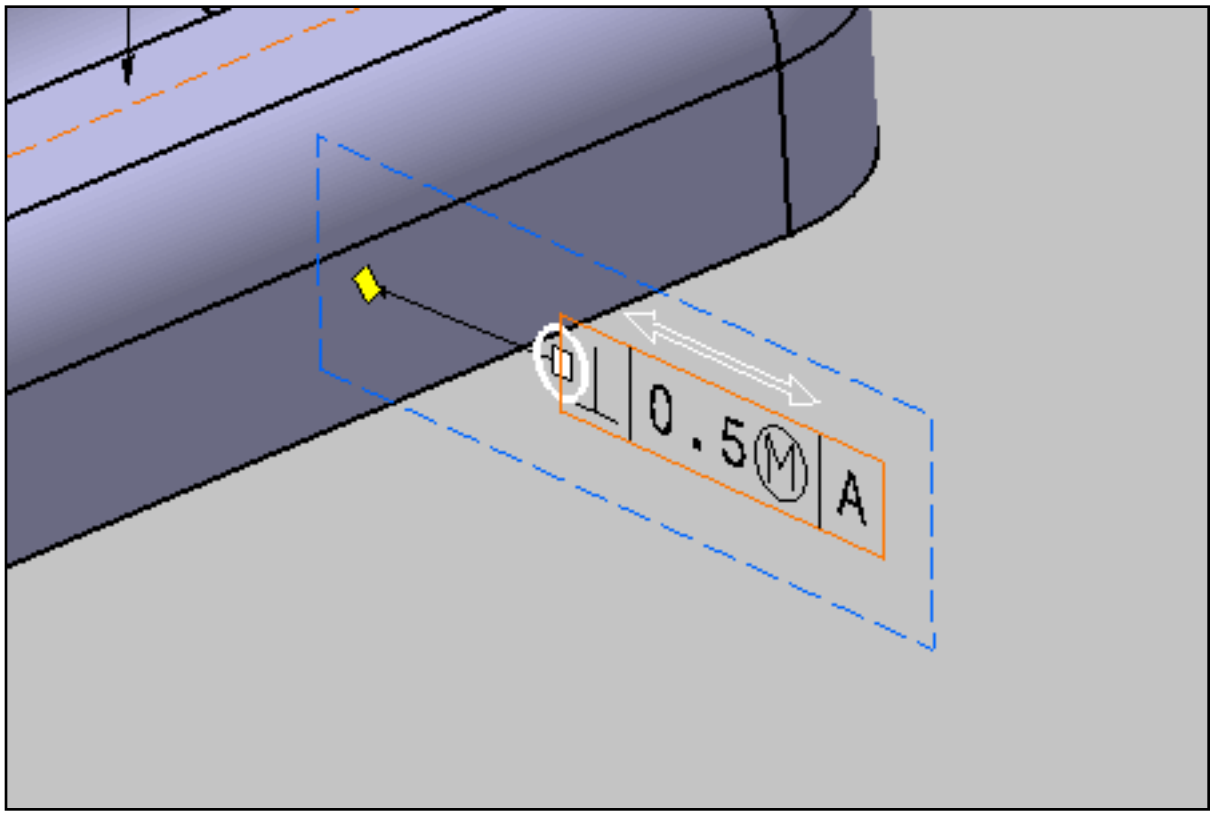


Modifiers are not displaying in tolerance and reference fields and appear with a "|" character.

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.





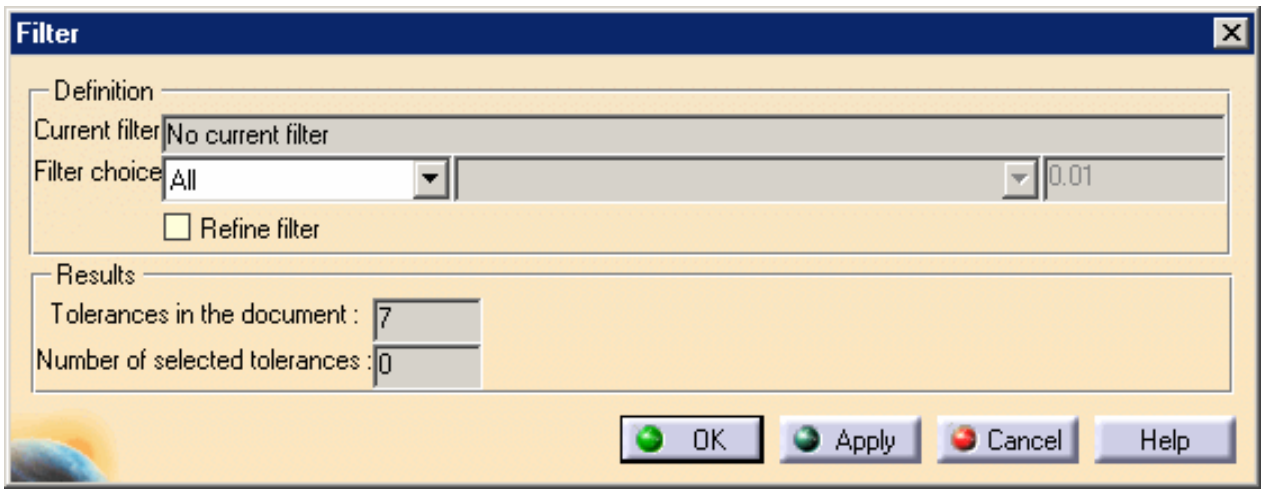
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Filtering Annotations

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.



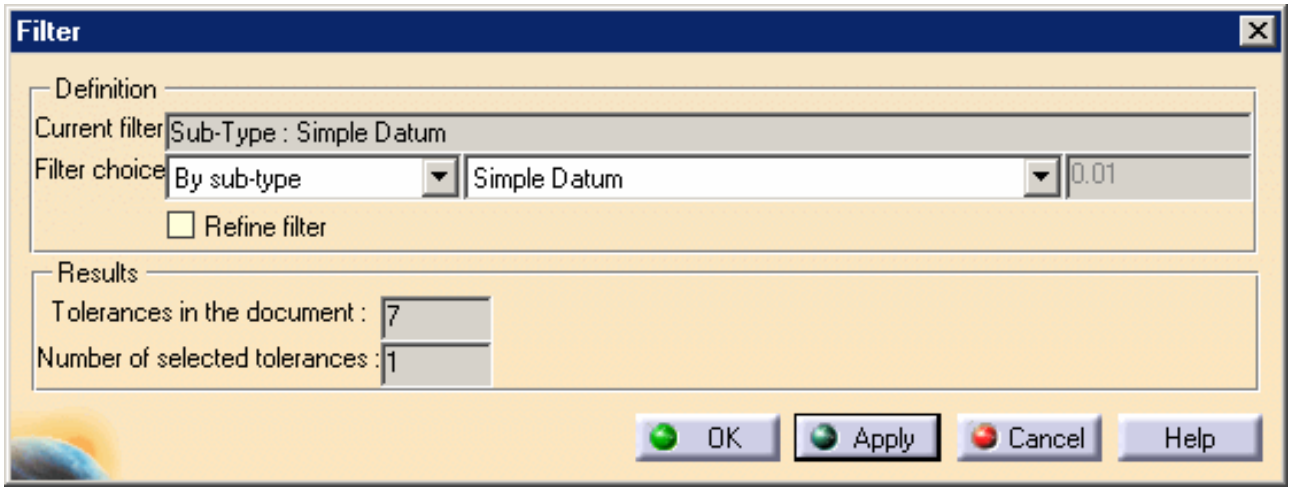
2. Set the **Filter** choice field to "By sub-type".

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

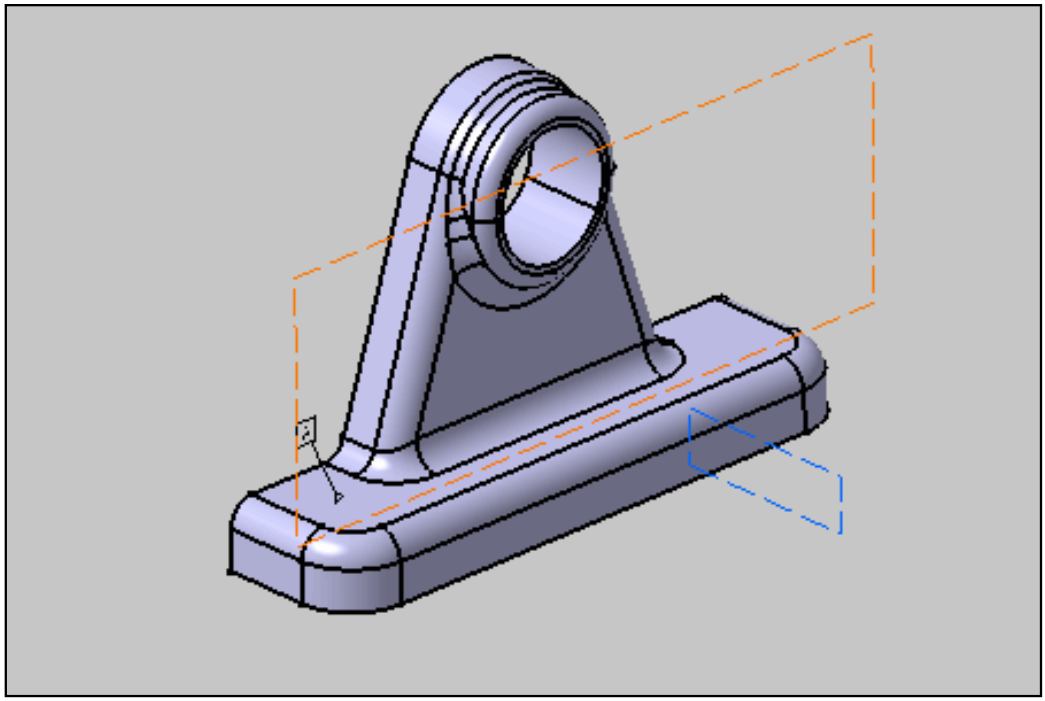
   Checking the **Refine filter** option filters out tolerances still filtered.

4. Click **Apply**.

   The Number of selected tolerances field displays "1".

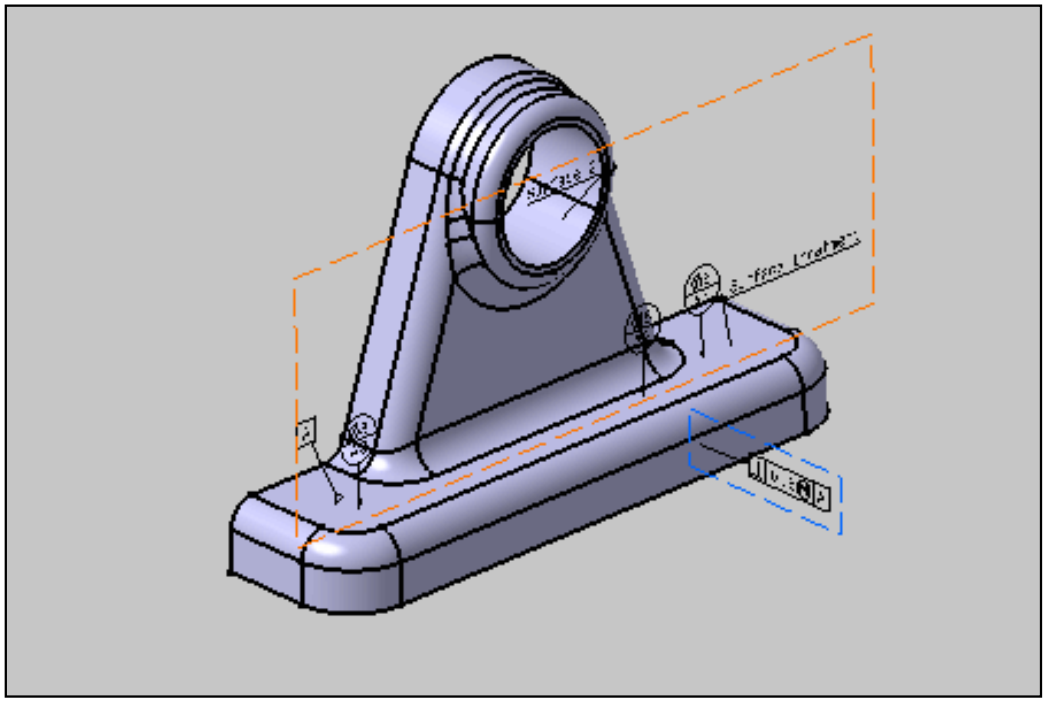


The only datum you created in this tutorial is consequently displayed attached to the geometry.



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

All annotations are visible again.



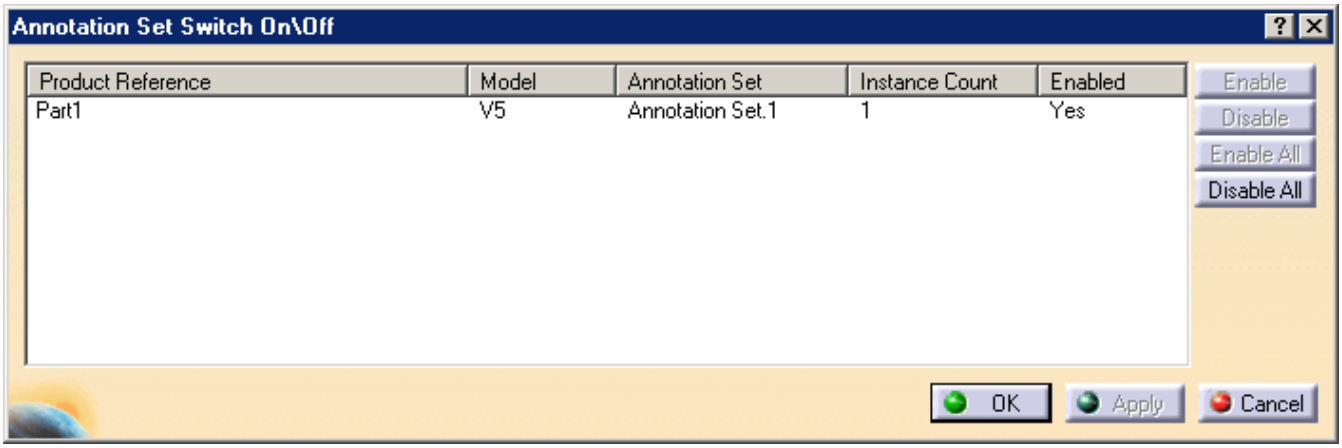
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Disabling 3D Annotations

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

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

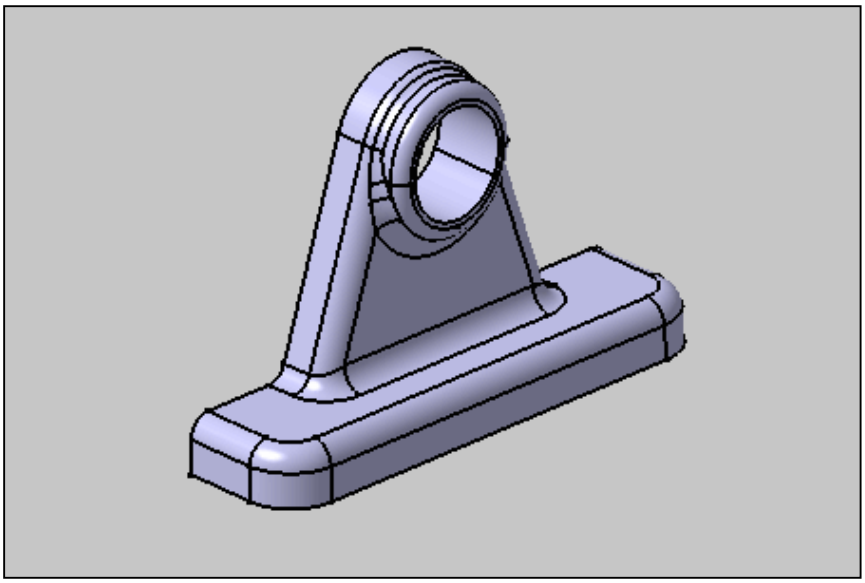
   The **Annotation Set Switch On/Off** dialog box is displayed.



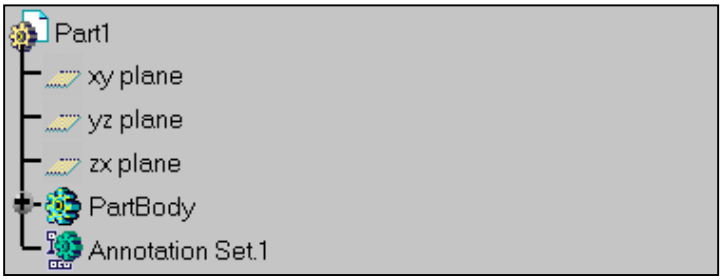
2. Click **Part1** then the **Disable All** button.

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.

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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.

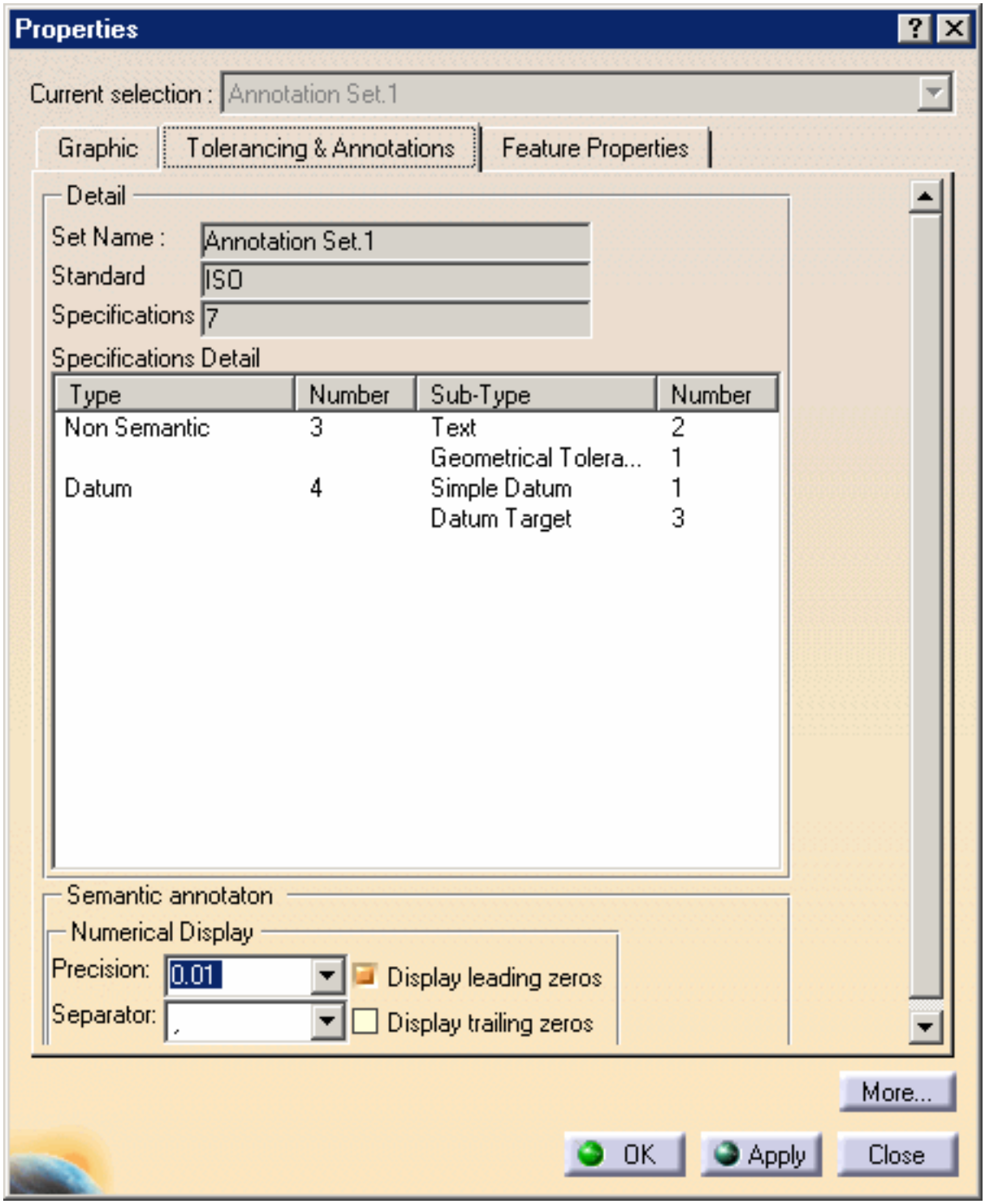
The dialog box that appears displays information about the set, namely:

The selected set name: as displayed in the specification tree

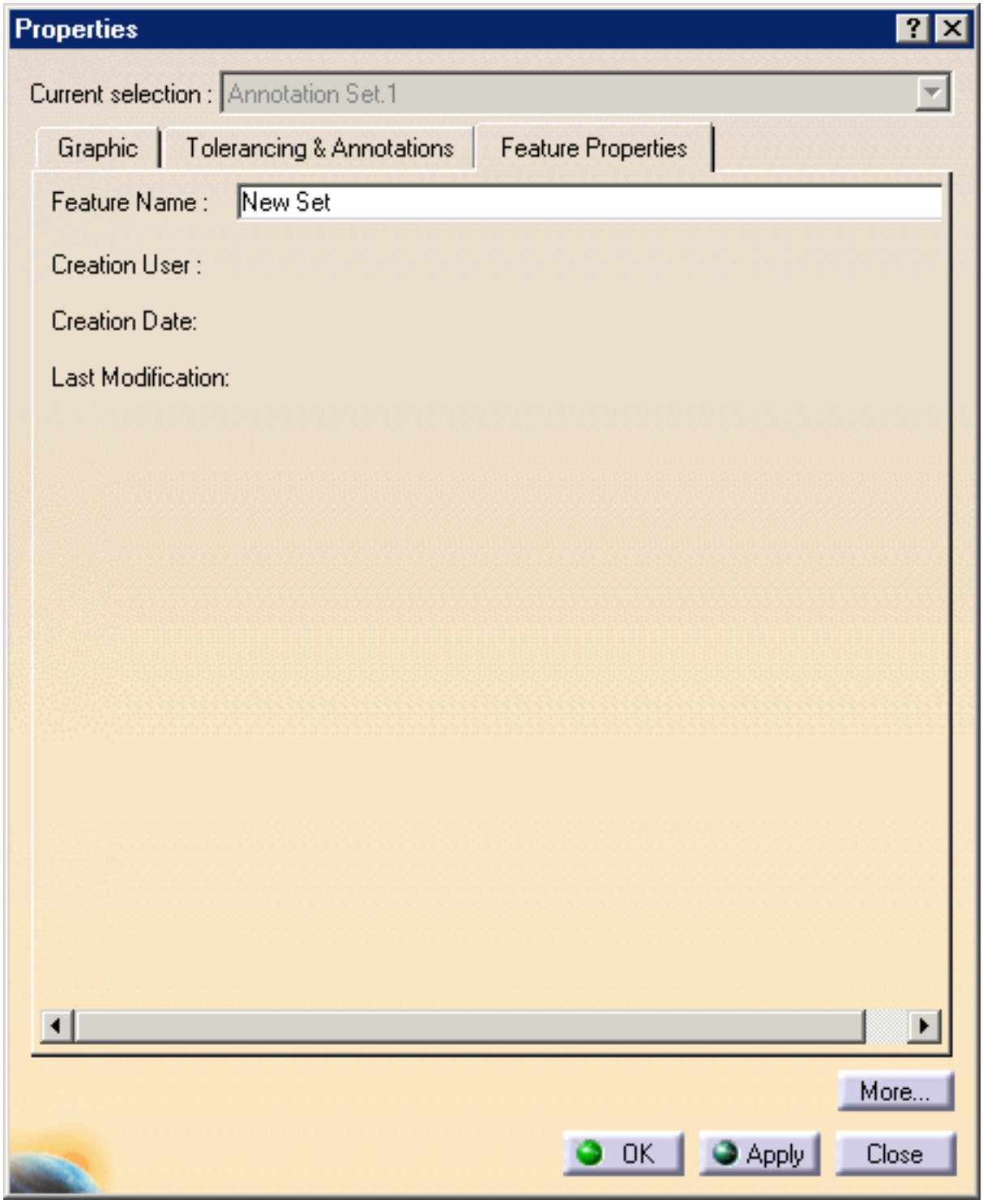
The standard used: ANSI

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.

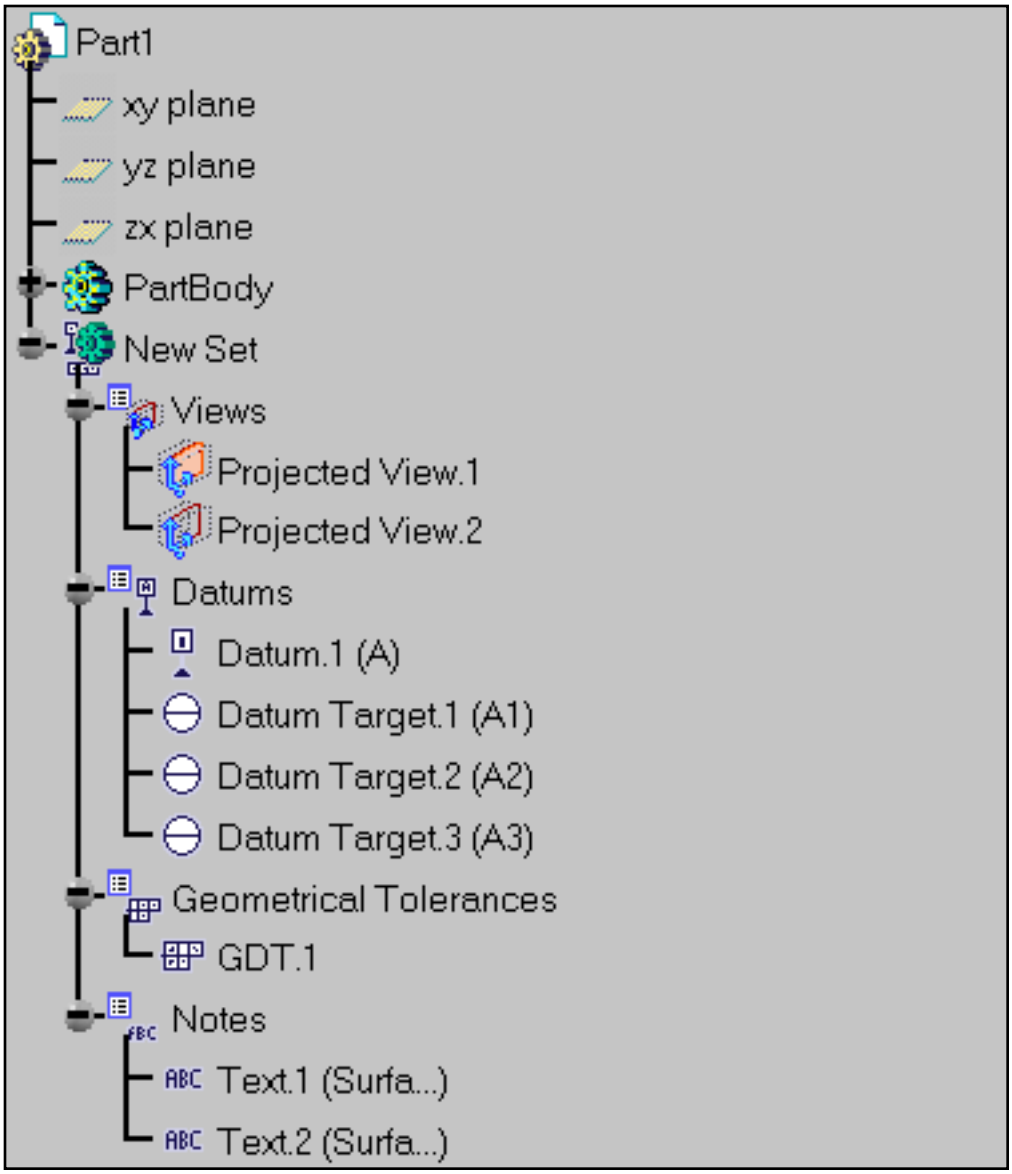


4. Click the **Feature Properties** tab (if not available, click **More**) and enter a new name in the **Feature Name** field.
   For instance, enter **New Set**.



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...)

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Basic Tasks

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

Choosing a Standard
Using the Tolerancing Advisor
Tolerancing Body in White
Creating Annotations
Managing Annotations
Managing Annotation Leaders
Managing Graphical Properties
Filtering Annotations
View/Annotation Planes

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 four conventional standards:

   ASME (American Society for Mechanical Engineers)

   ANSI (American National Standards Institute)

   ISO (International Organization for Standardization)

   JIS (Japanese Industrial Standard)

   and three CATIA-CADAM standards:

   CCDANSI (CATIA-CADAM American National Standards Institute)

   CCDISO (CATIA-CADAM International Organization for Standardization)

   CCDJIS (CATIA-CADAM 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.



Up Next

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.

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Introducing the Tolerancing Advisor

This task introduces the Tolerancing Advisor.

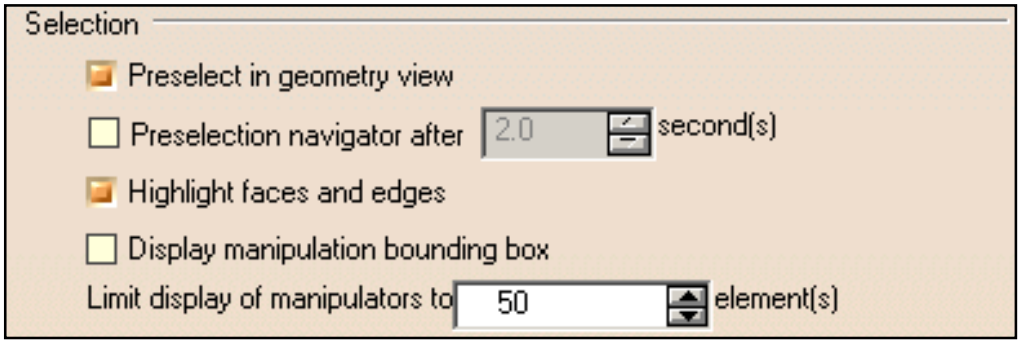
The Tolerancing Advisor lets you create allowed annotations according to the selected geometrical element or existing annotation.

Allowed annotations are:

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

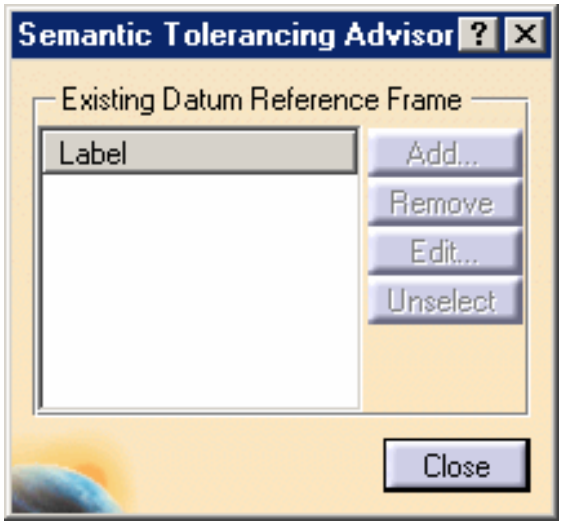
Open the Tolerancing_Annotations_01 CATPart document.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

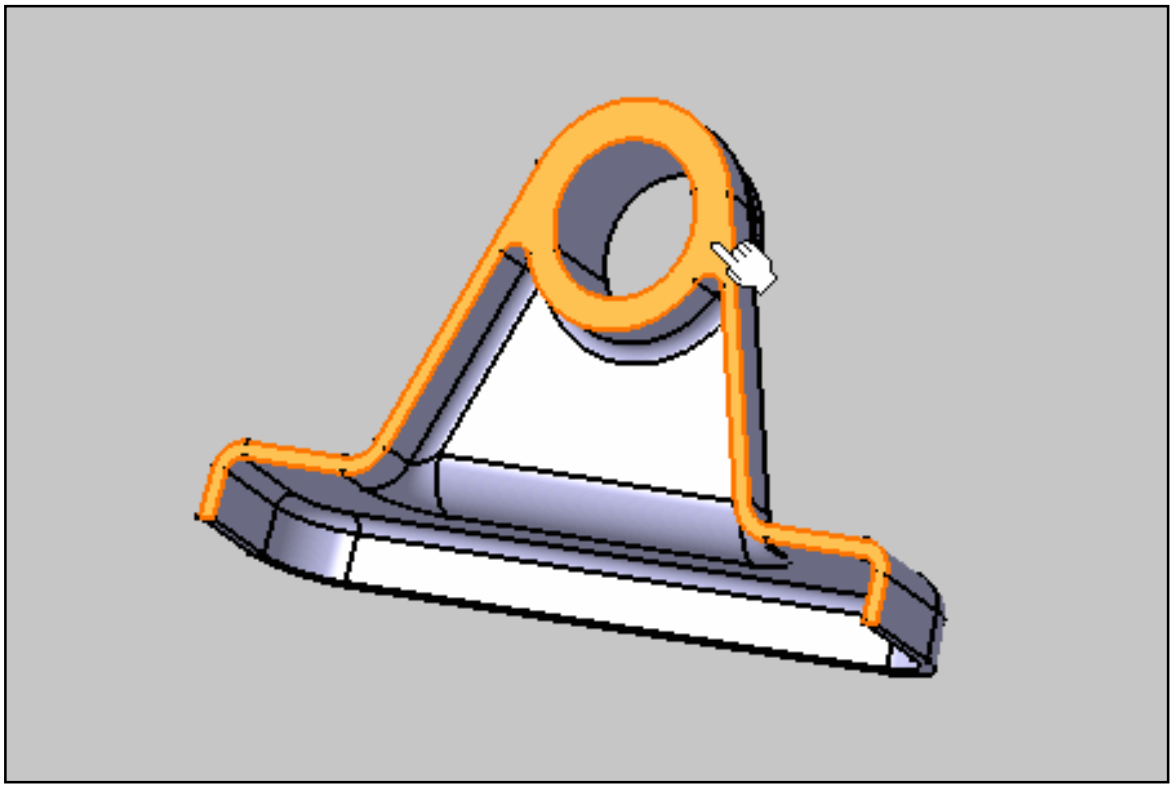


1. Click the **Tolerancing Advisor** icon: 

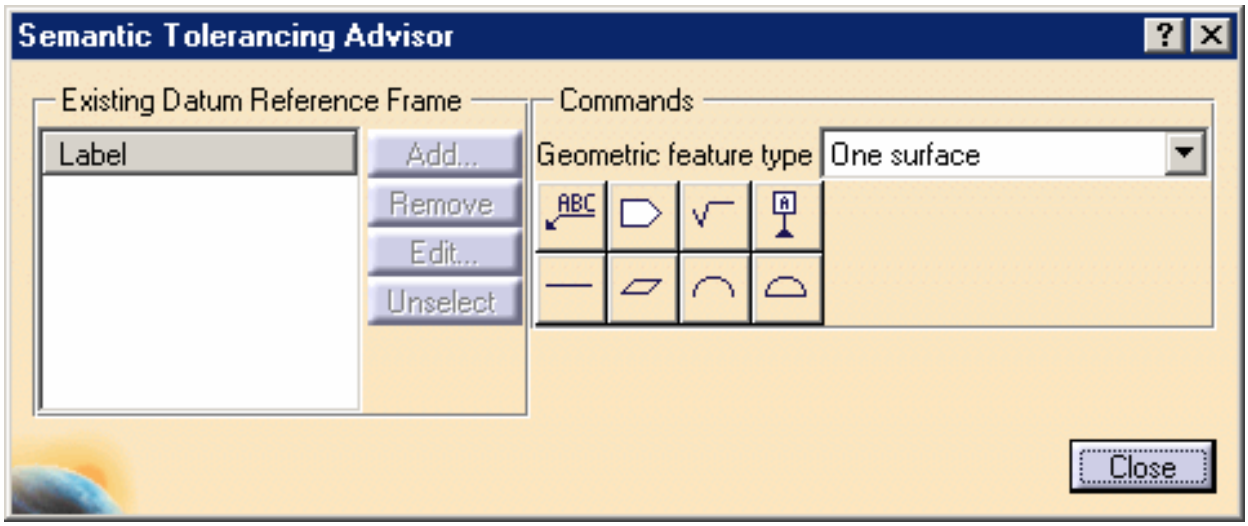
   The **Semantic Tolerancing Advisor** dialog box appears.
   This is the minimal appearance for this dialog box because no geometrical element or annotation has been selected and no datum reference frame has been created yet.



2. Select the surface as shown on the part.



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

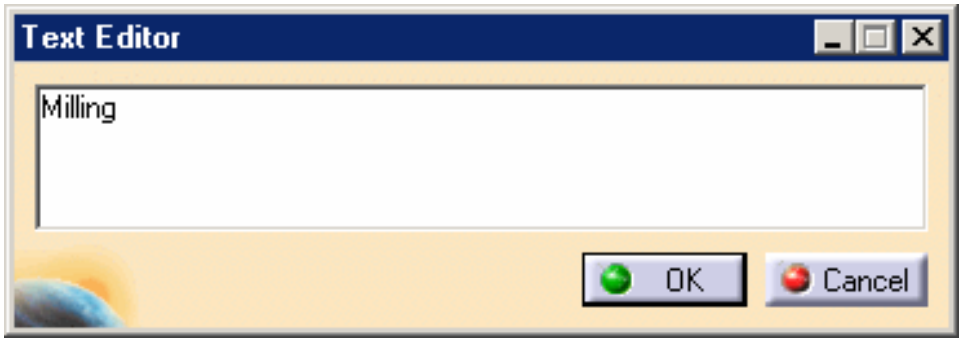


The **Commands** frame contains all the semantic annotations that will be created in relation with the selected element and the geometrical feature type.

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

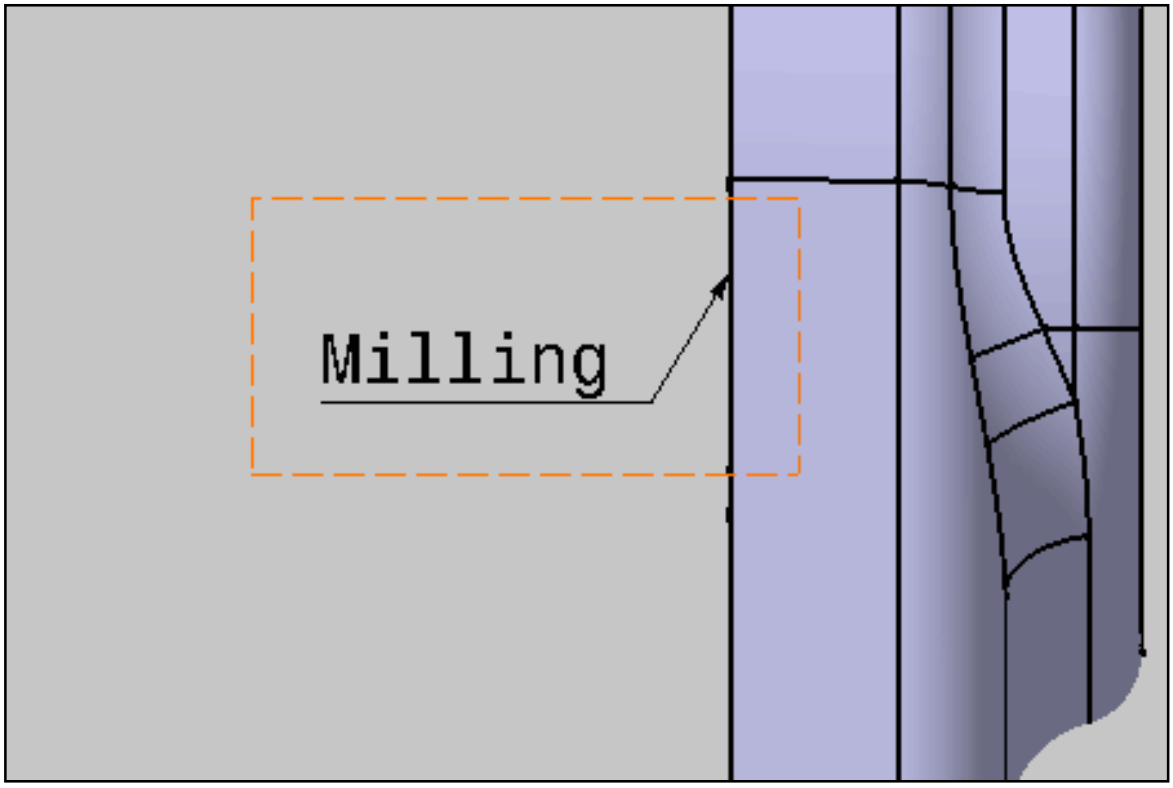
3. Click the **Text with Leader** icon (One surface): 

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

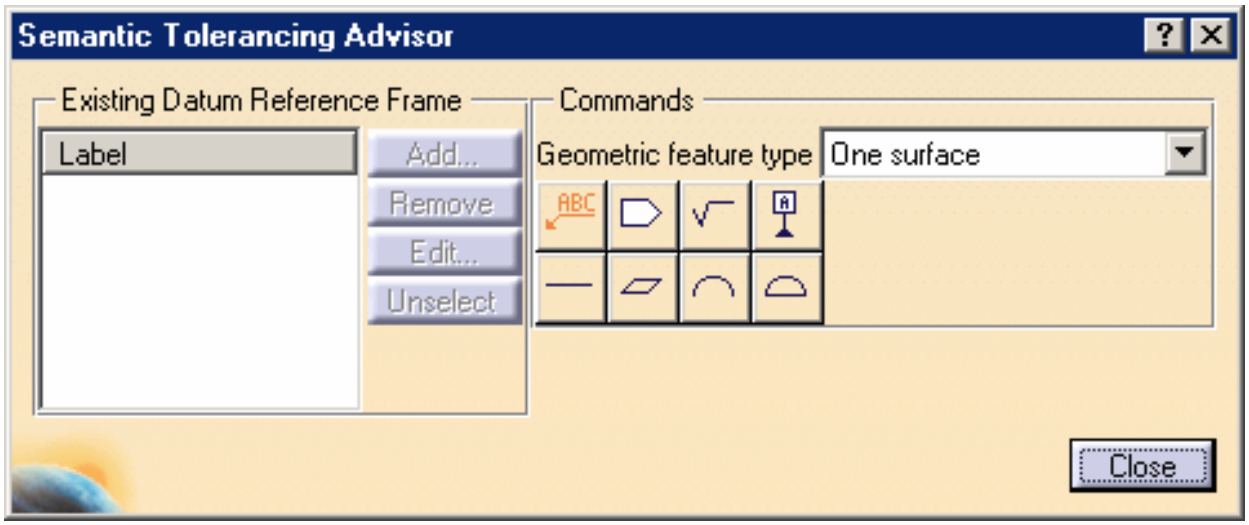


5. Click **OK** in the dialog box.

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.

Up Next

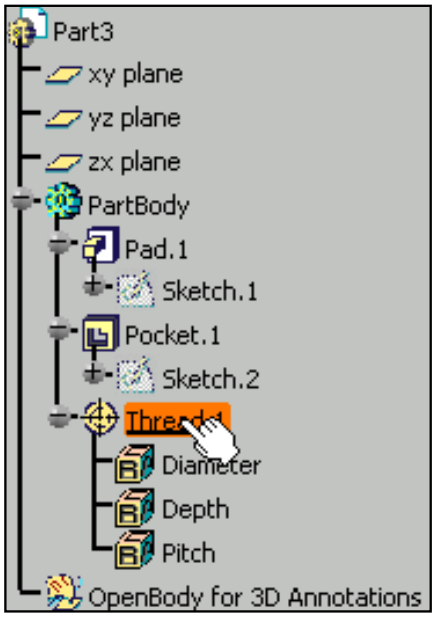
Dimensioning and Tolerancing Threads using the Tolerancing Advisor

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.

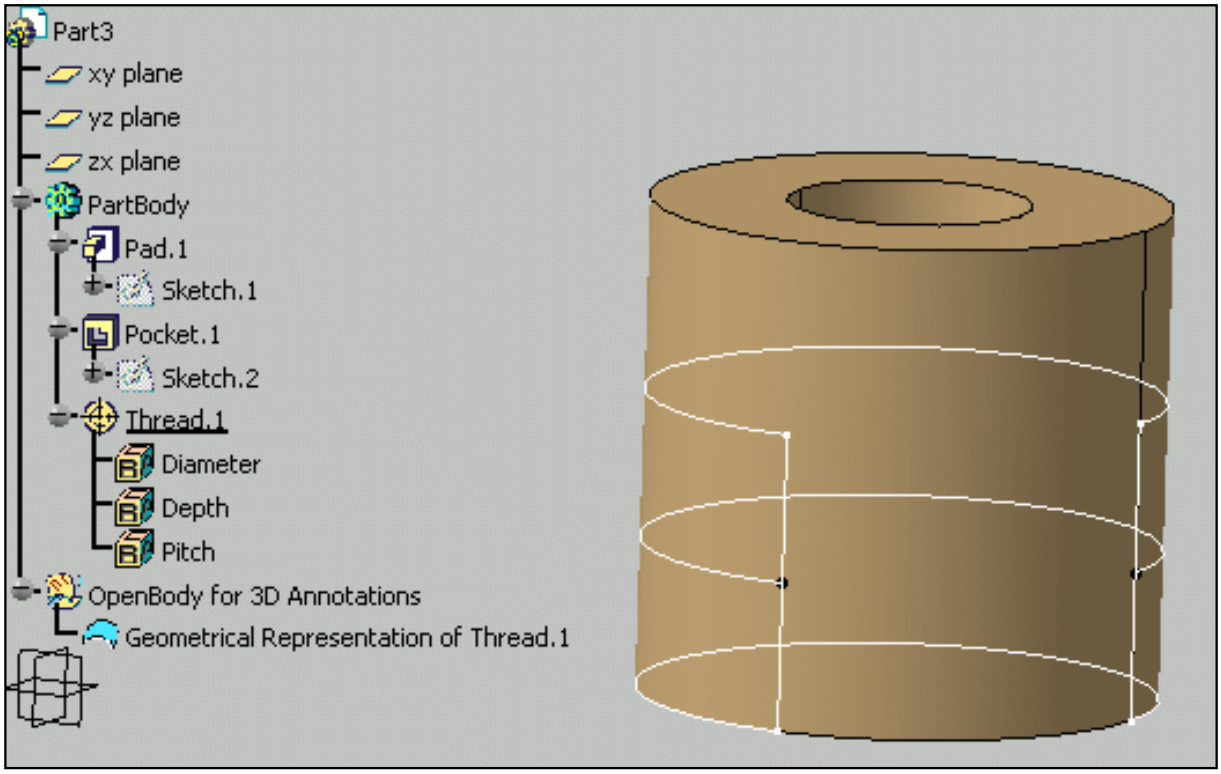


The **Thread Representation Creation** dialog box is updated to indicate that the thread representation will be created for the selected thread.

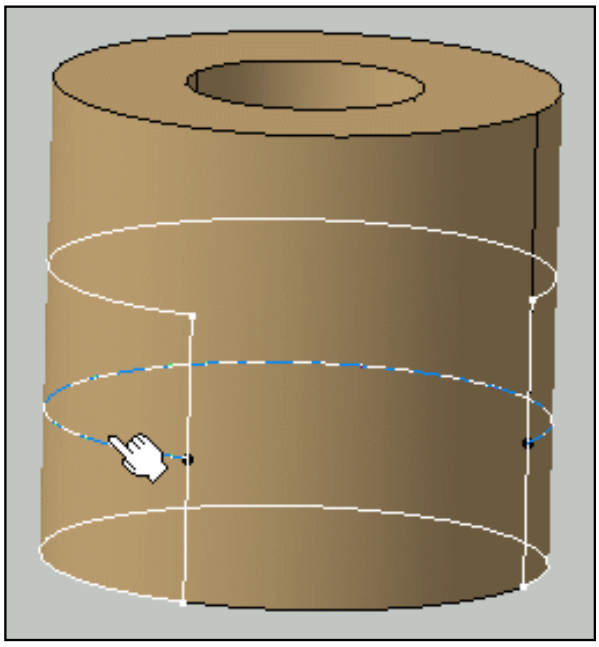


In the case of numerous threads, selecting the **All threads** option lets you create the thread representations of all of them.

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.

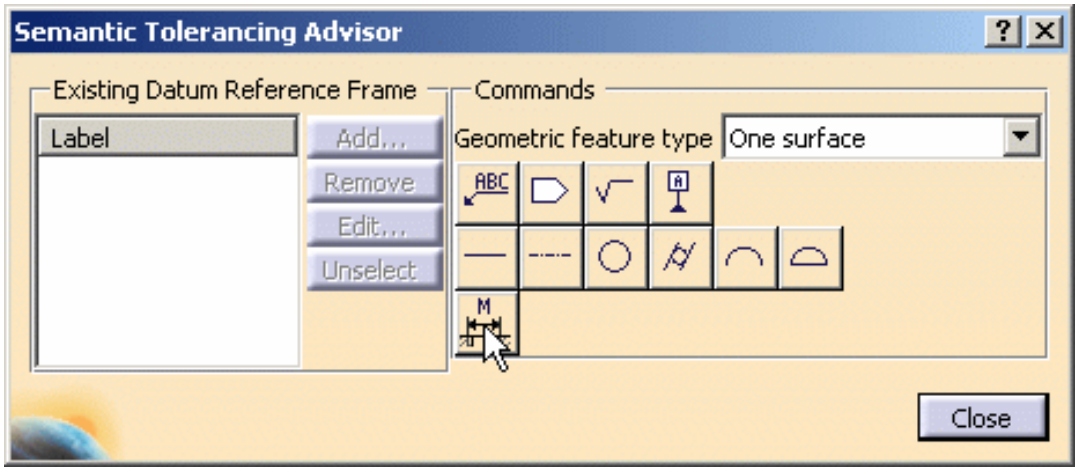


4. Click the **Tolerancing Advisor** icon:  The **Semantic Tolerancing Advisor** dialog box is displayed.
5. Select the median 3/4 circle arc which symbolizes the thread helical surface.



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

6. Click the **Diameter** icon.



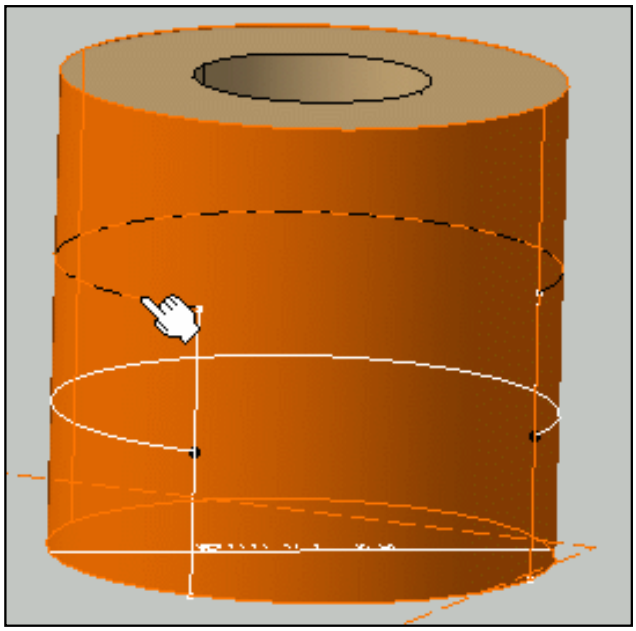
The thread diameter dimension is previewed and the **Limit of Size Definition** dialog box is displayed, offering the following options:

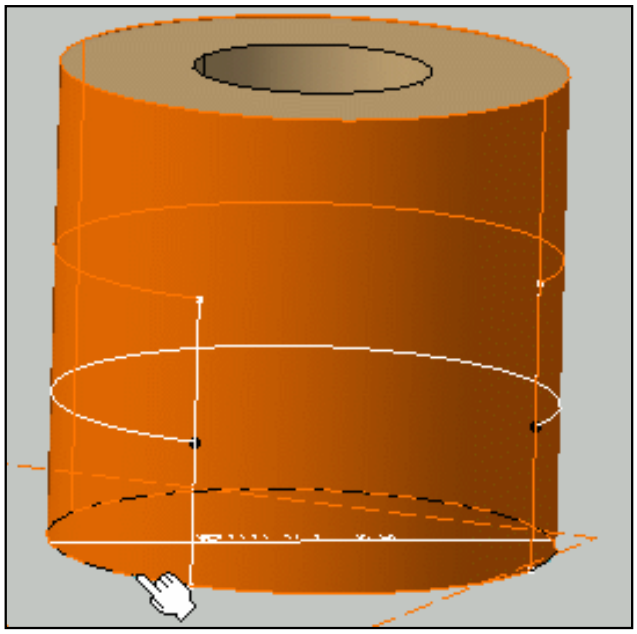
- **Pitch**: lets you display the pitch value in the thread dimension.
- **Tol class**: lets you define and display the tolerance class value in the thread dimension.

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



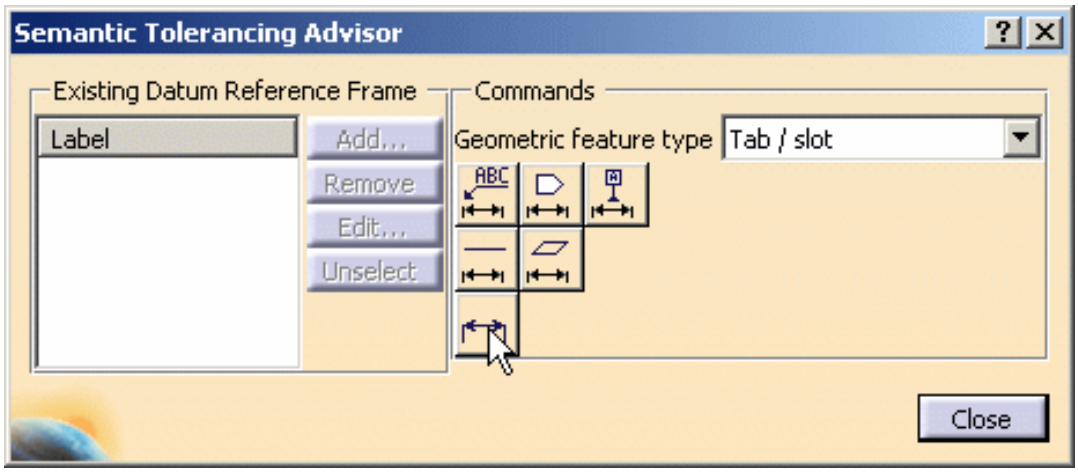
9. Click **OK** to validate. The thread diameter dimension is created.
10. Back in the **Semantic Tolerancing Advisor** dialog box, multi-select (using the **Ctrl** key, for example) the 3/4 circle arcs which symbolize the thread starting and ending planes.





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

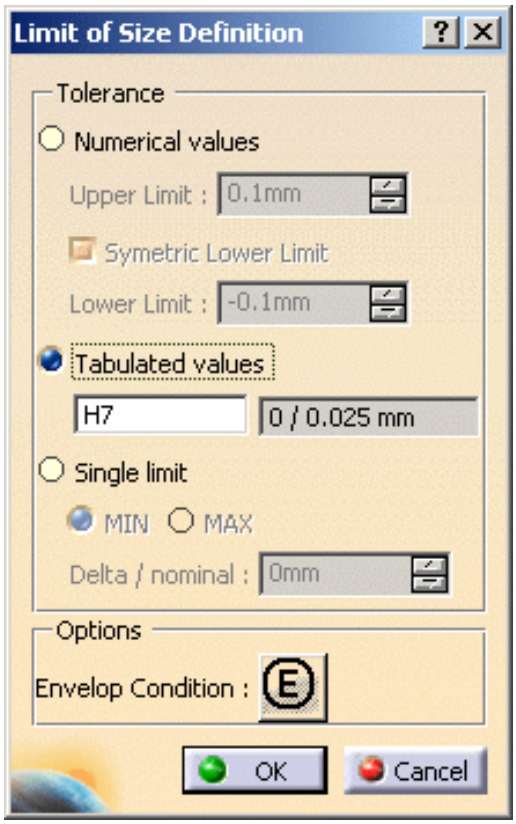
11. Click the **Distance Creation** icon.



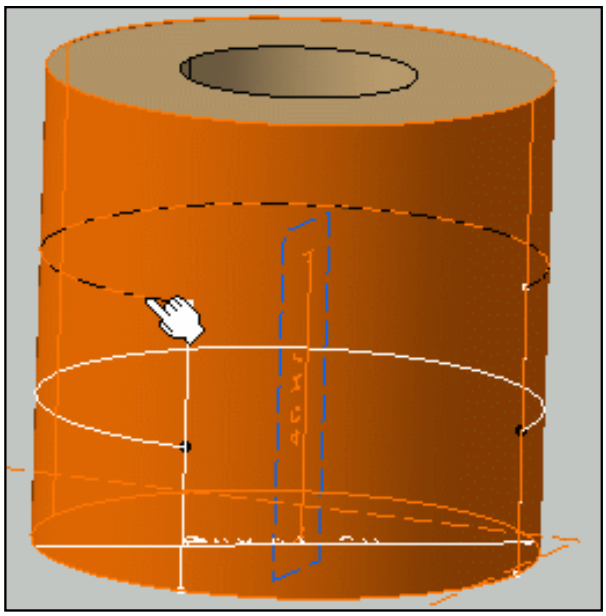
The thread length dimension is previewed and the **Limit of Size Definition** dialog box is displayed, offering the following options:

- **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.

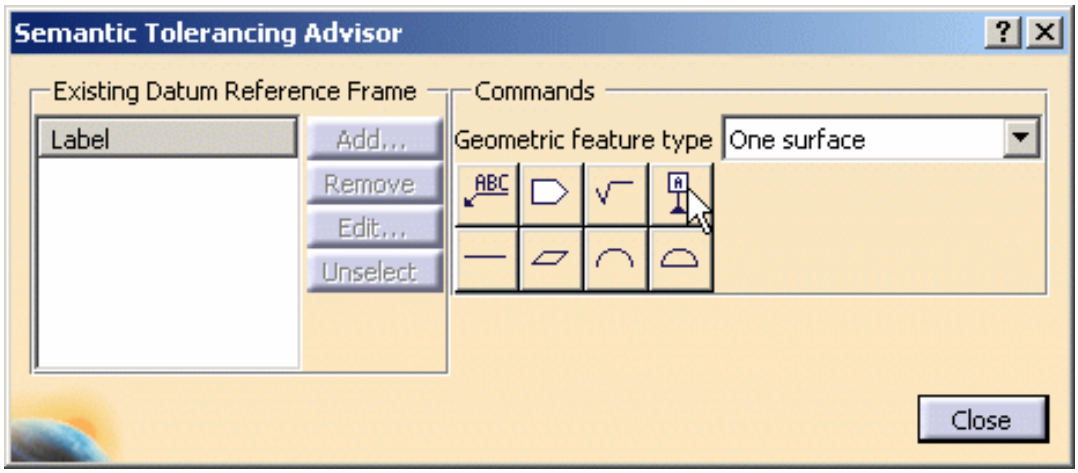
12. Select the **Tabulated values** option.



13. Click **OK** to validate.
14. When a dialog box appears, informing you that the annotation will be created on another view, click **OK**. The thread length dimension is created.
15. Back in the **Semantic Tolerancing Advisor** dialog box, select the 3/4 circle arc which symbolizes the thread starting plane.



16. Click the **Semantic Datum** icon.

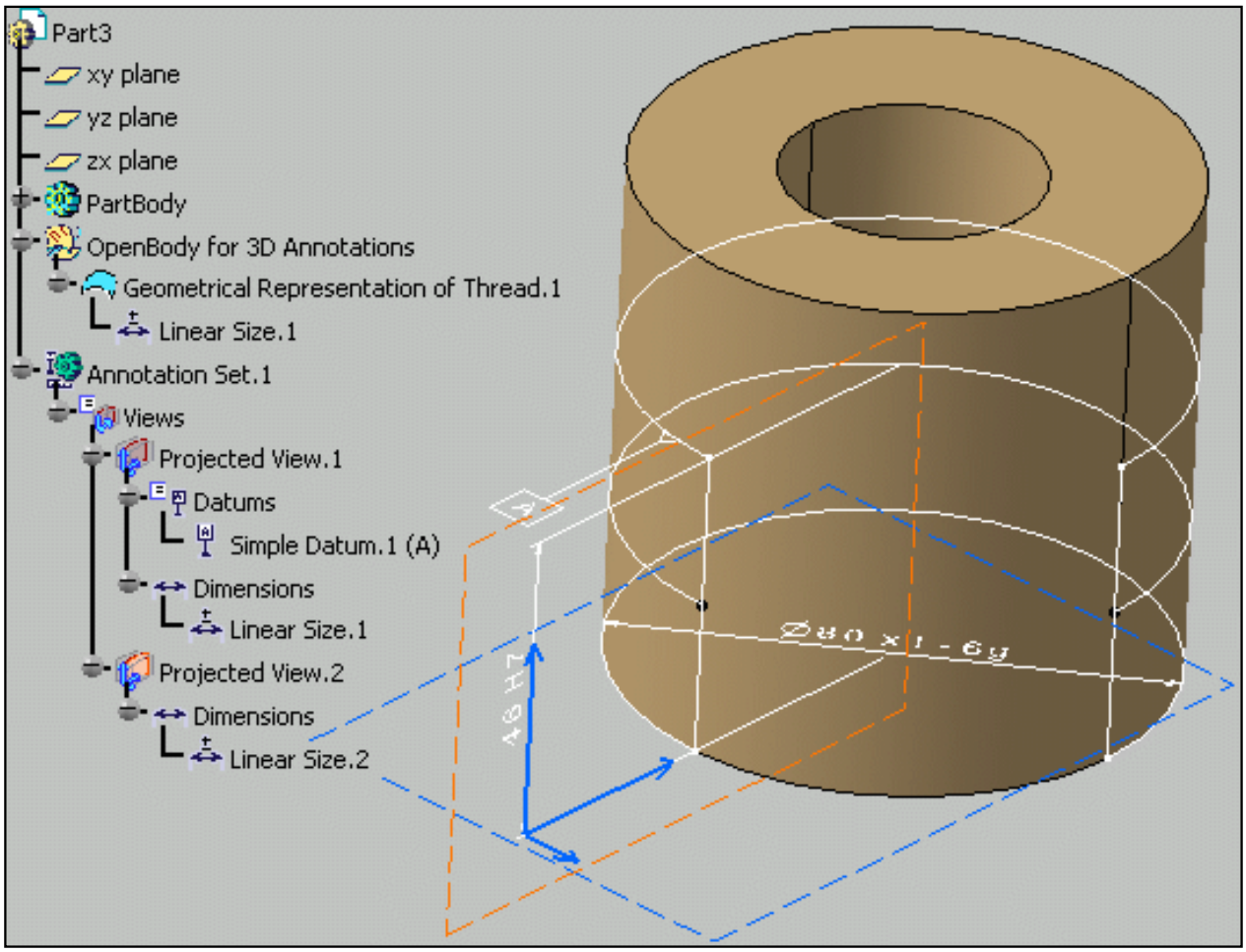


The **Datum Definition** dialog box is displayed.

17. 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.

18. 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
M8x1-6g

Back Up

Tolerancing Body in White

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

Creating Datum and Datum Targets
Creating Dimensions and Associated Datum
Creating a Datum Reference Frame
Tolerancing Body in White Holes
Tolerancing Body in White Surface

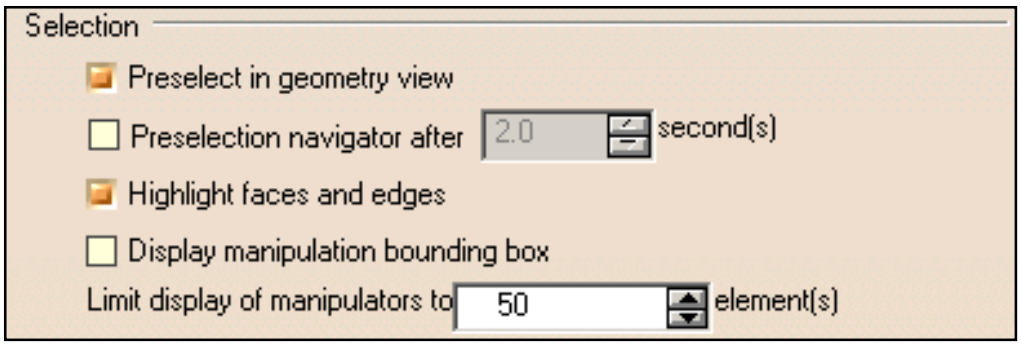
Back Up Next

Creating Datum and Datum Targets

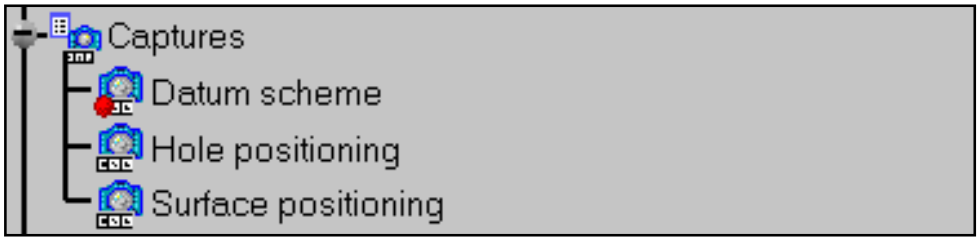
This task shows you how to create datum and datum targets on body in white surfaces.

Open the Tolerancing_Annotations_06 CATPart document.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

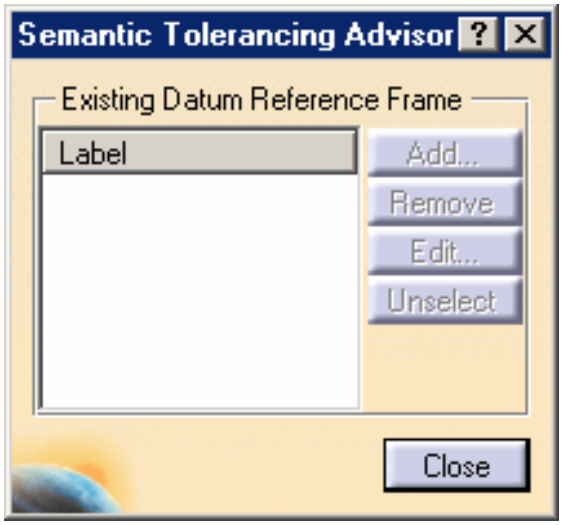


1. Right-click the **Datum scheme** capture and select **Set Current** form the contextual menu: all created annotations will be added to this capture as long as it is current.

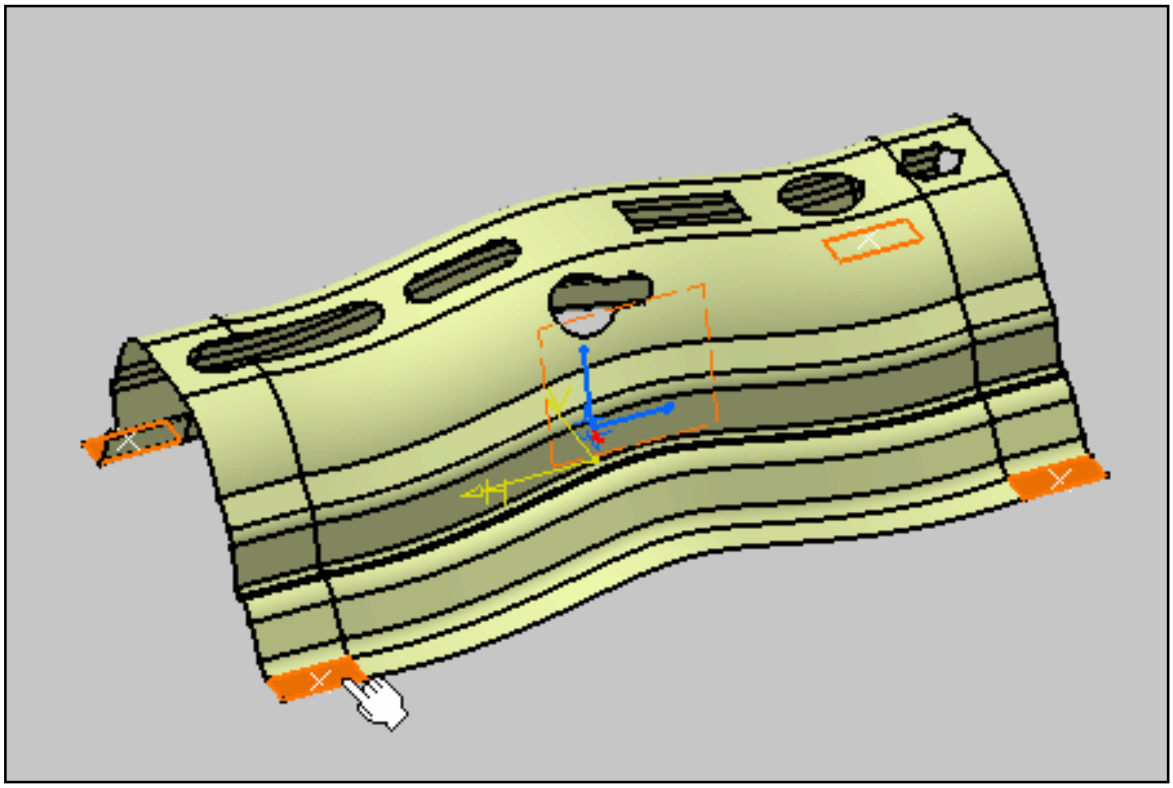


2. Click the **Tolerancing Advisor** icon: 

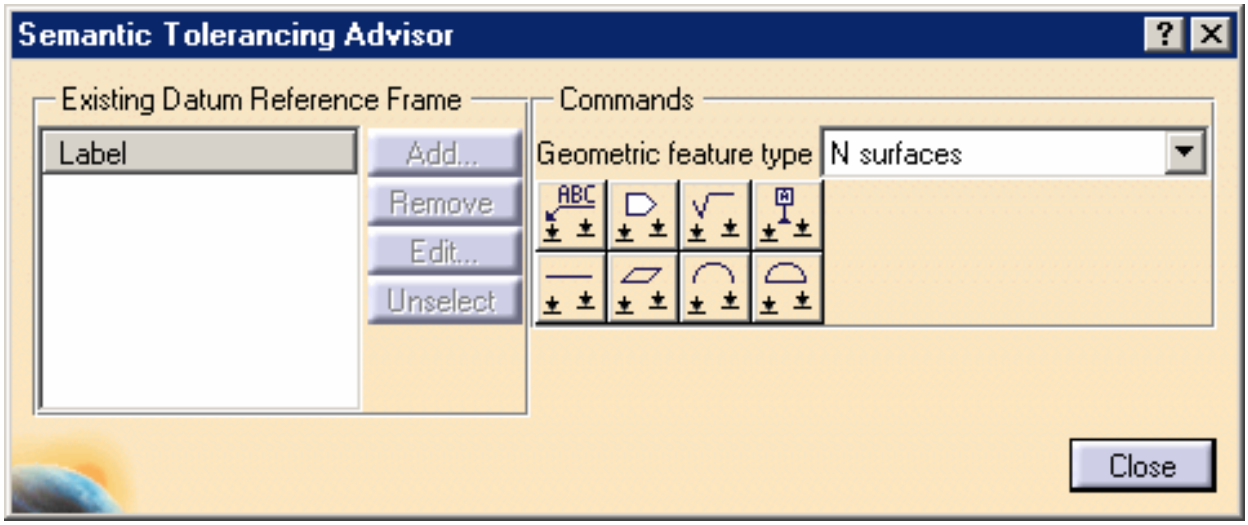
The **Semantic Tolerancing Advisor** dialog box appears.



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

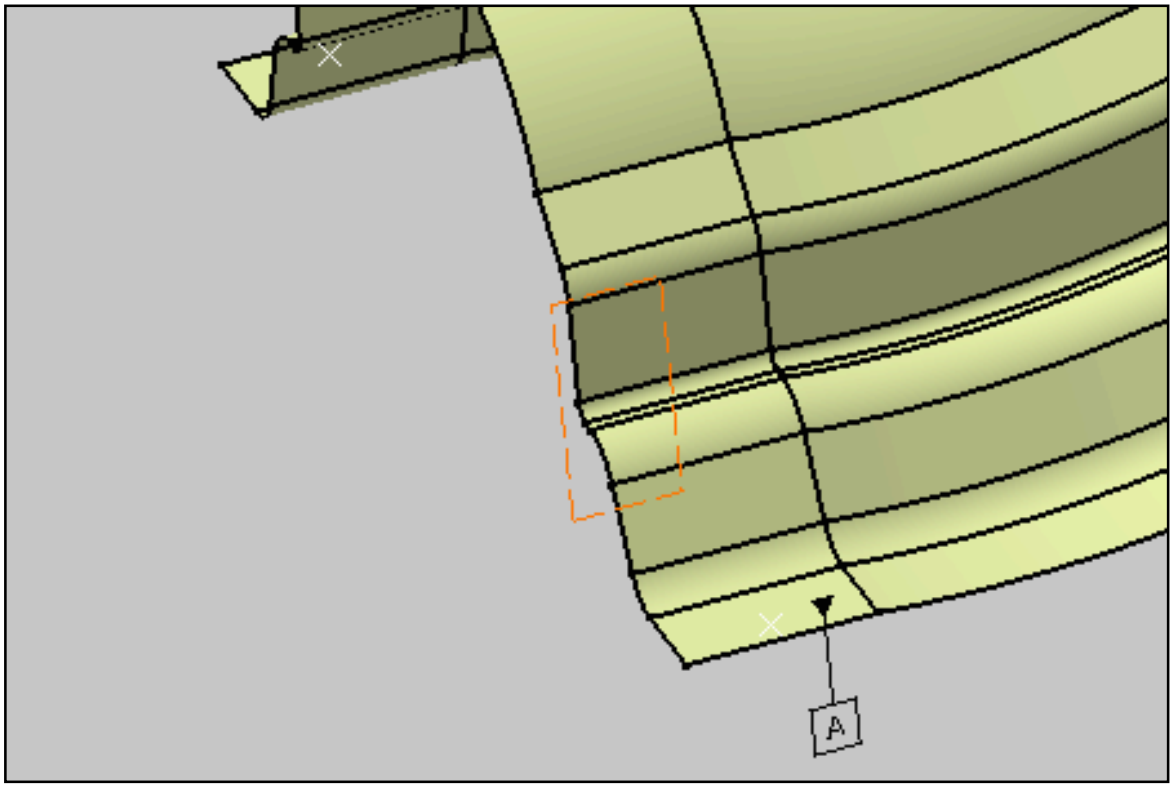


The **Semantic Tolerancing Advisor** dialog box appears.
The buttons and options available in the dialog box depend on your selection.

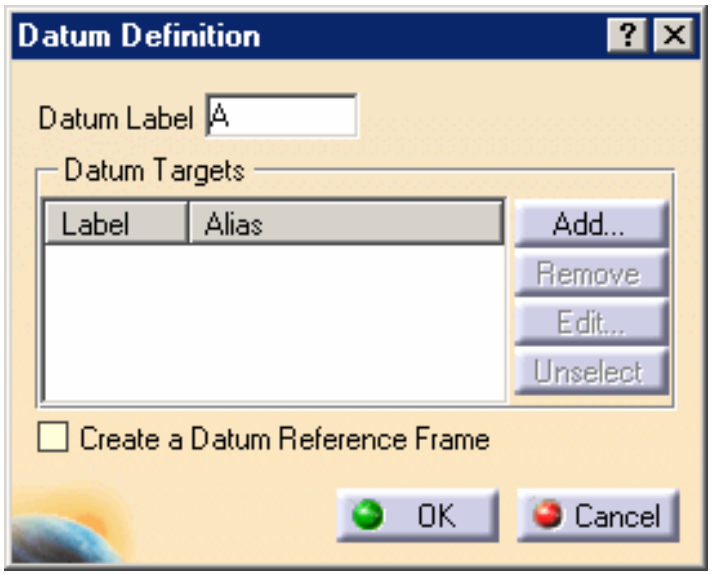


4. Click the **Semantic Datum** icon (N surfaces): 

The datum is created.



The **Datum Definition** dialog box appears.

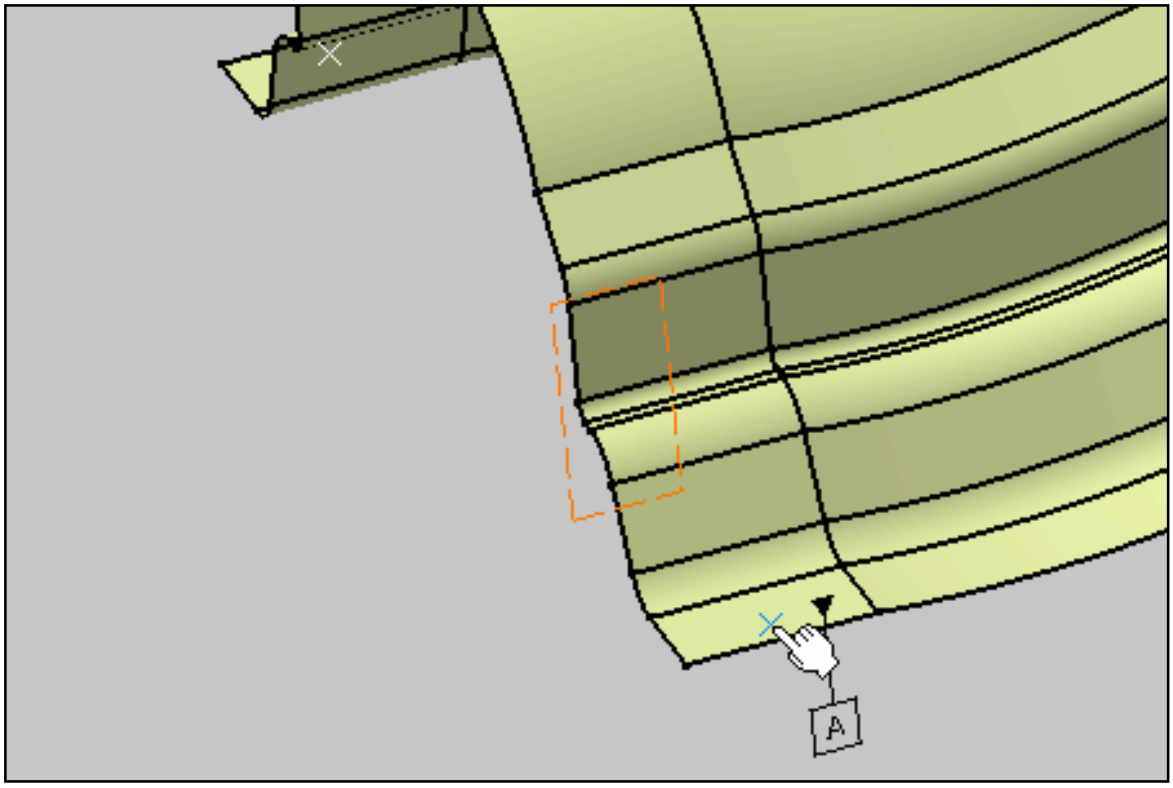


5. Click **Add** in the **Datum Definition** dialog box.

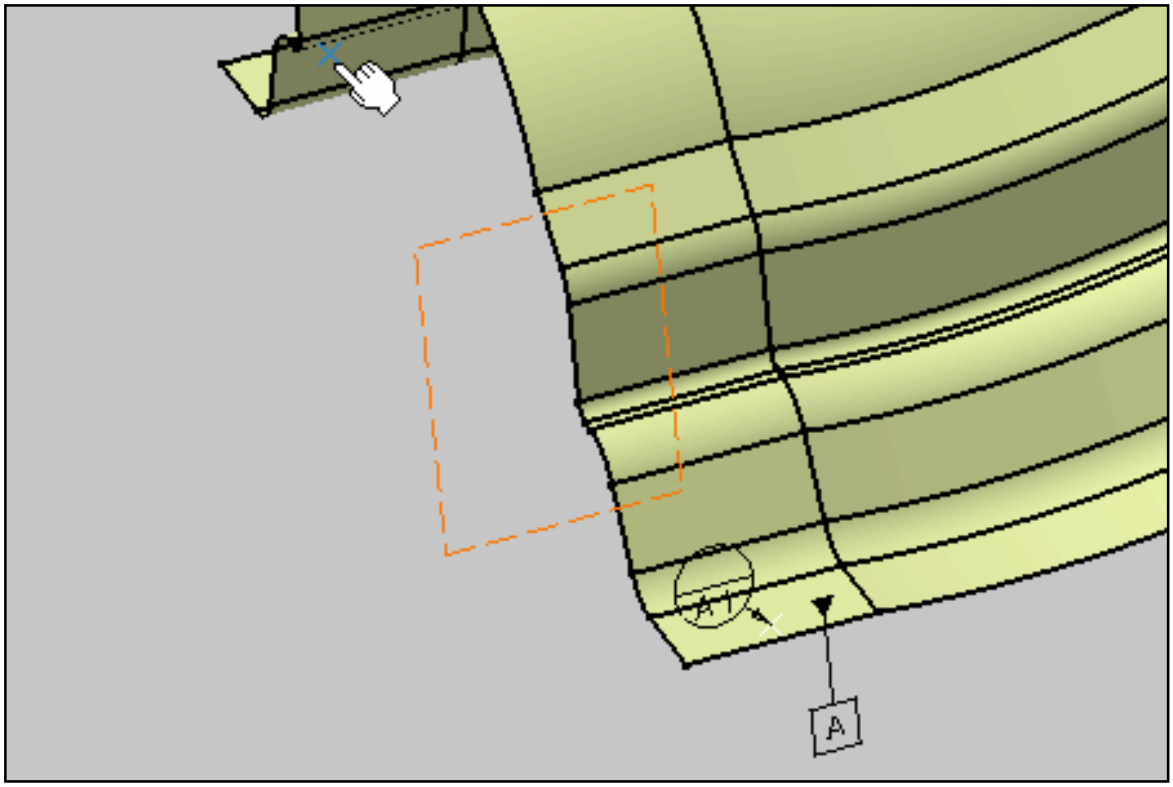
   The **Datum Target** dialog box appears. Keep the options as is.

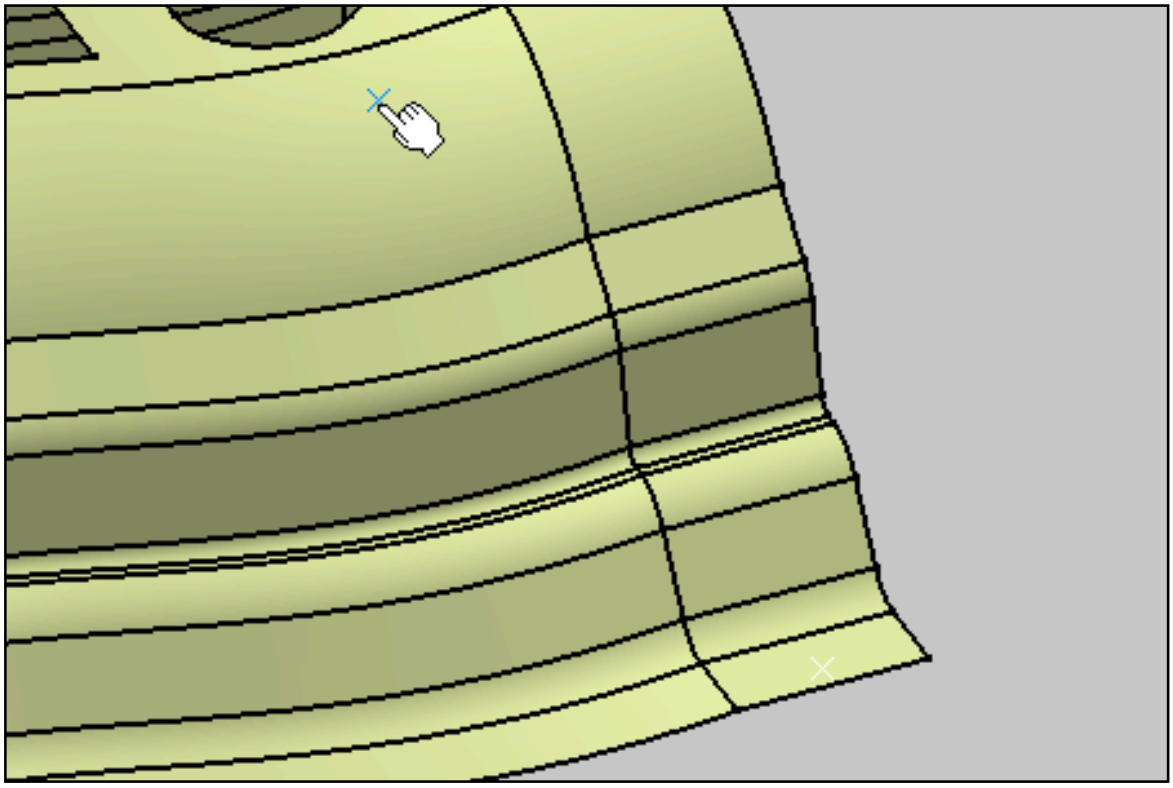


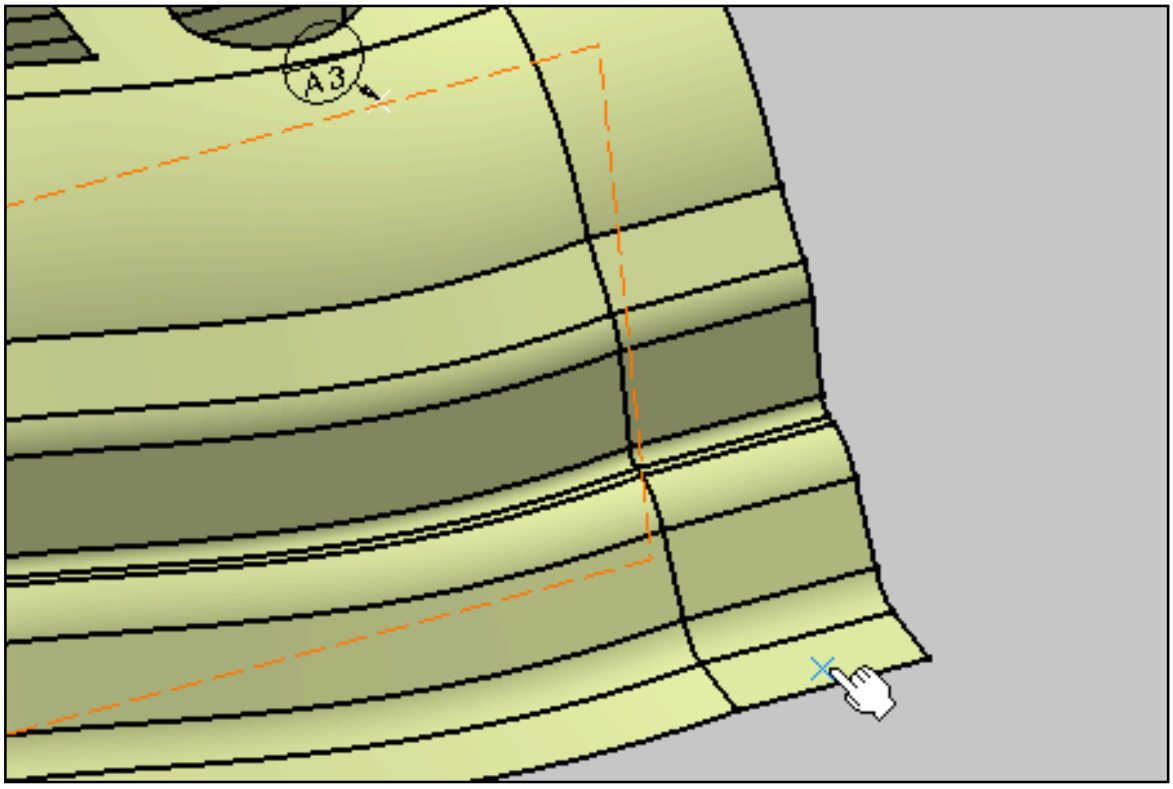
6. Select the target points as shown on the part.



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

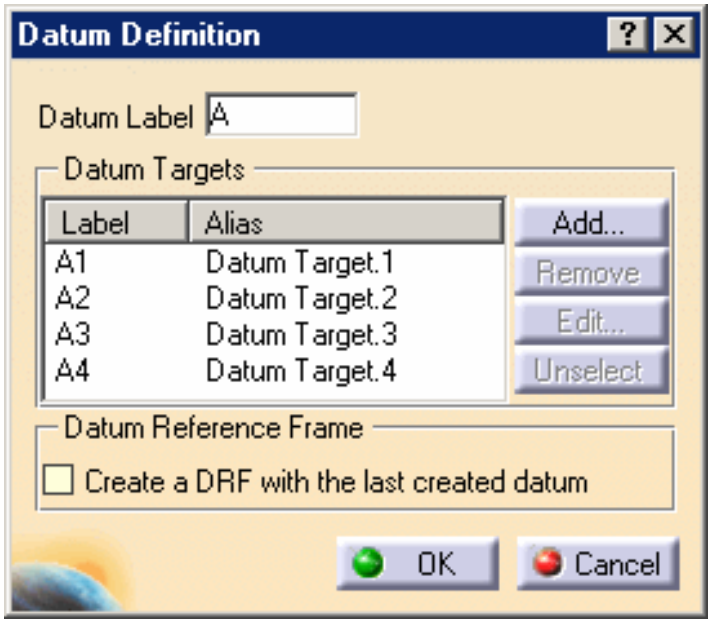






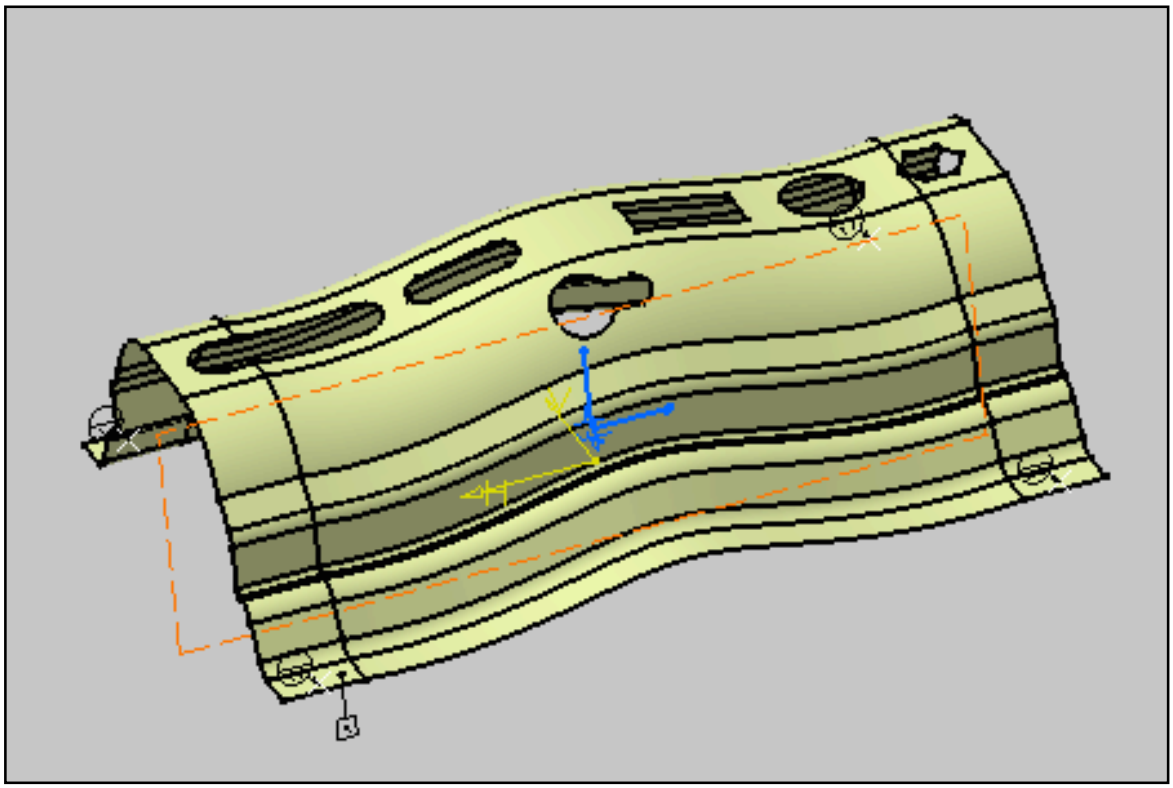
7. Click **OK** in the last **Datum Target** dialog box to end the datum target creation.

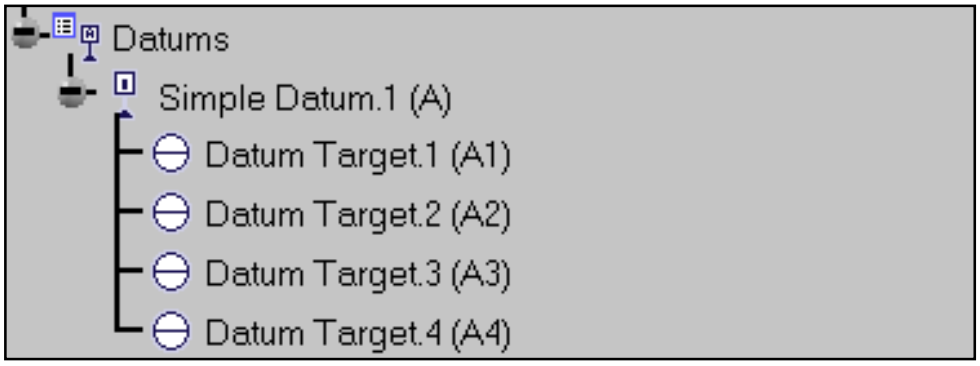
   The **Datum Definition** dialog box is updated.



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.

Up Next

Creating Dimensions and Associated Datum

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

Five hole types are available for tolerancing:

Cylindrical hole.

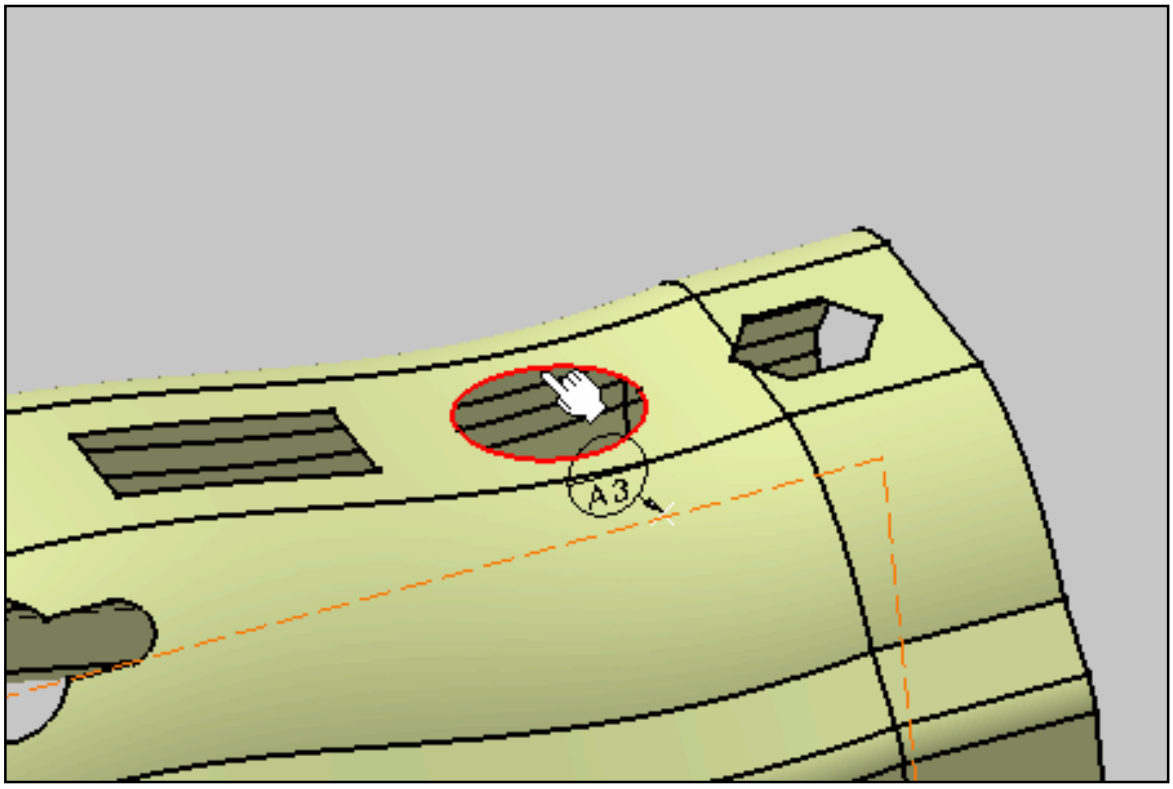
Elongated hole.

Rectangular hole with angular corners.

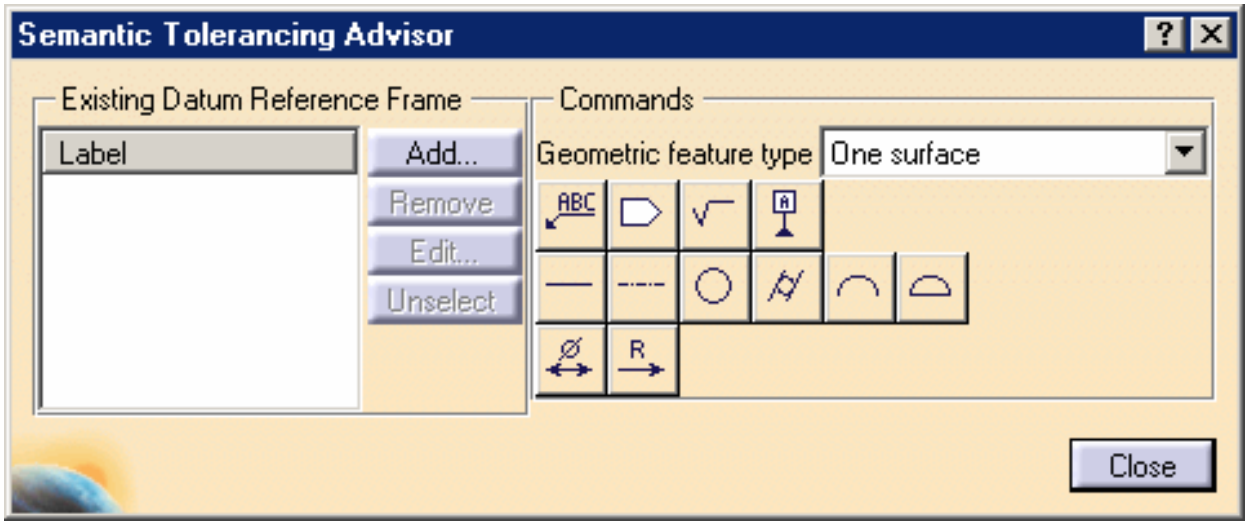
Rectangular hole with rounded corners.

Other hole from previous.

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

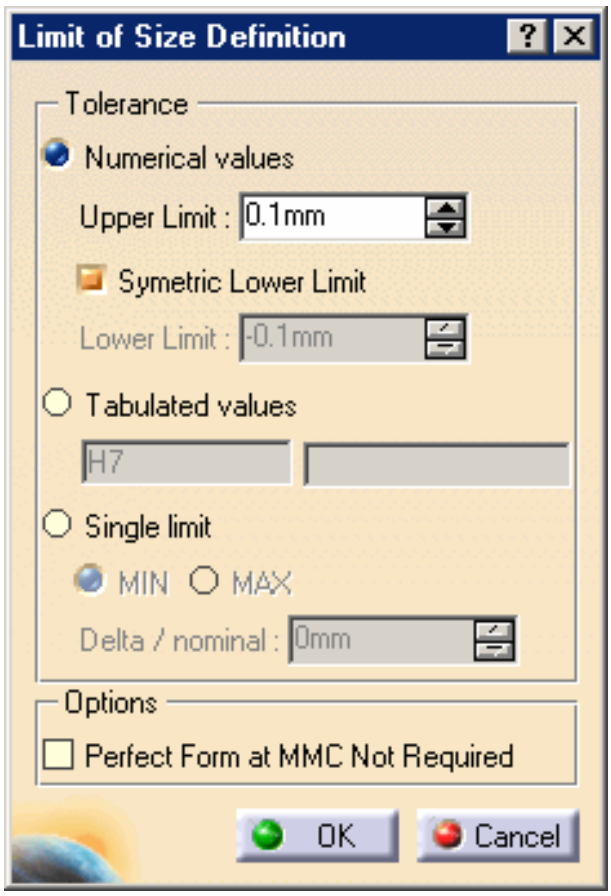


The **Semantic Tolerancing Advisor** dialog box is updated.



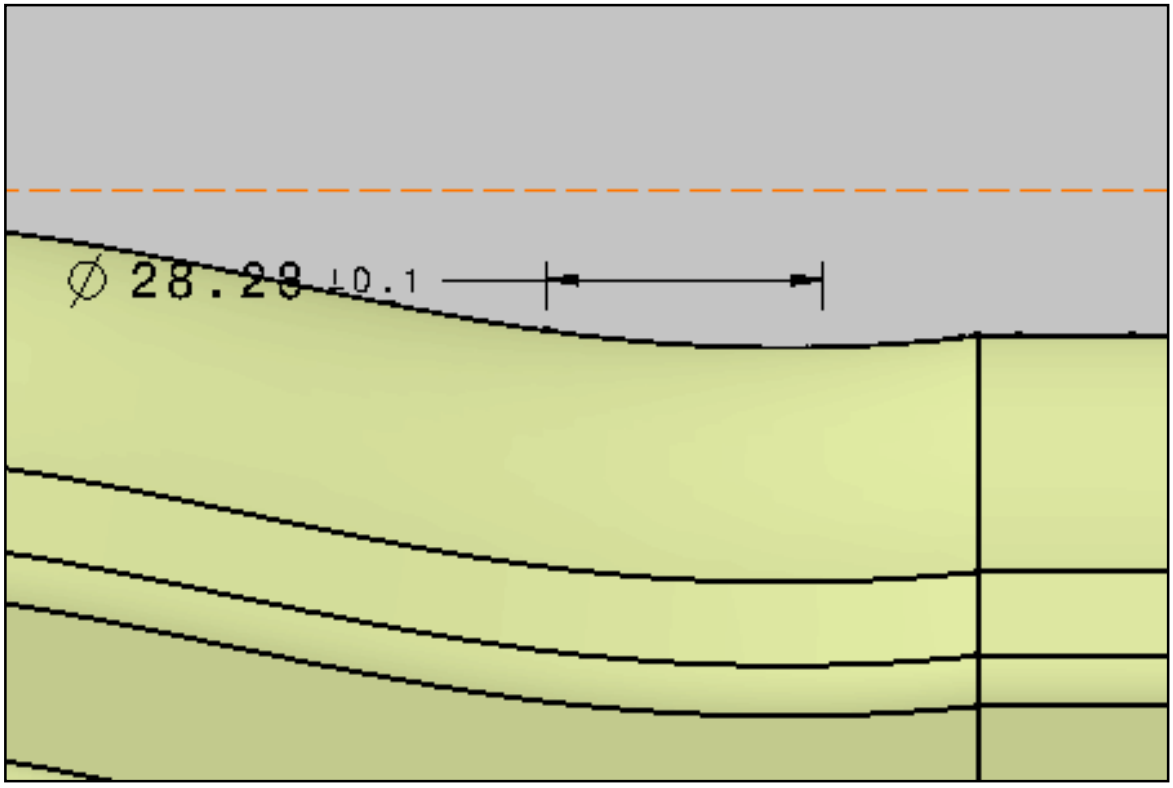
2. Click the **Diameter** icon (One surface): 

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



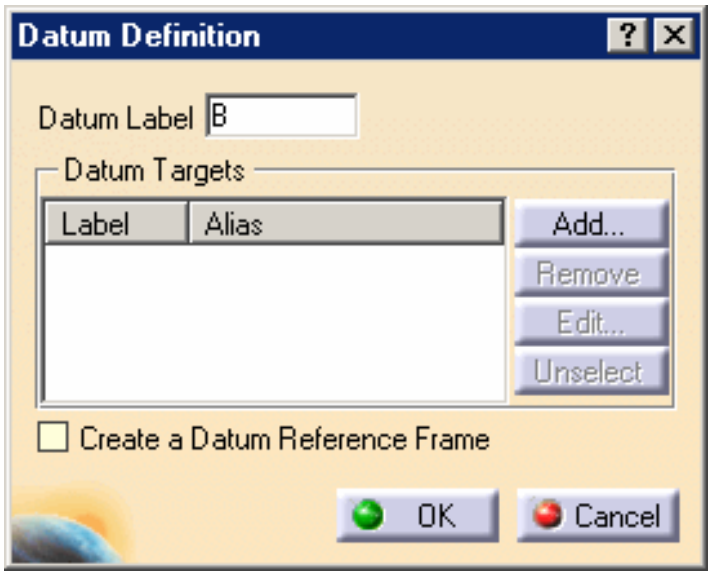
3. Click the **OK** in the dialog box.

The dimension is created.



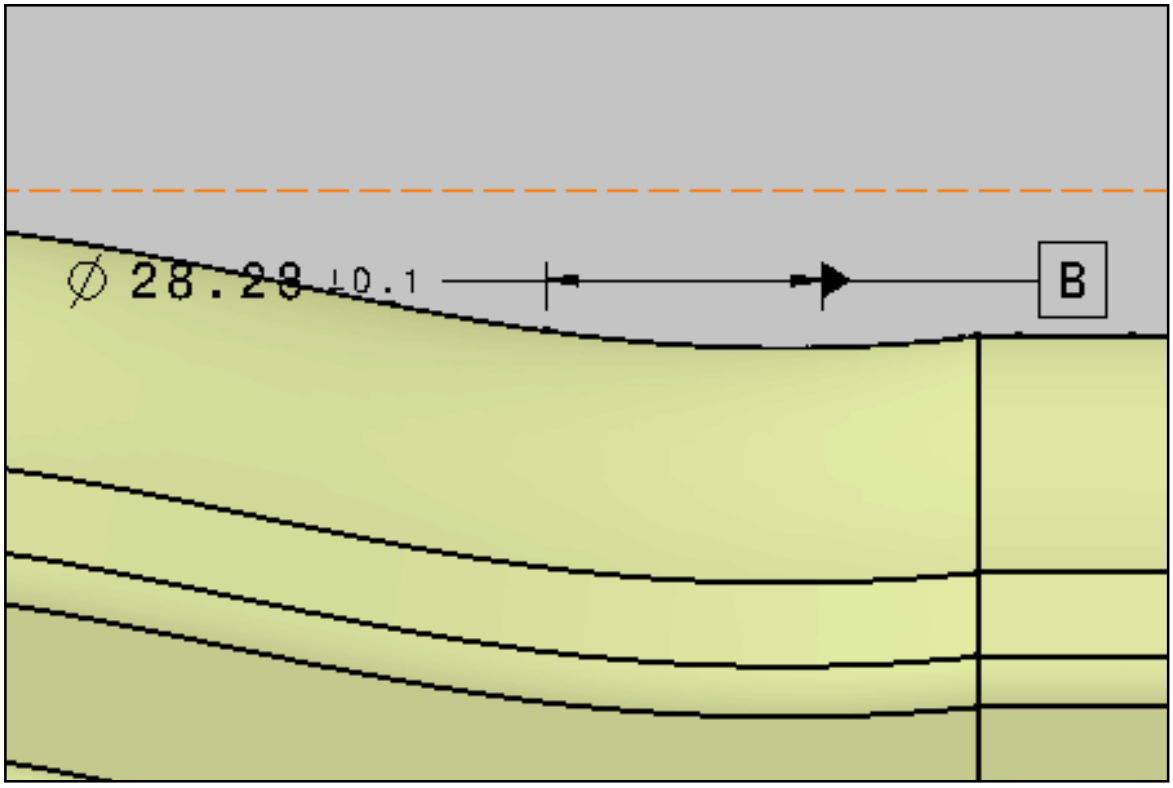
4. Click the **Semantic Datum** icon (One surface): 

   The **Datum Definition** dialog box appears.

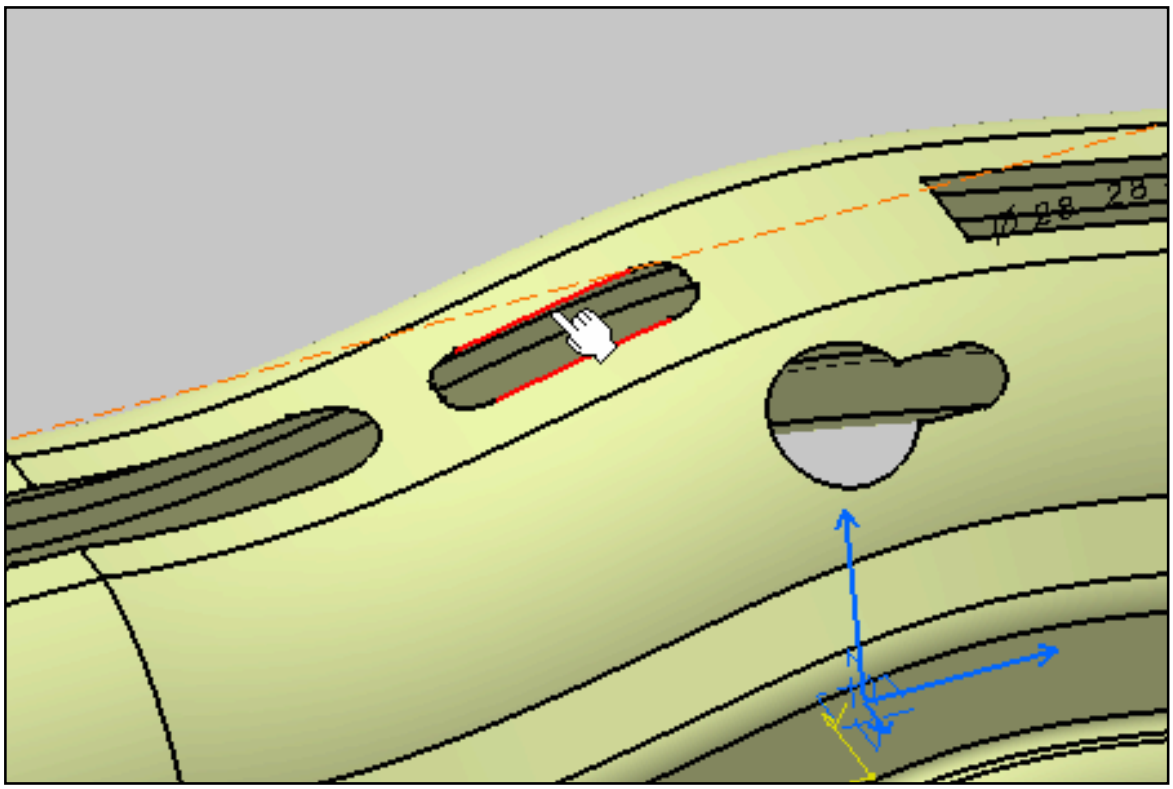


5. Click the **OK** in the dialog box.

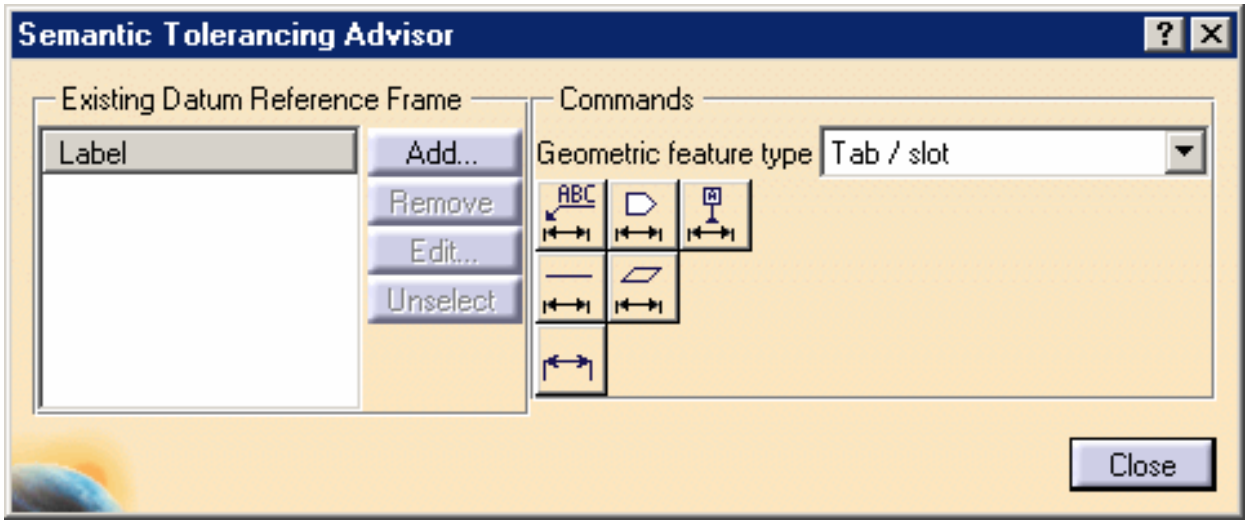
   The datum is added to the dimension.



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



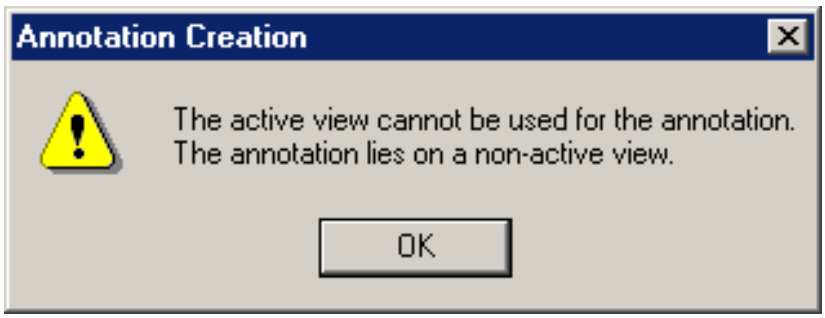
The **Semantic Tolerancing Advisor** dialog box is updated.



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

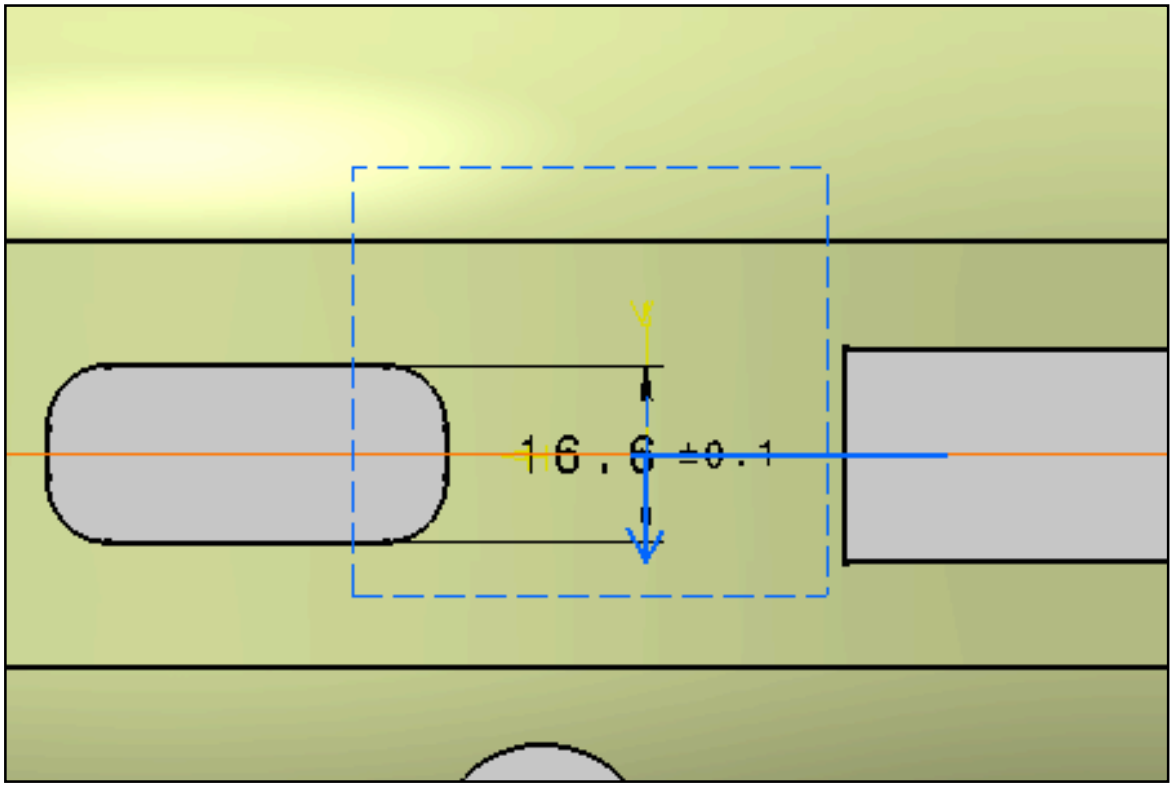
   The **Limit of Size Definition** dialog box appears. Keep options as is.

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



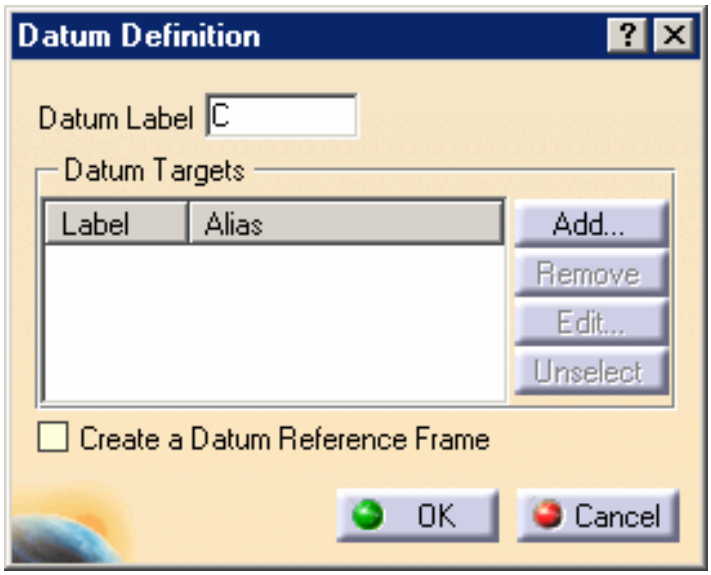
9. Click the **OK** in the dialog box.

   The dimension is created.



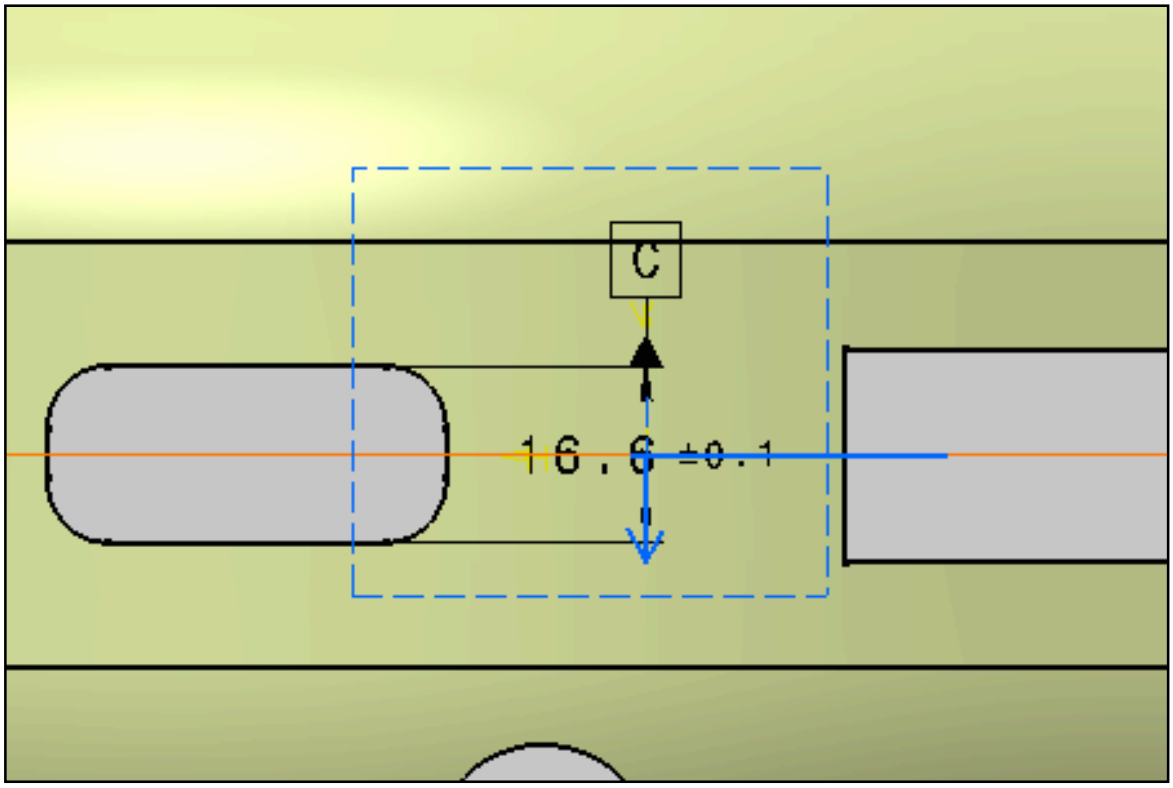
10. Click the **Semantic Datum** icon (Tab/Slot): 

The **Datum Definition** dialog box appears.

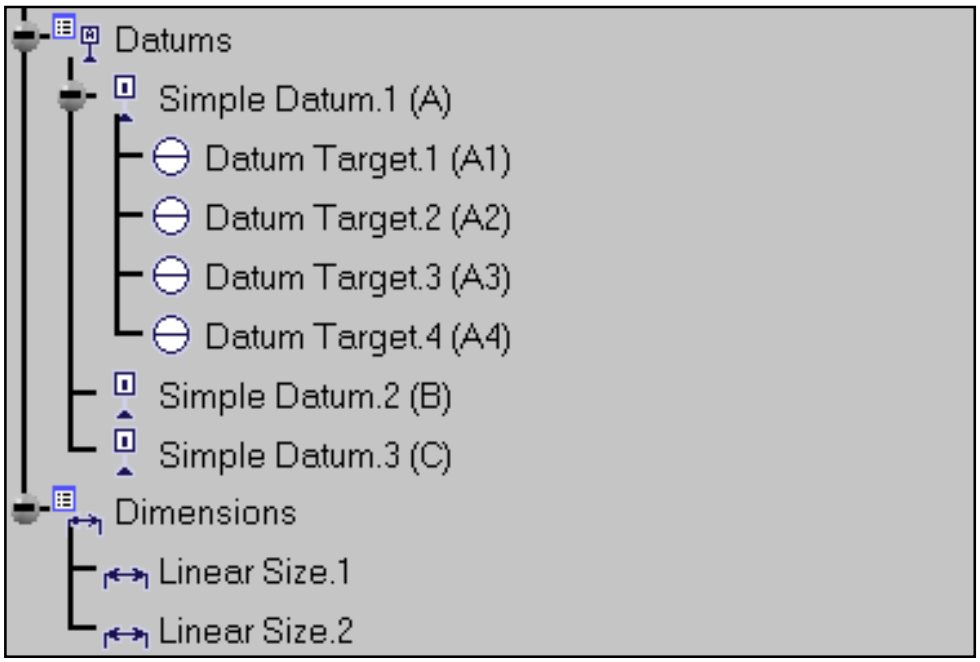


11. Click the **OK** in the dialog box.

The datum is added to the dimension.



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



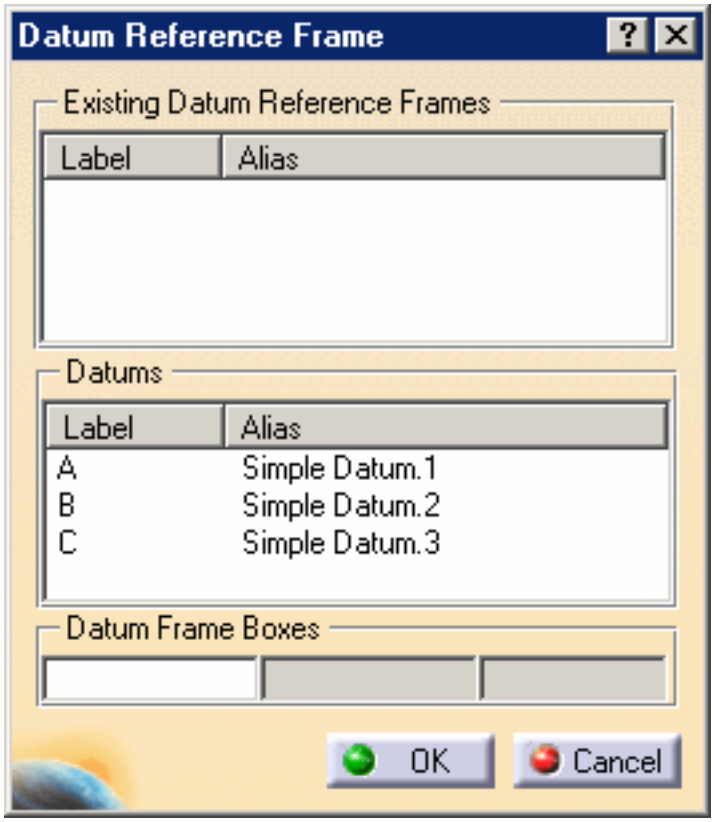
12. Do not close the **Semantic Tolerancing Advisor** dialog box to perform the next task.

Creating a Datum Reference Frame

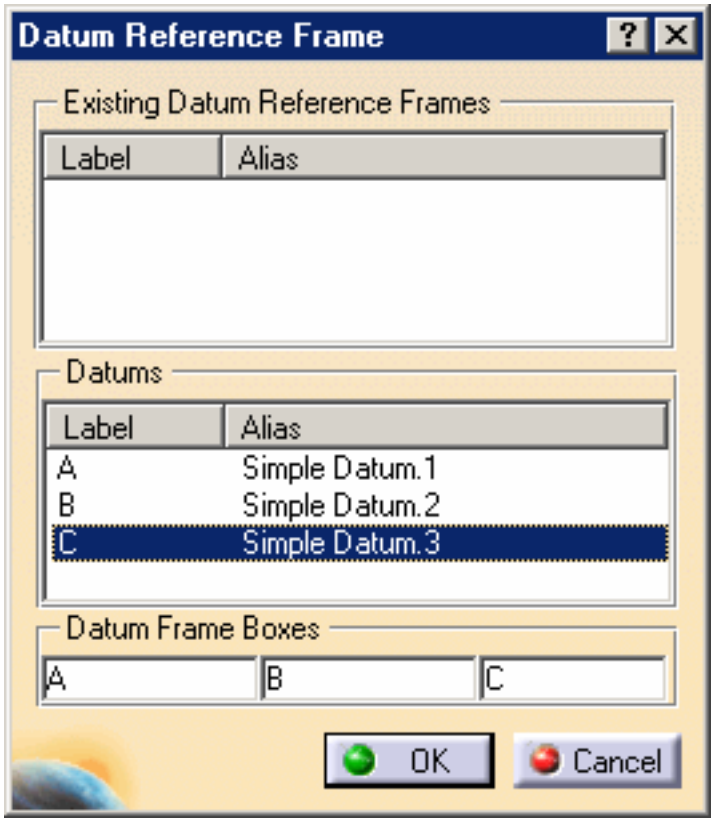
This task shows you how to create a datum reference frame on body in white surfaces.

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

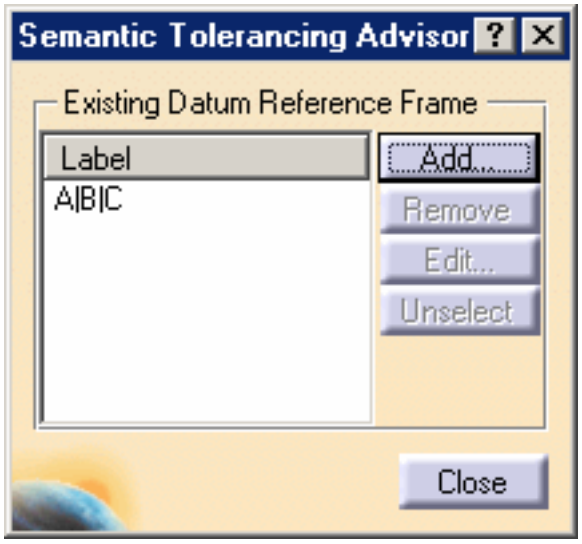
   The **Datum Reference Frame** dialog box appears.



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.



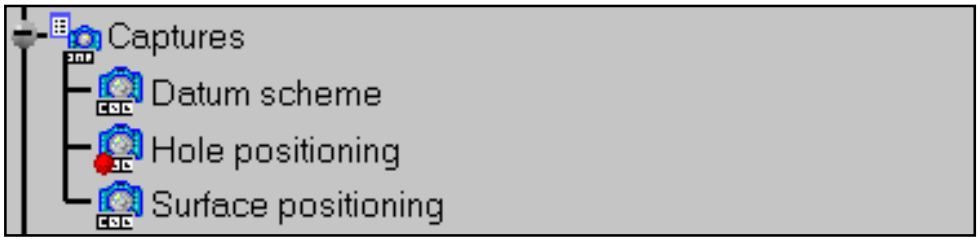
4. Do not close the **Semantic Tolerancing Advisor** dialog box to perform the next task.

Back Up Next

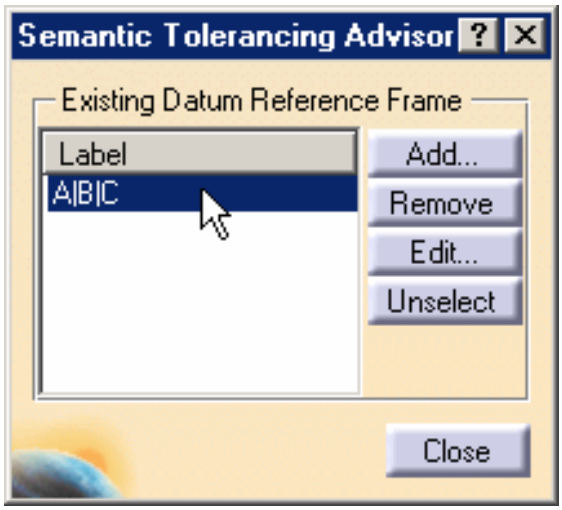
Tolerancing Body in White Holes

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

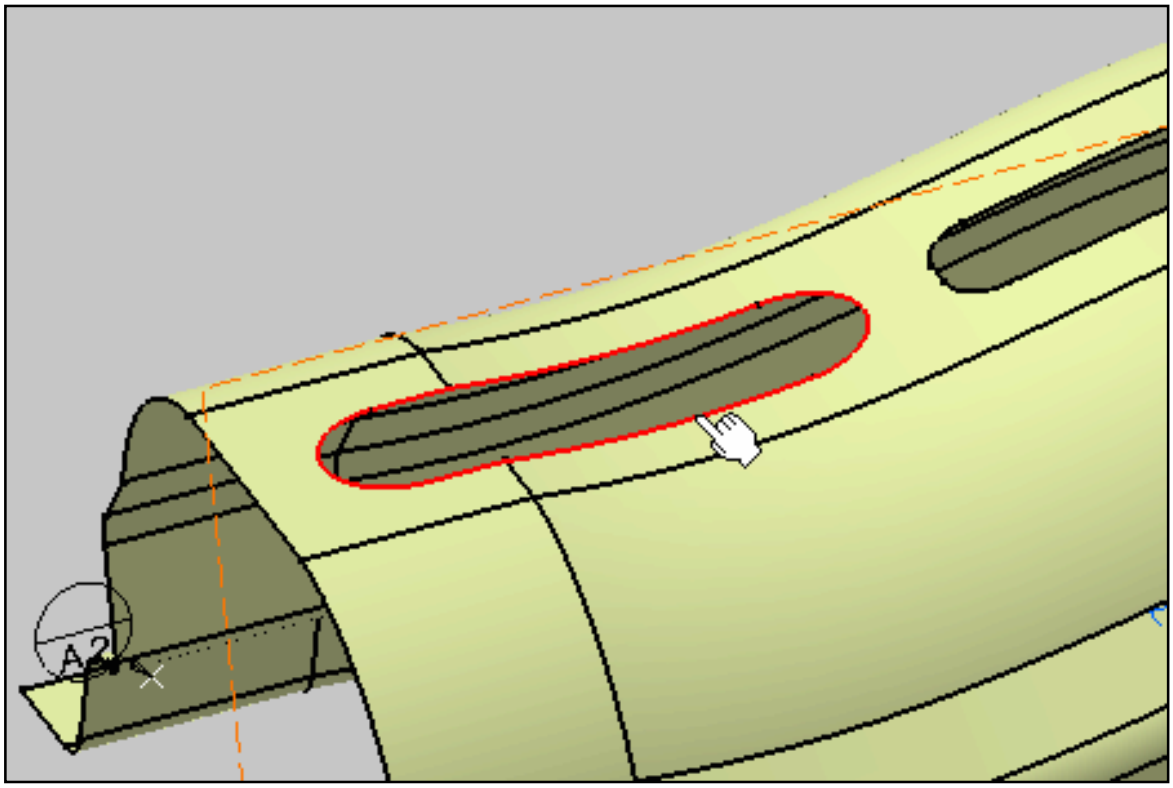
1. Right-click the **Datum scheme** capture and select **Unset Current** form the contextual menu, right-click the **Hole positioning** capture and select **Set Current** form the contextual menu.

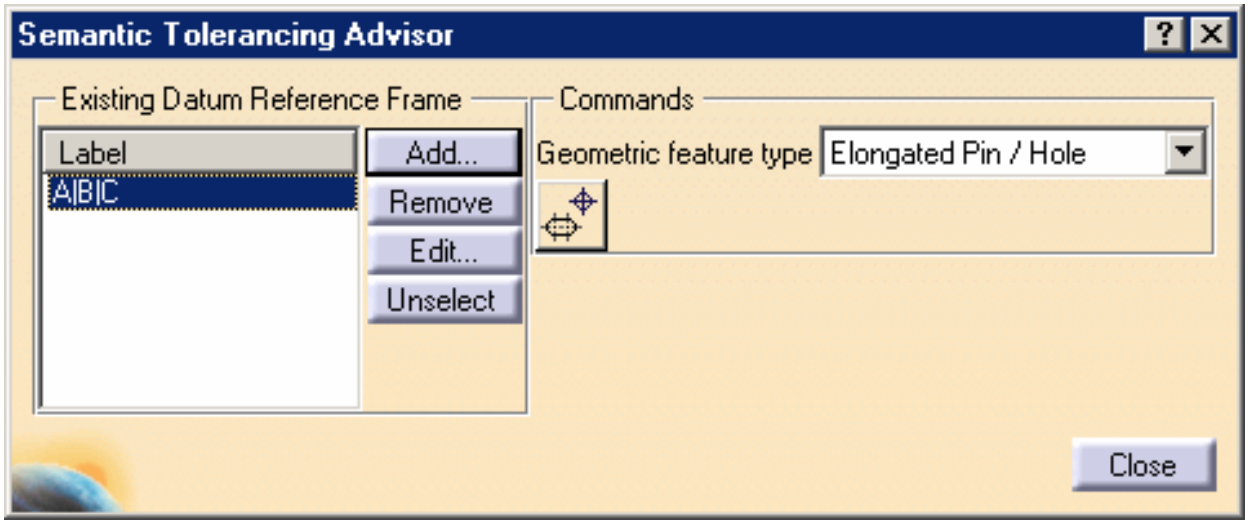


2. Select the datum reference frame in the **Semantic Tolerancing Advisor** dialog box.



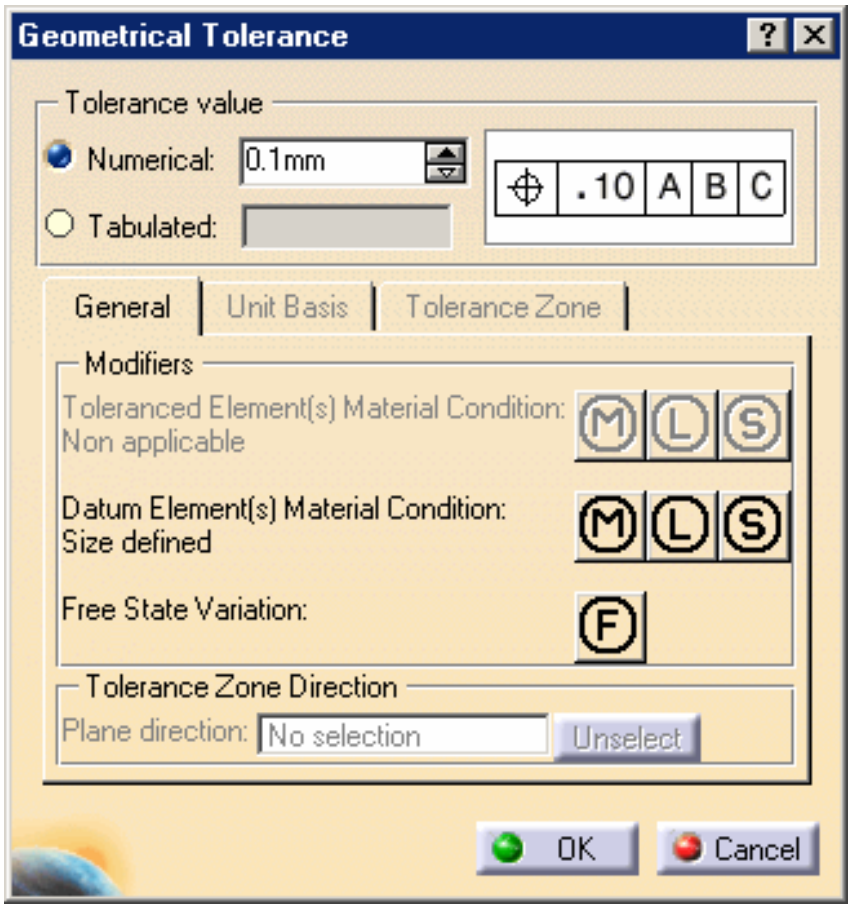
3. Select the whole hole edges as shown on the part.





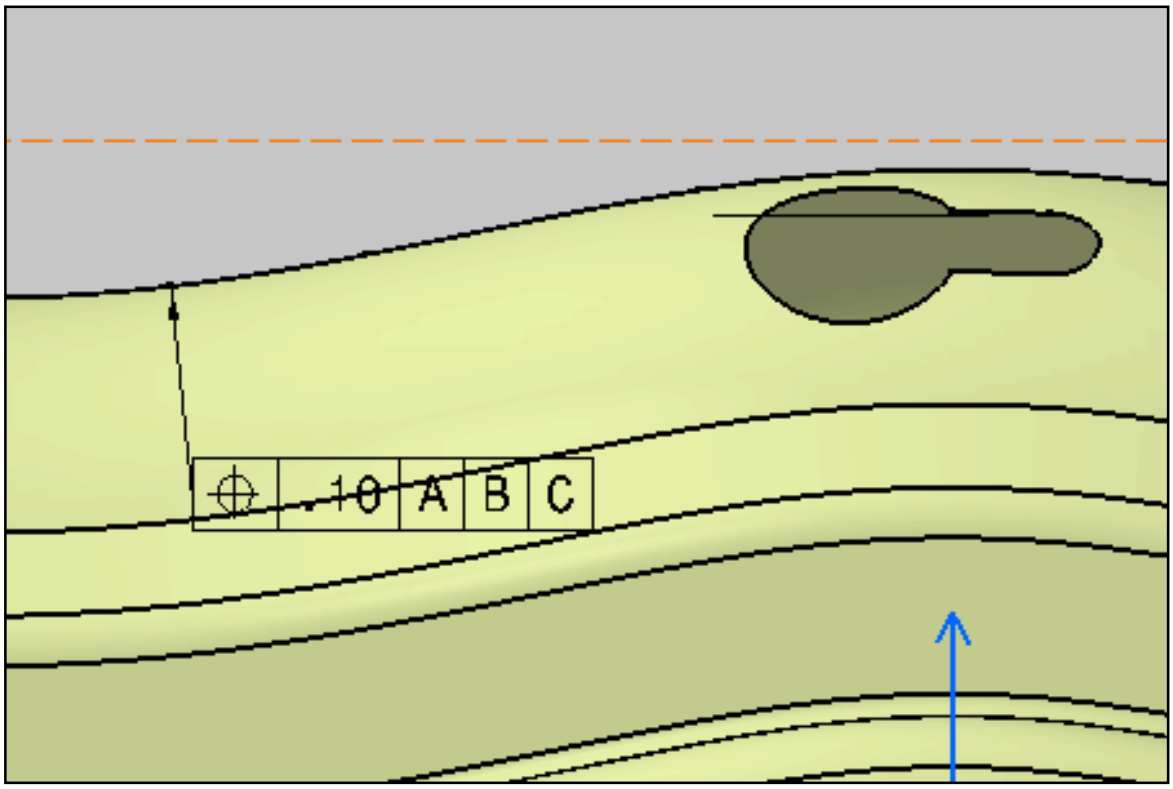
The **Semantic Tolerancing Advisor** dialog box is updated. Note the geometric feature type: **Elongated Pin / Hole**.

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

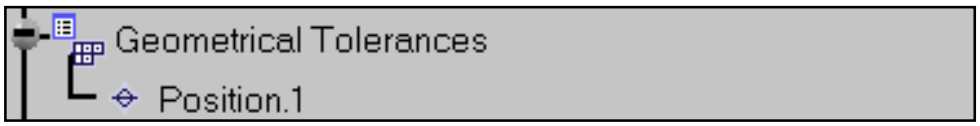


The **Geometrical Tolerance** dialog box appears. Keep options as is.

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

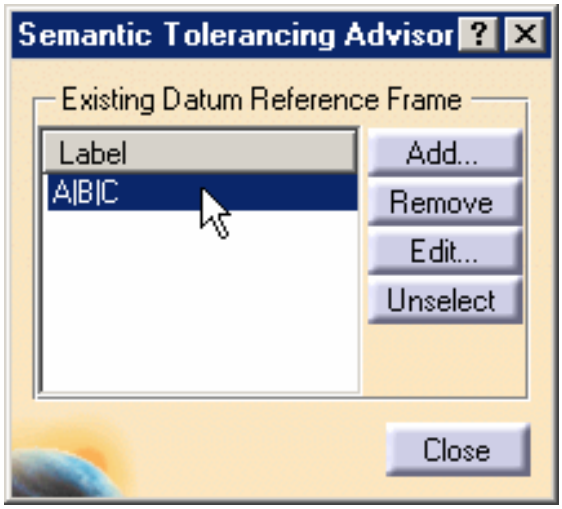


The geometrical tolerances is created.

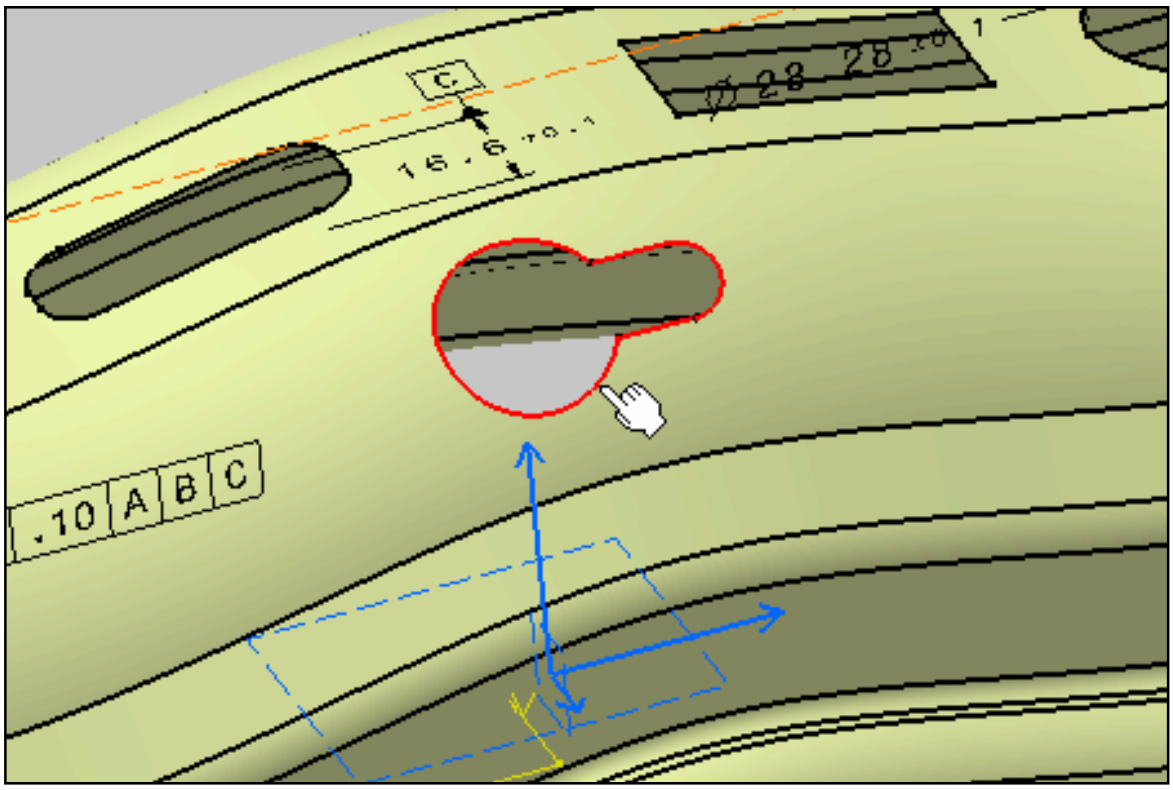


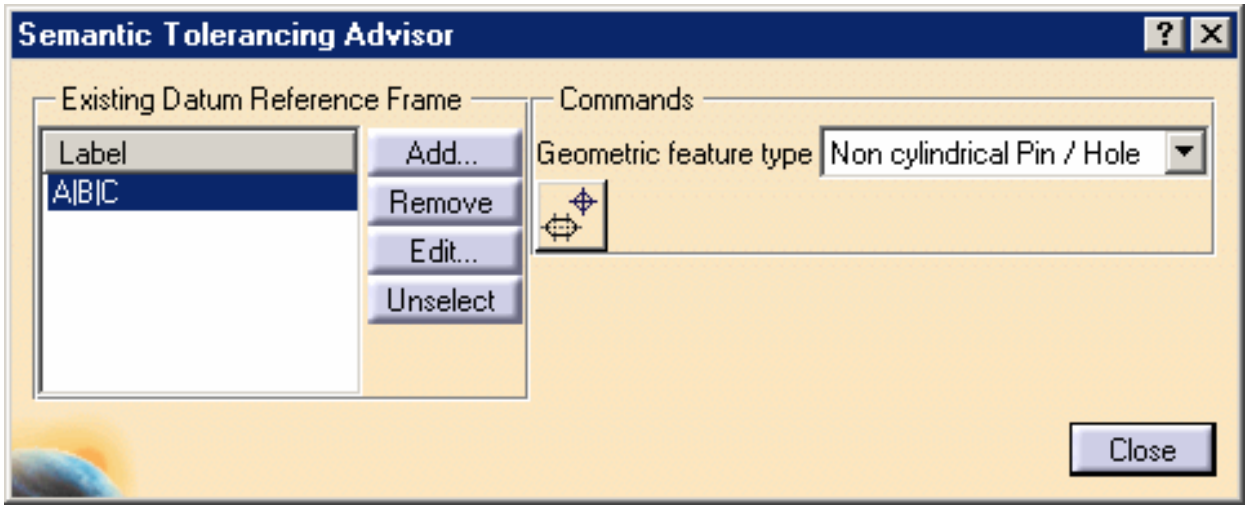
The geometrical tolerances is created in the specification tree.

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



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



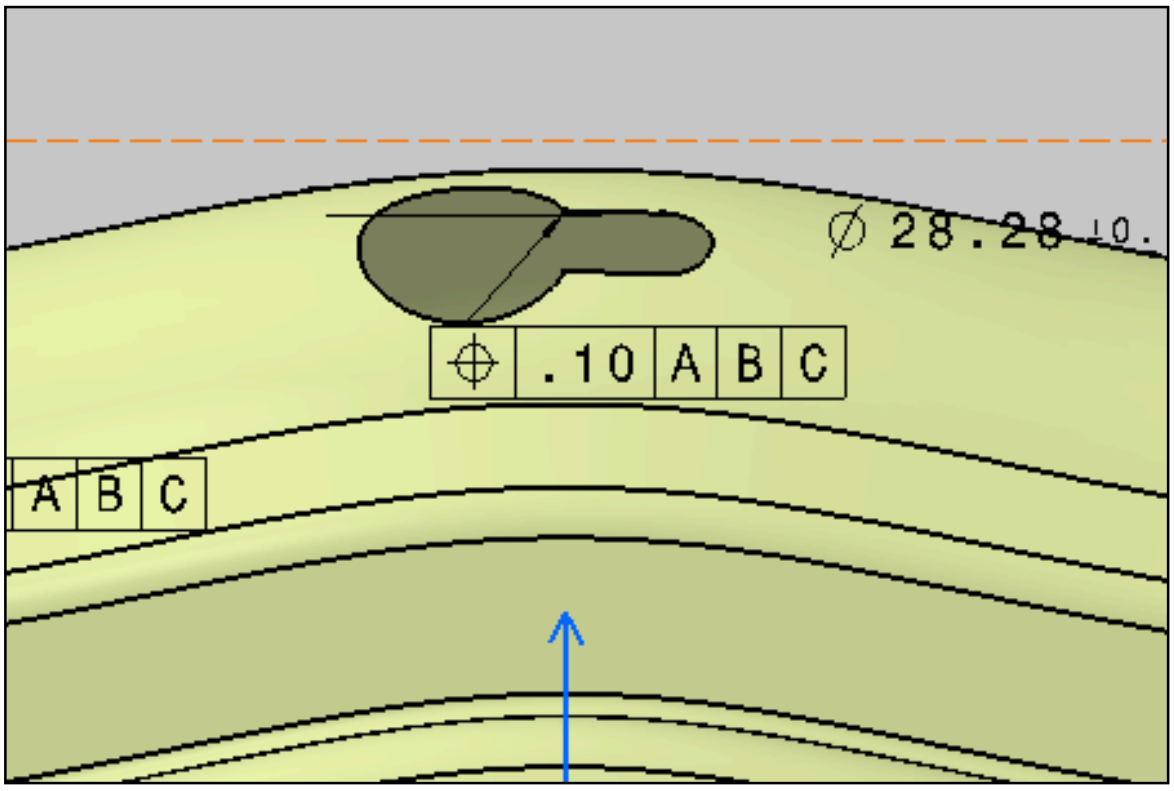


The **Semantic Tolerancing Advisor** dialog box is updated. Note the geometric feature type: **Non cylindrical Pin / Hole**.

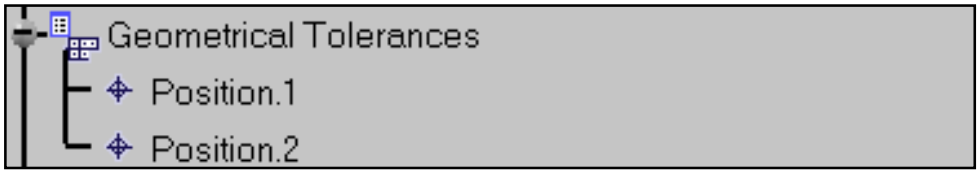
8. Click the **Position with DRF Specification** icon: 

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.

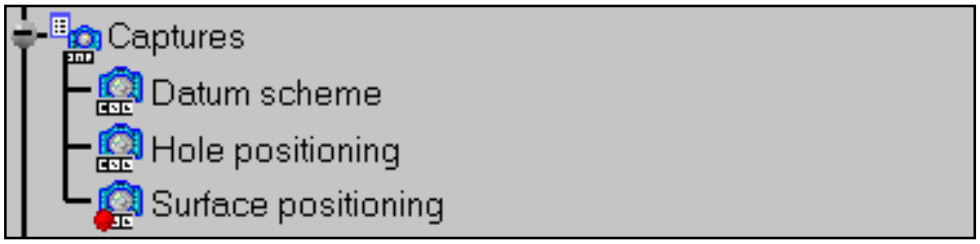
10. Do not close the **Semantic Tolerancing Advisor** dialog box to perform the next task.

Back Up Next

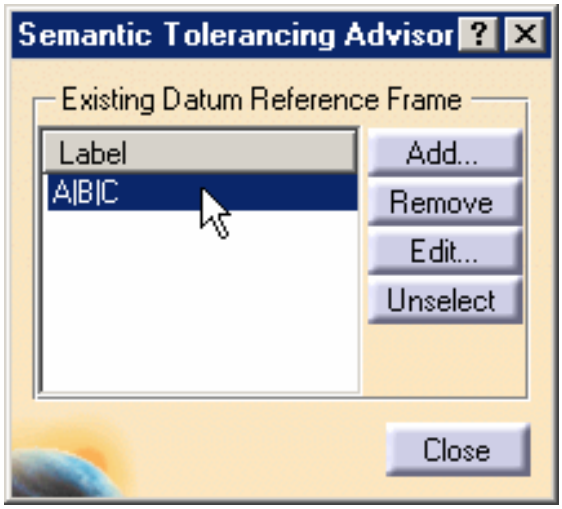
Tolerancing Body in White Surface

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

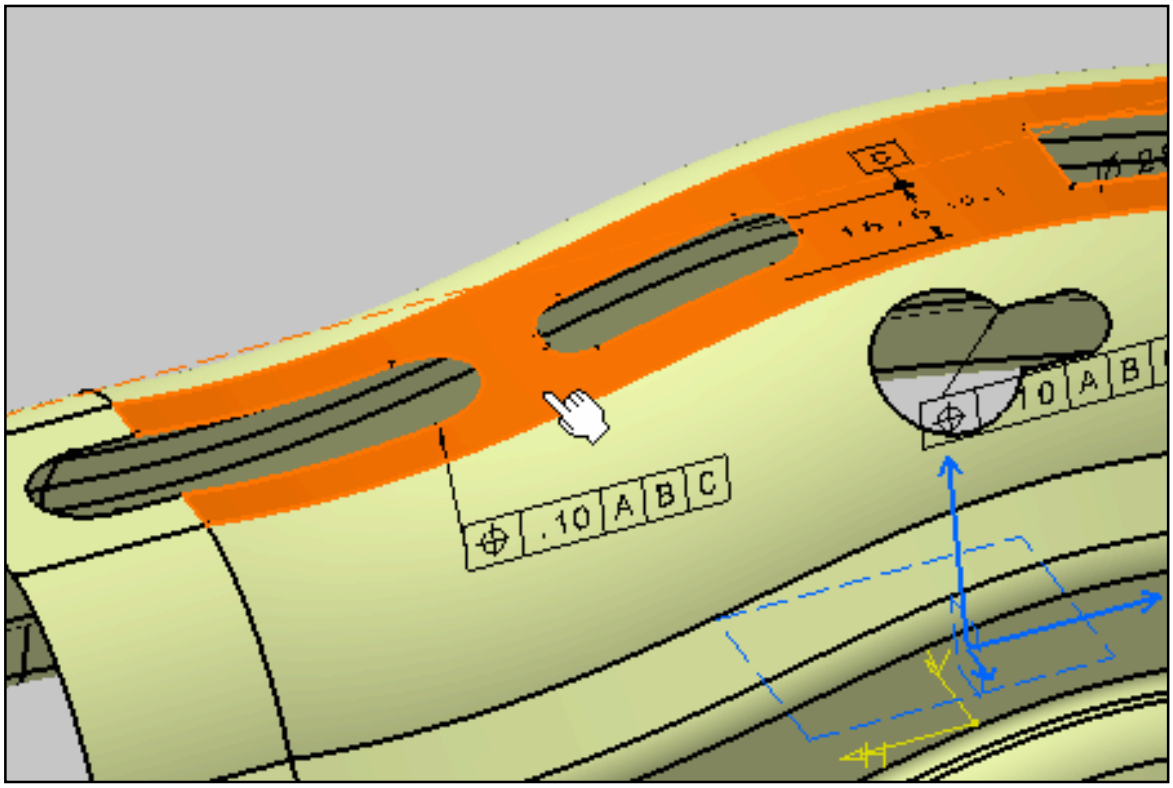
1. Right-click the **Hole positioning** capture and select **Unset Current** form the contextual menu, right-click the **Surface positioning** capture and select **Set Current** form the contextual menu.

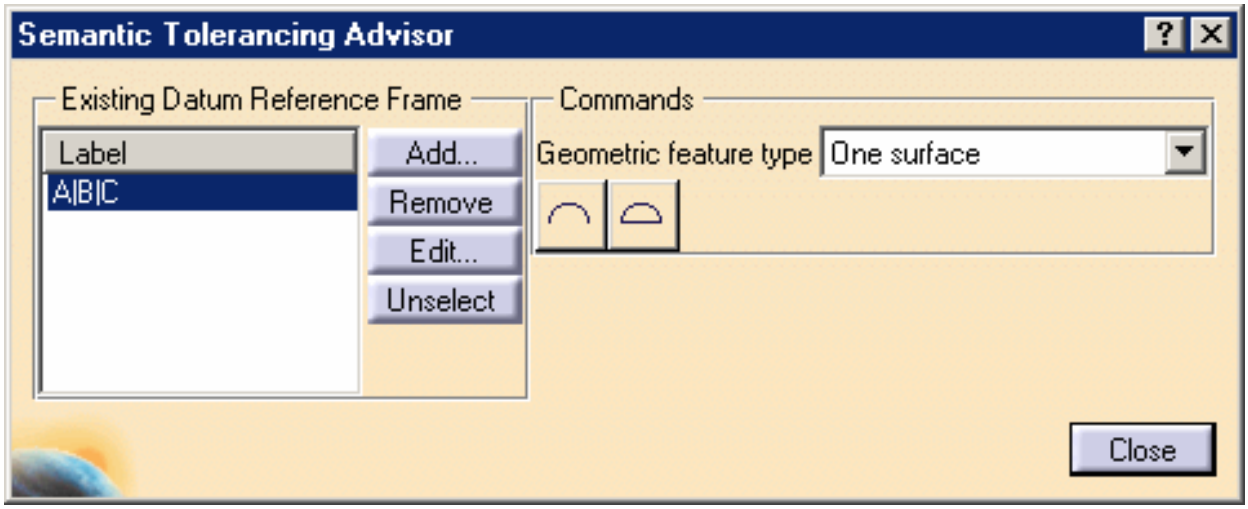


2. Select the datum reference frame in the **Semantic Tolerancing Advisor** dialog box.



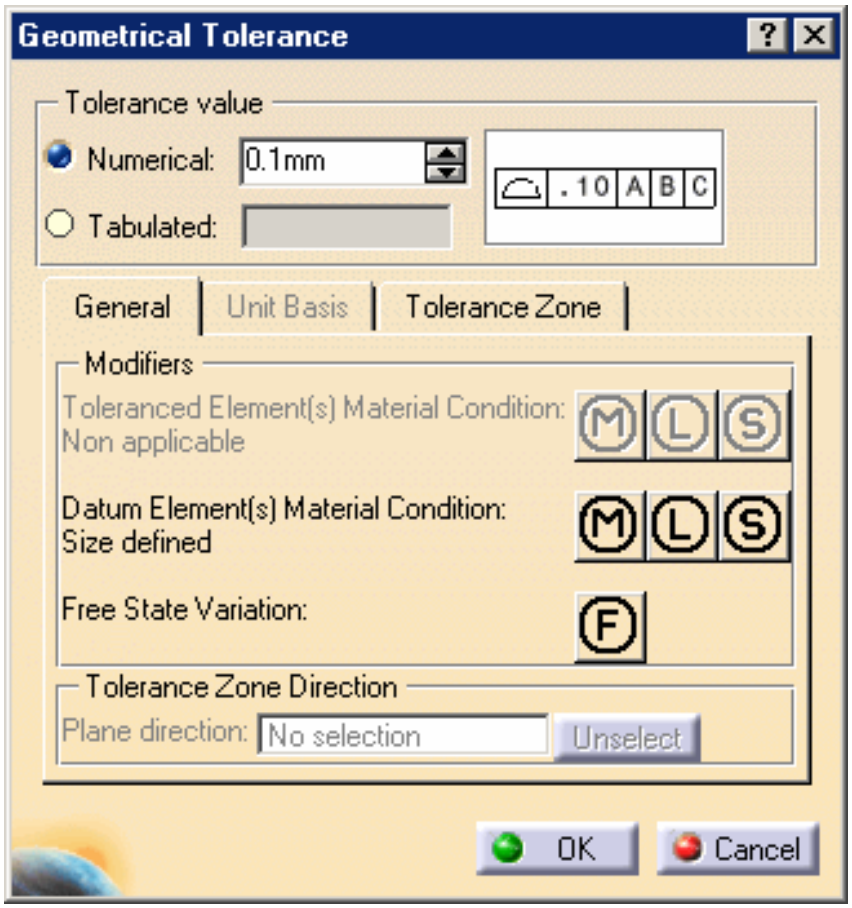
3. Select the surface as shown on the part.





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

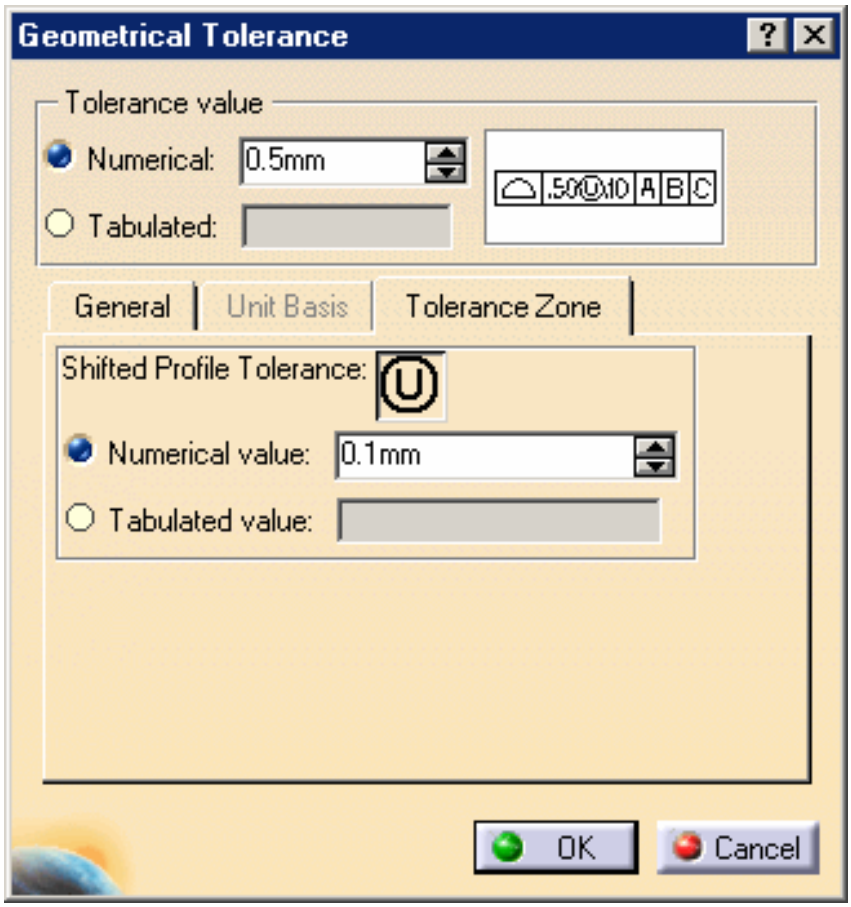
4. Click the **Profile of a Surface** icon: 



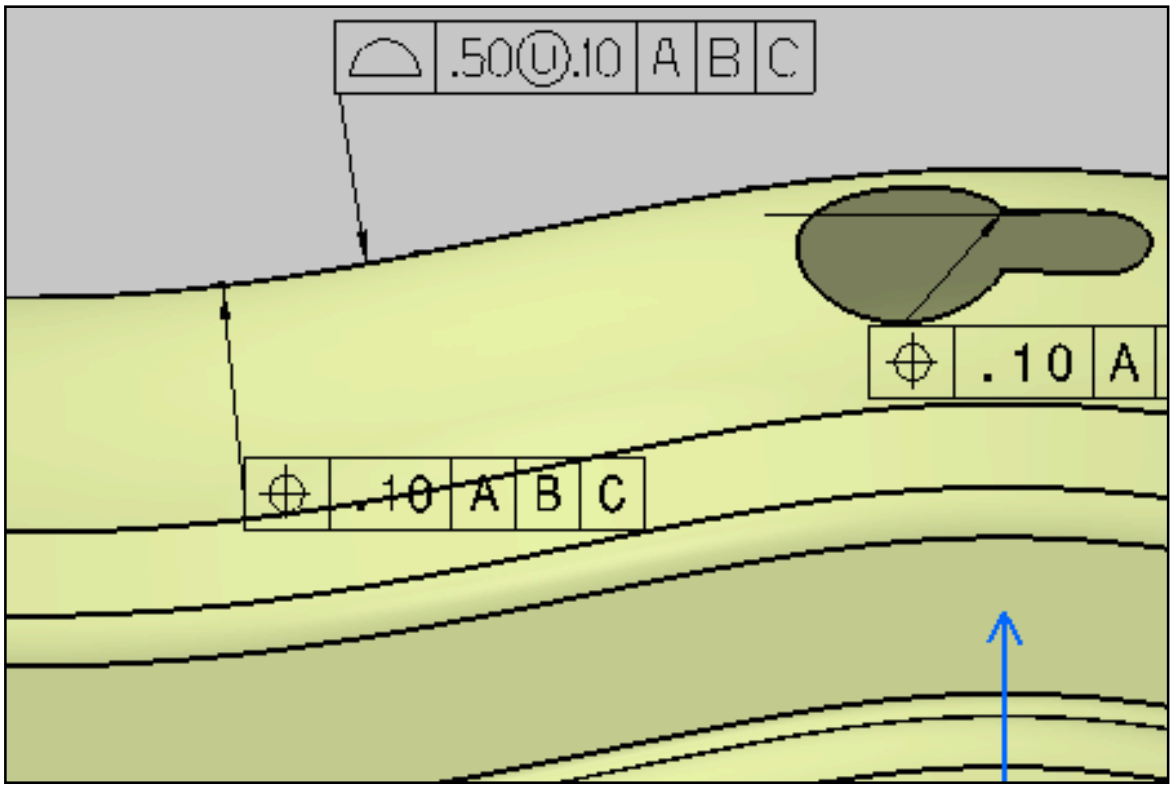
The **Geometrical Tolerance** dialog box appears.

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.

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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.

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Creating Texts

This task shows you how to create an annotation text.

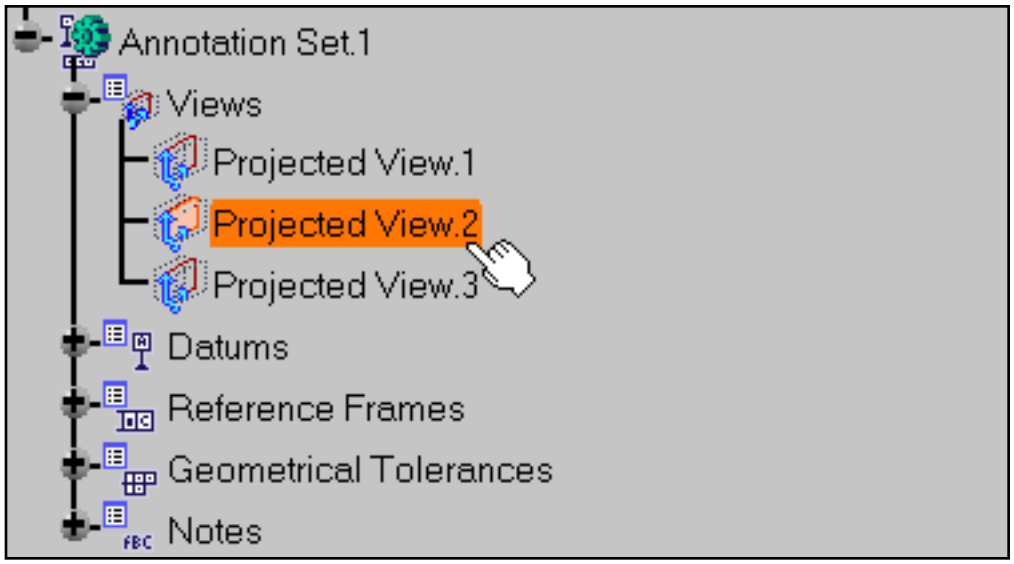
Three kinds of text may be created:

- 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 Tolerancing_Annotations_04 CATPart document.

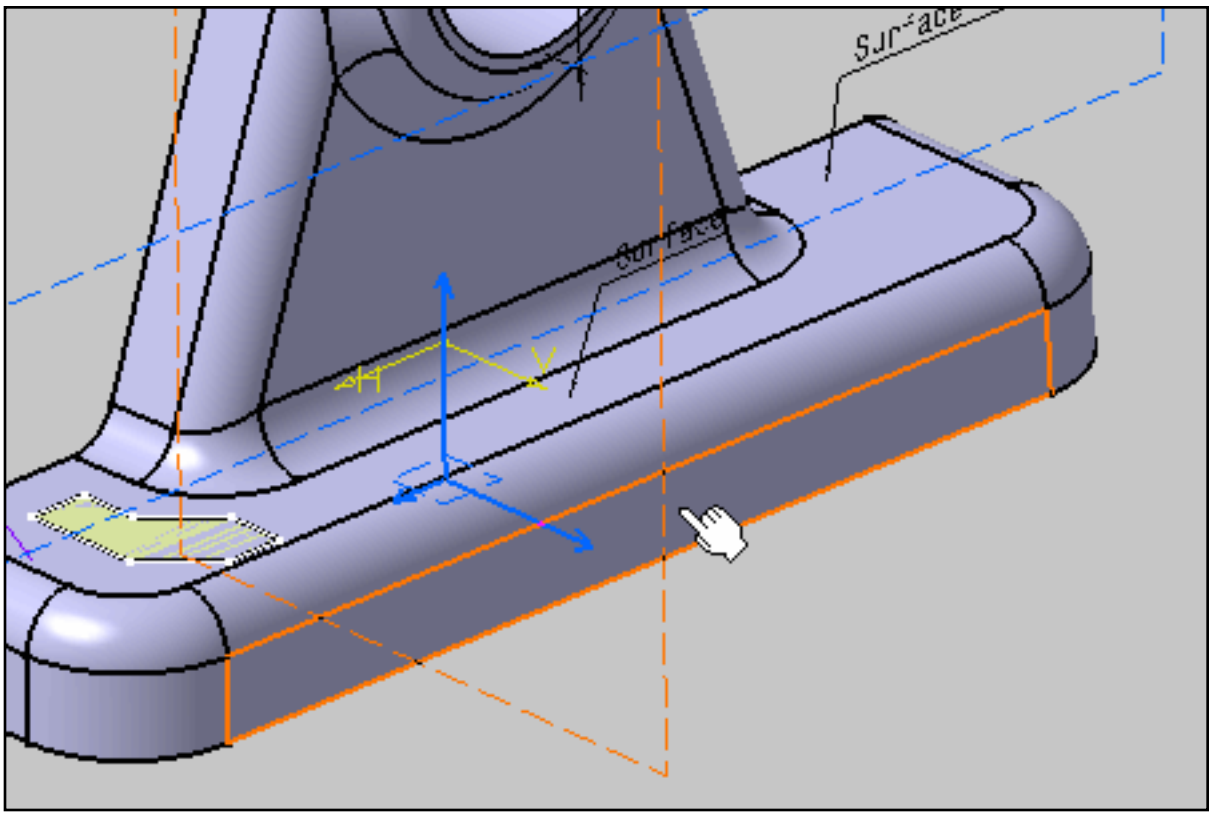
1. Activate the **Projected View.2** annotation plane.



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

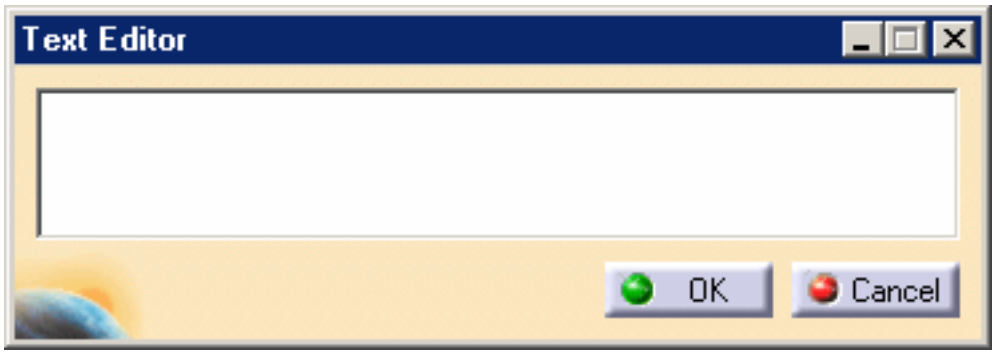
3. Select the face as shown to define a location for the arrow end of the leader.

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.

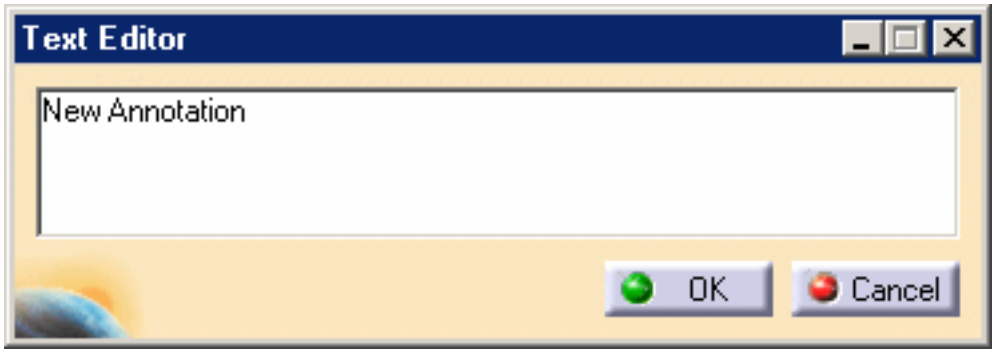


If the active view is not valid, a message appears informing you that you cannot use the active view.
This means that the annotation will be displayed in an annotation plane normal to the selected face.
For more information, see View/Annotation Planes.

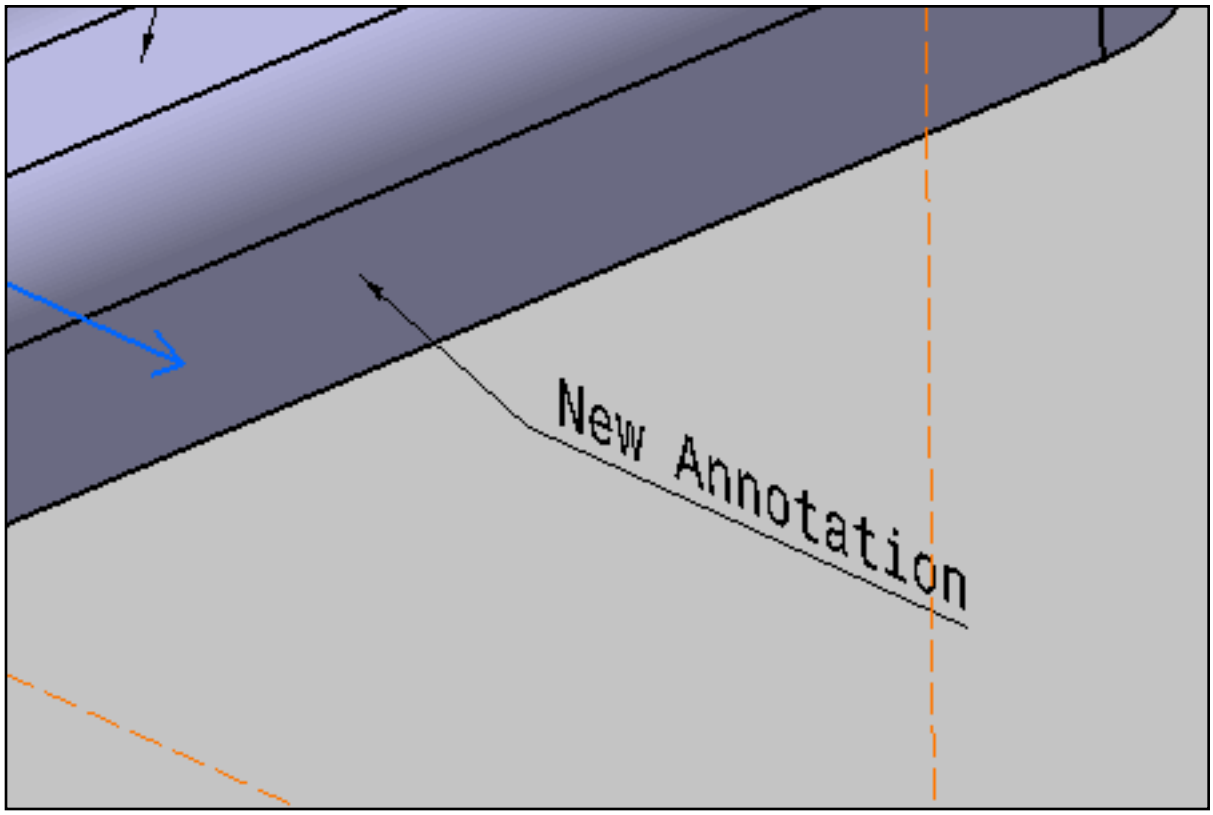
The **Text Editor** dialog box appears.



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.

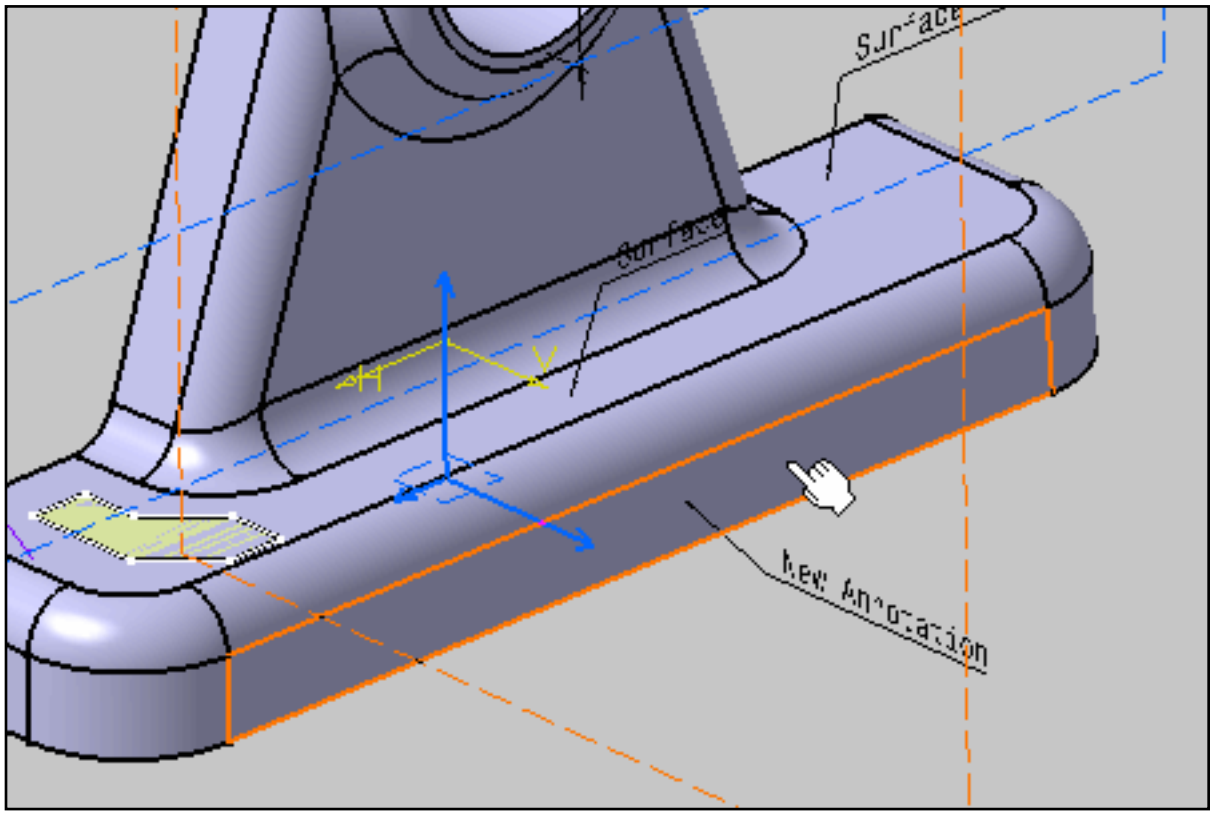
The leader is associated with the element you selected. If you move either the text or the element, the leader stretches to maintain its association with the element.

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.

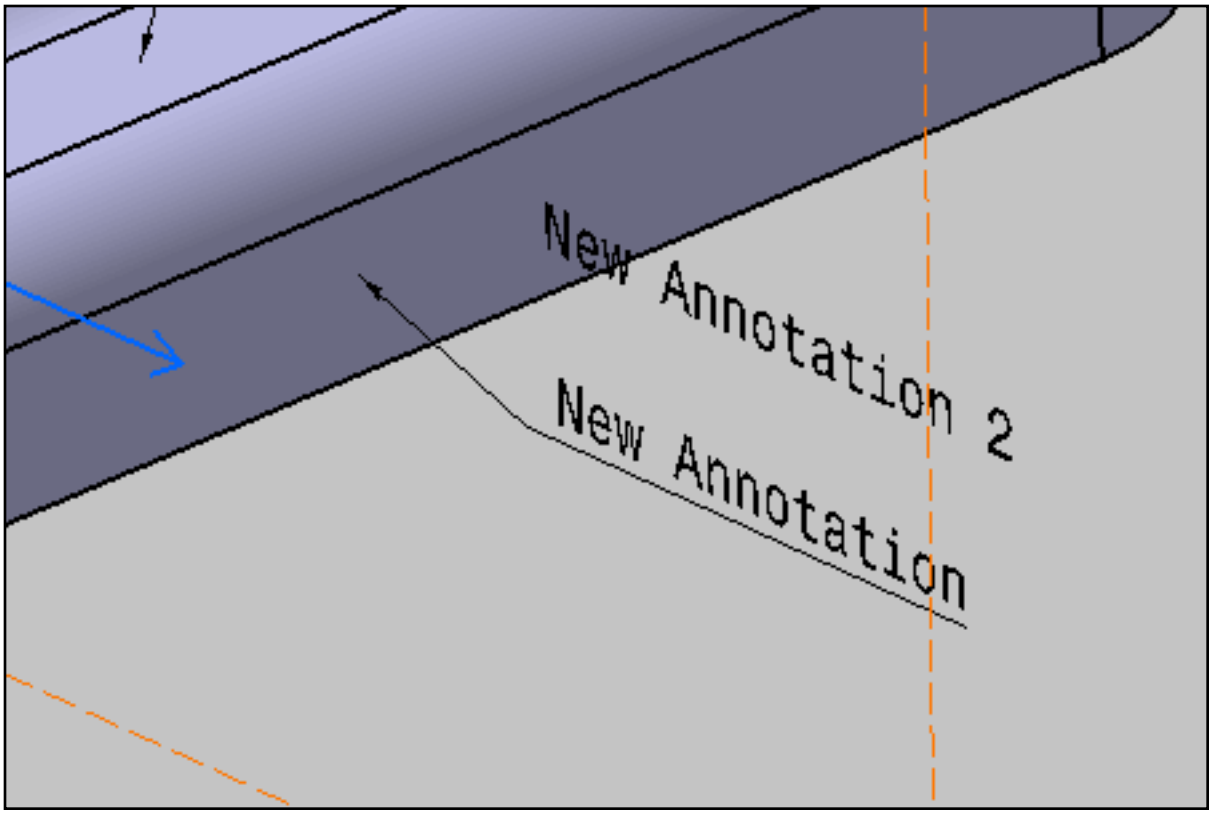
6. Click the **Text** icon: 

7. Select the face as shown.



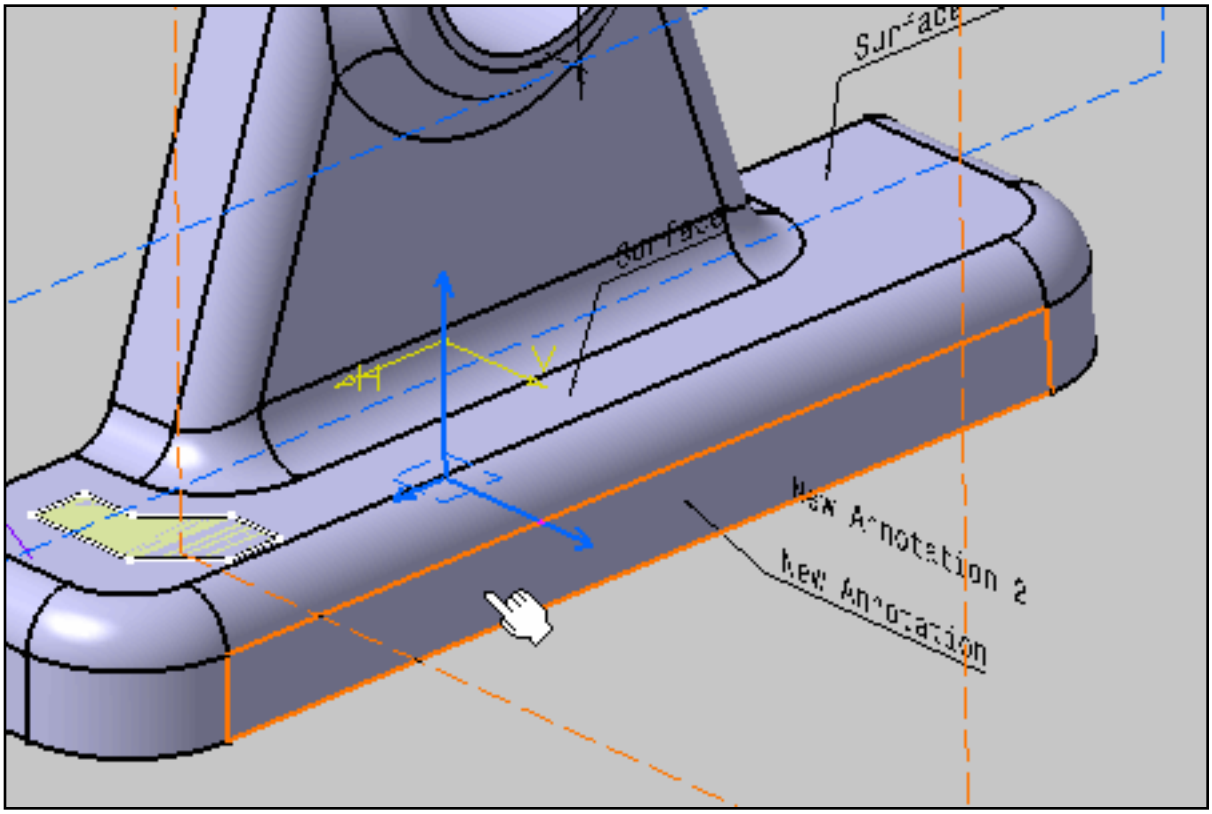
8. Enter your text, for example "New Annotation 2" in the dialog box and click **OK**.

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



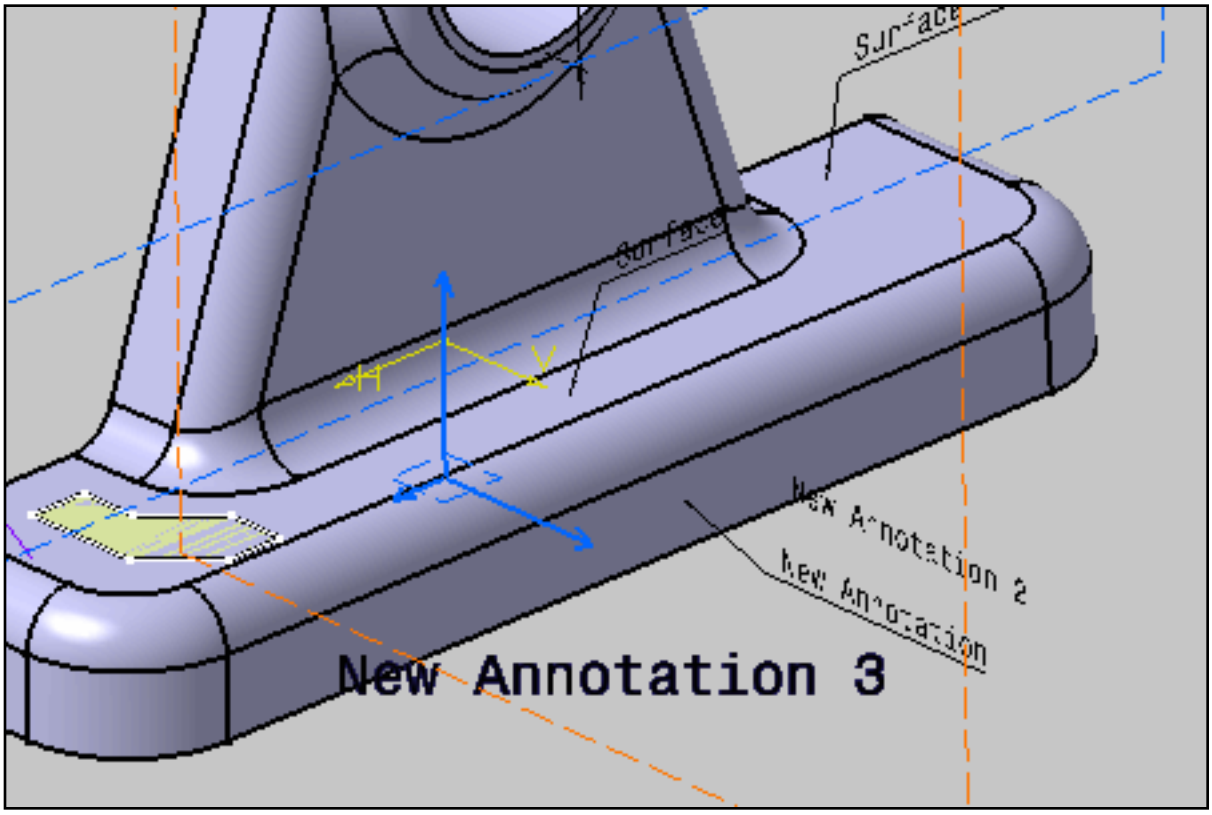
9. Click the **Text Parallel to Screen** icon: 

10. Select the face as shown.



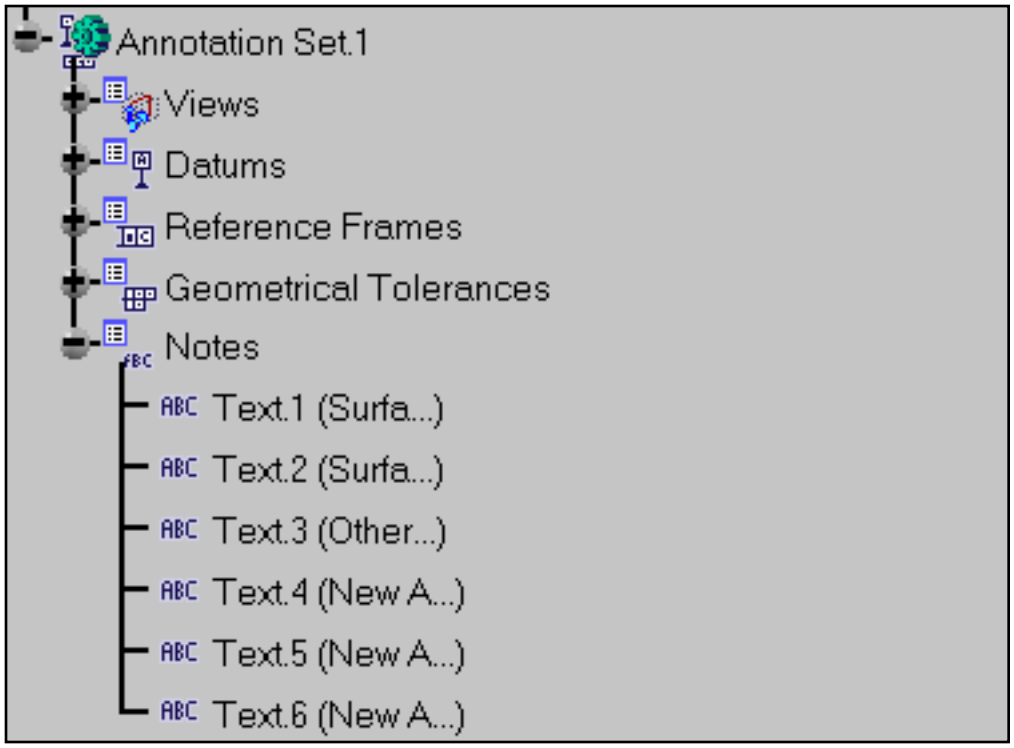
11. Enter your text, for example "New Annotation 3" in the dialog box and click **OK**.

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.

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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.

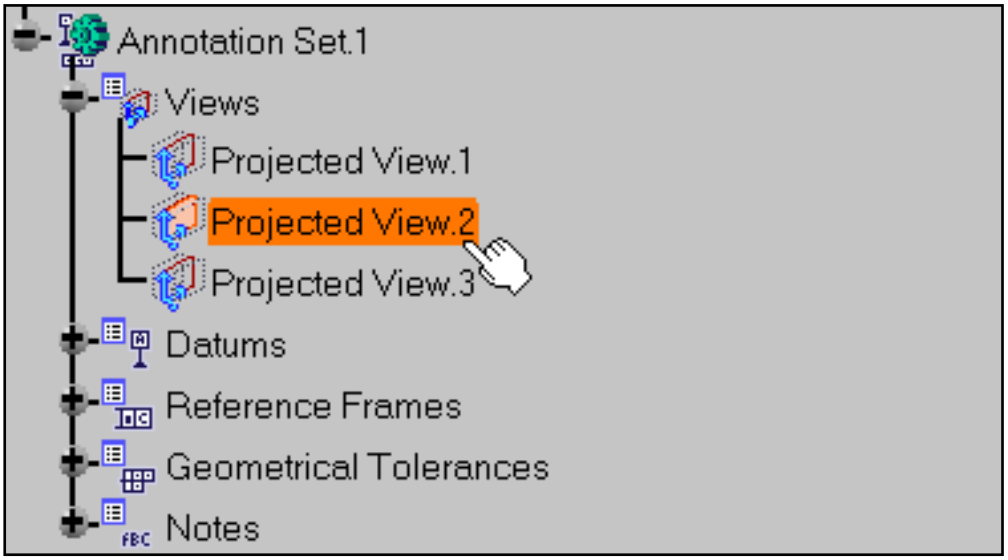
Two kinds of flag note may be created:

- 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 Tolerancing_Annotations_04 CATPart document.

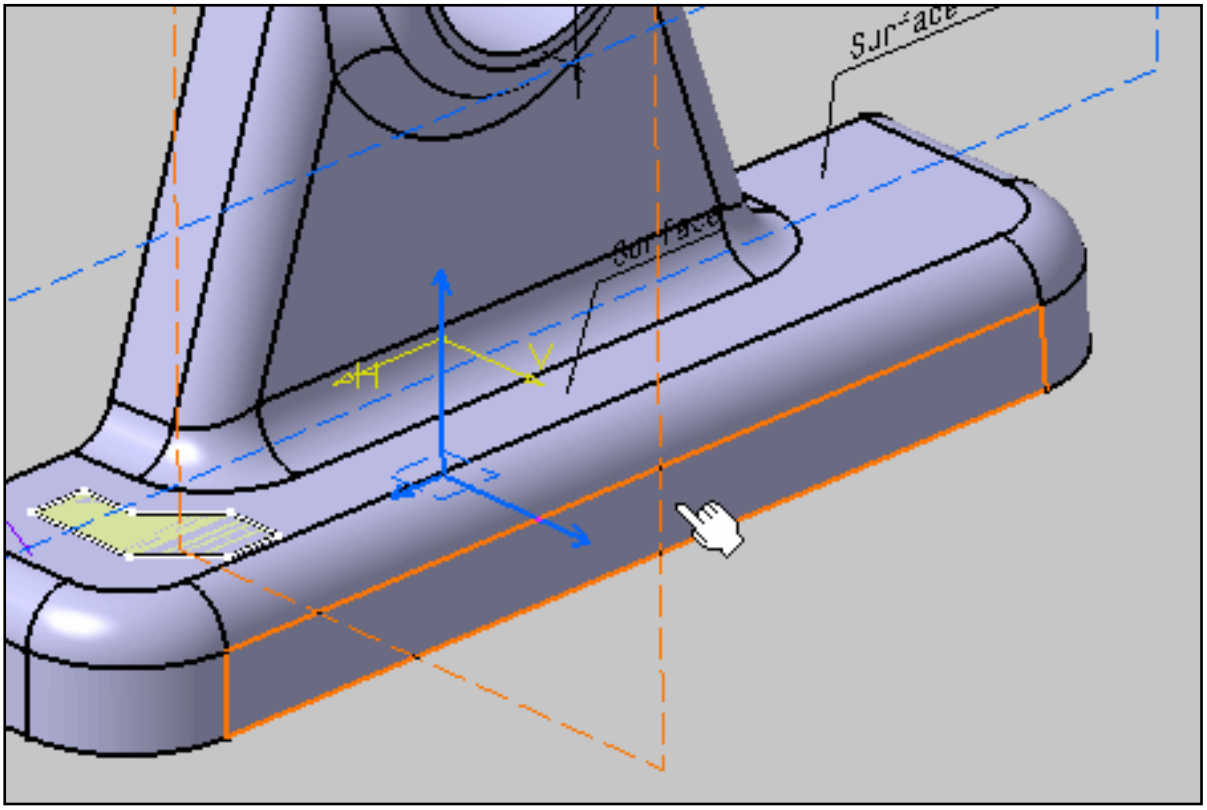
1. Activate the **Projected View.2** annotation plane.



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

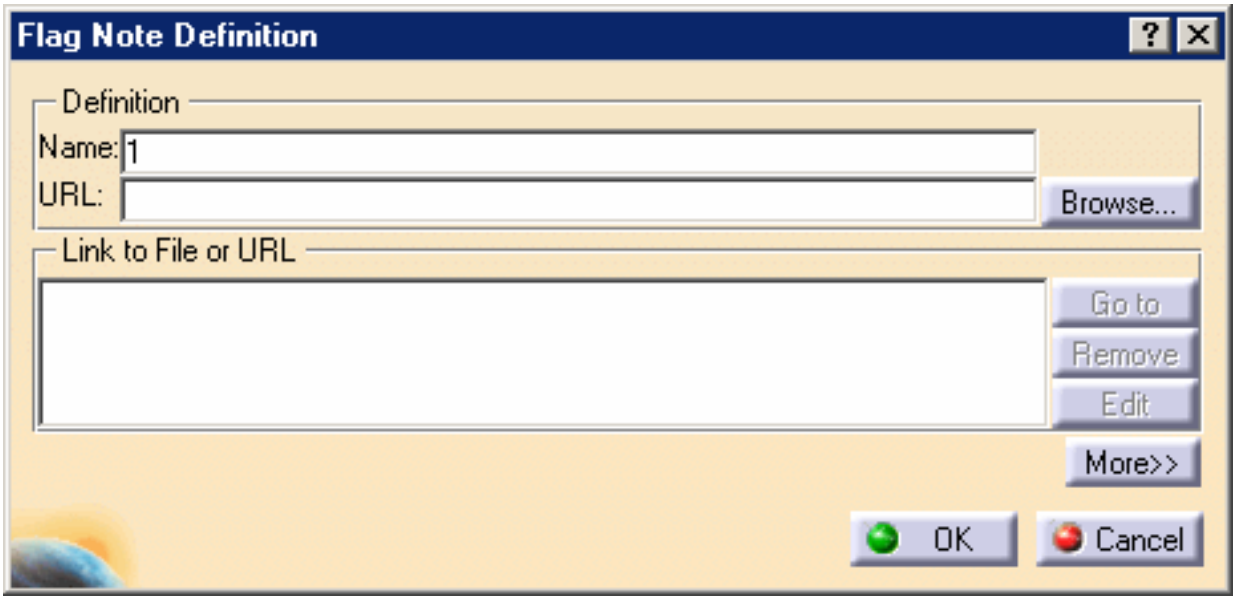
3. Select the face as shown to define a location for the arrow end of the leader.

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.

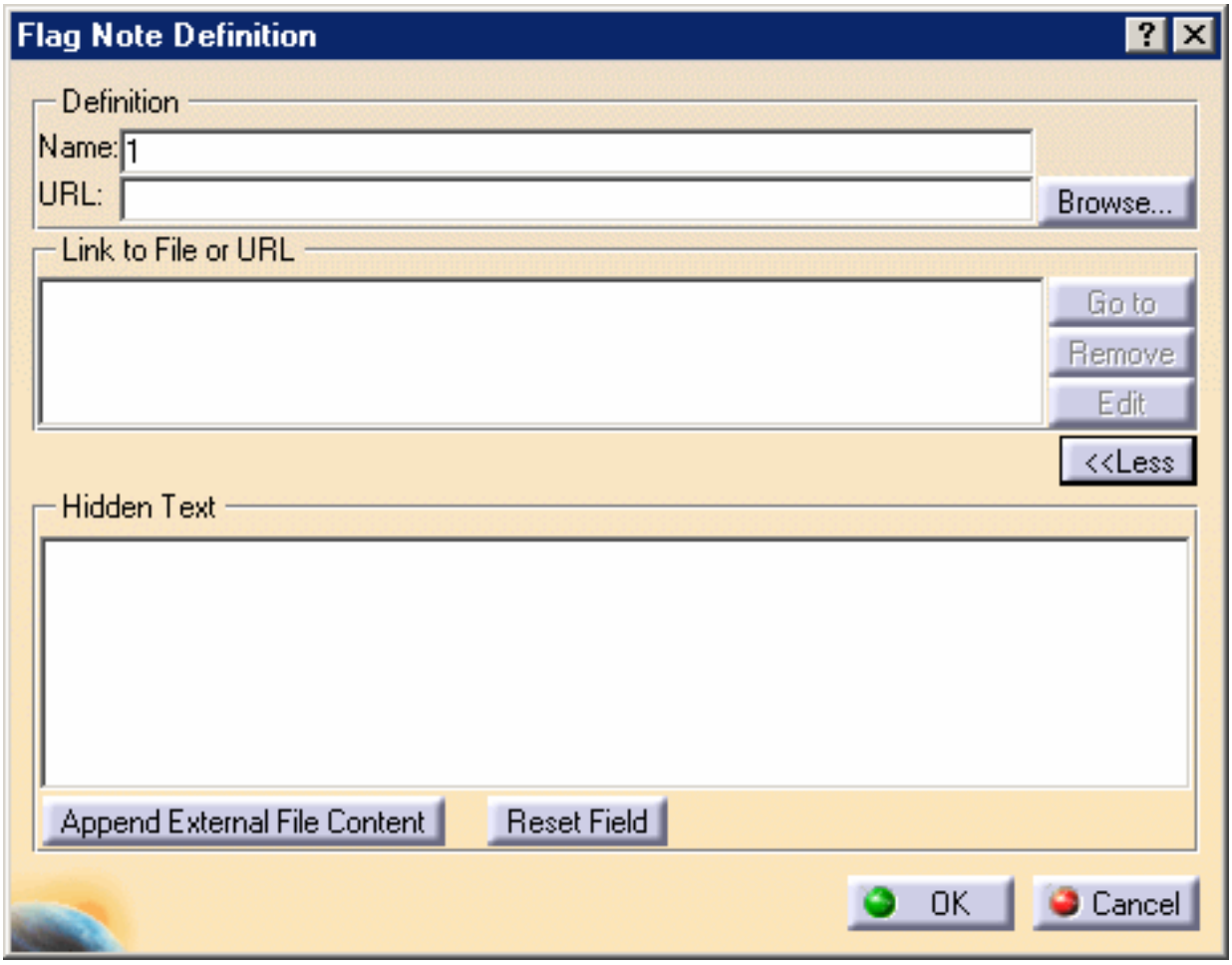
To remove one of them, select it and click the **Remove** button.

To edit one of them, select it and click the **Edit** button.

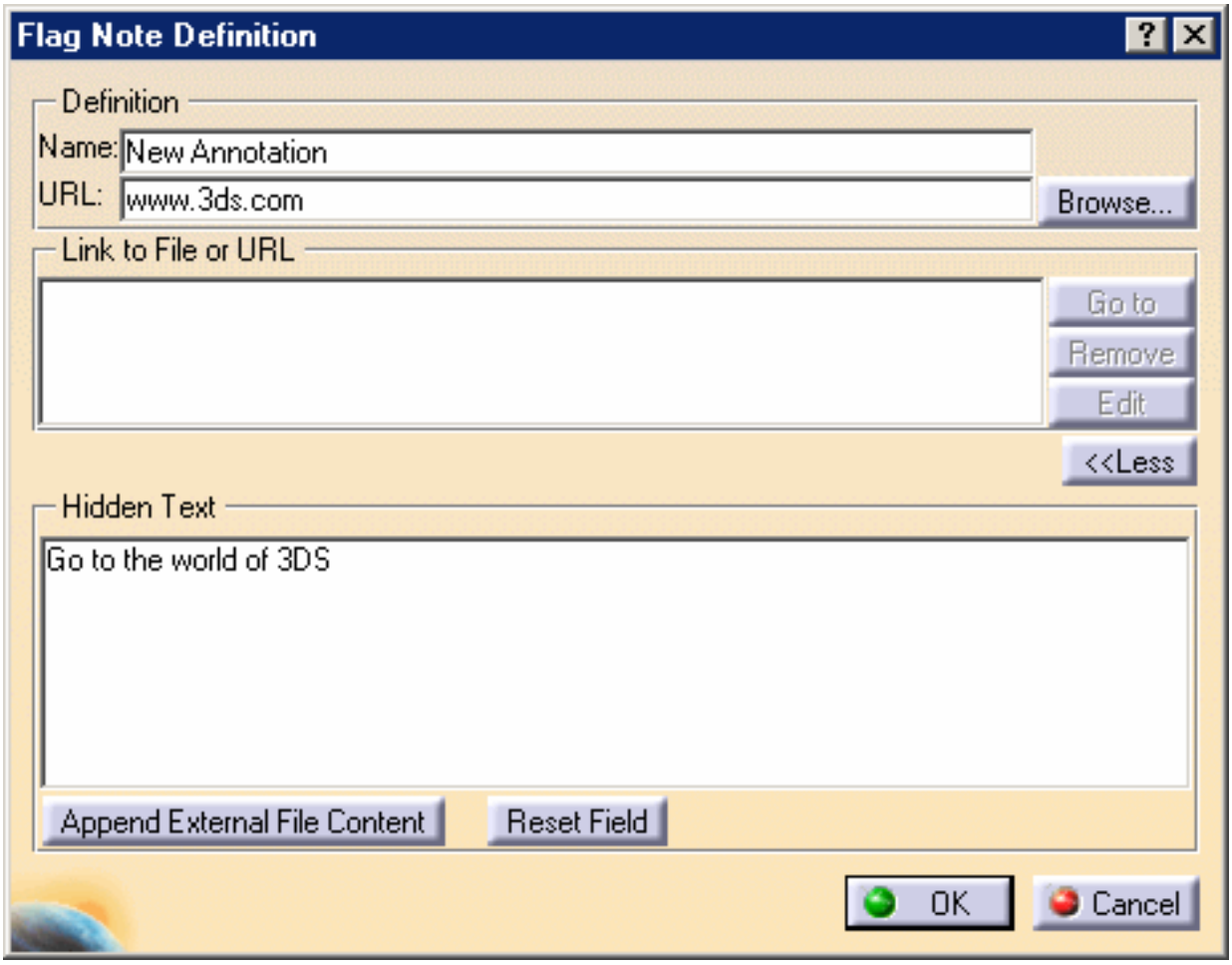
Clicking the **More>>** button lets you define a hidden text which is displayed with the list of links in a tooltip when the mouse pointer stays on the flagnote.

You can write your text or import it from a text file.

You can leave the field blank.

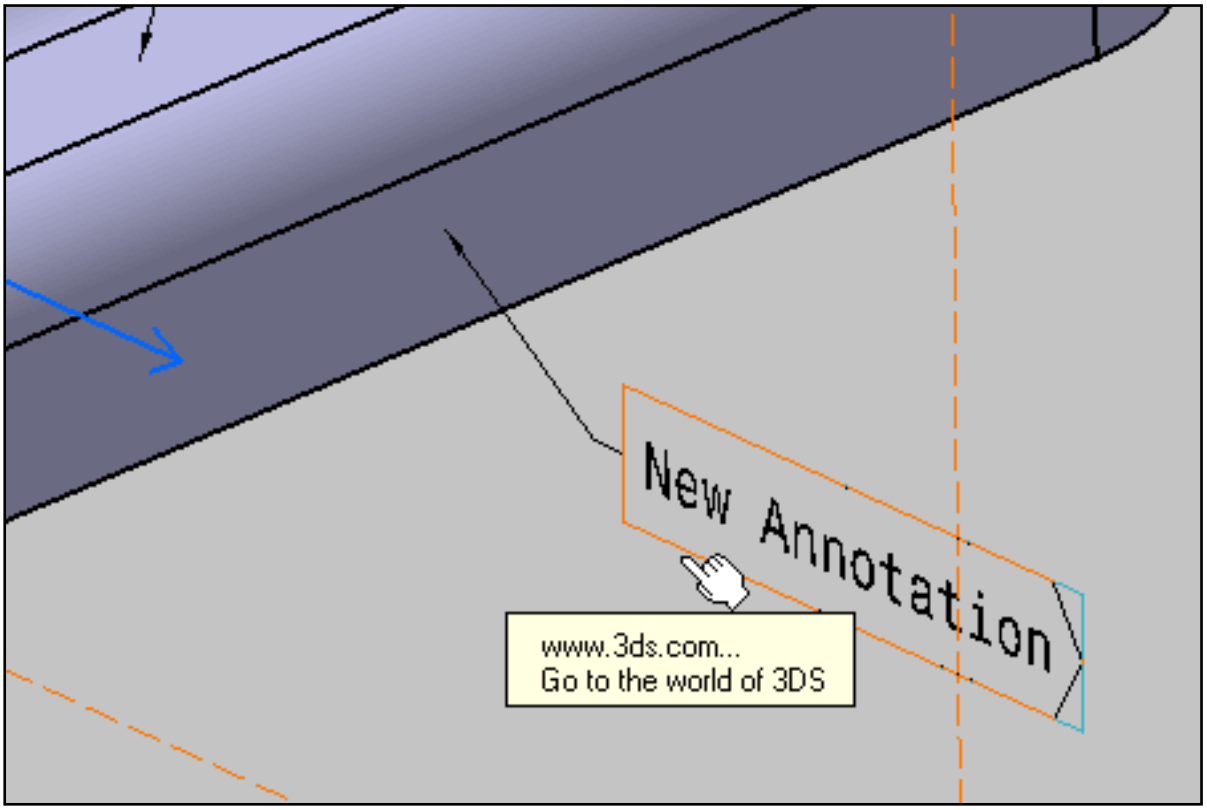


4. Define the flag note:
   - the name: *New Annotation*
   - a link: *www.3ds.com*
   - a hidden text: *Go to the world of 3DS*



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.



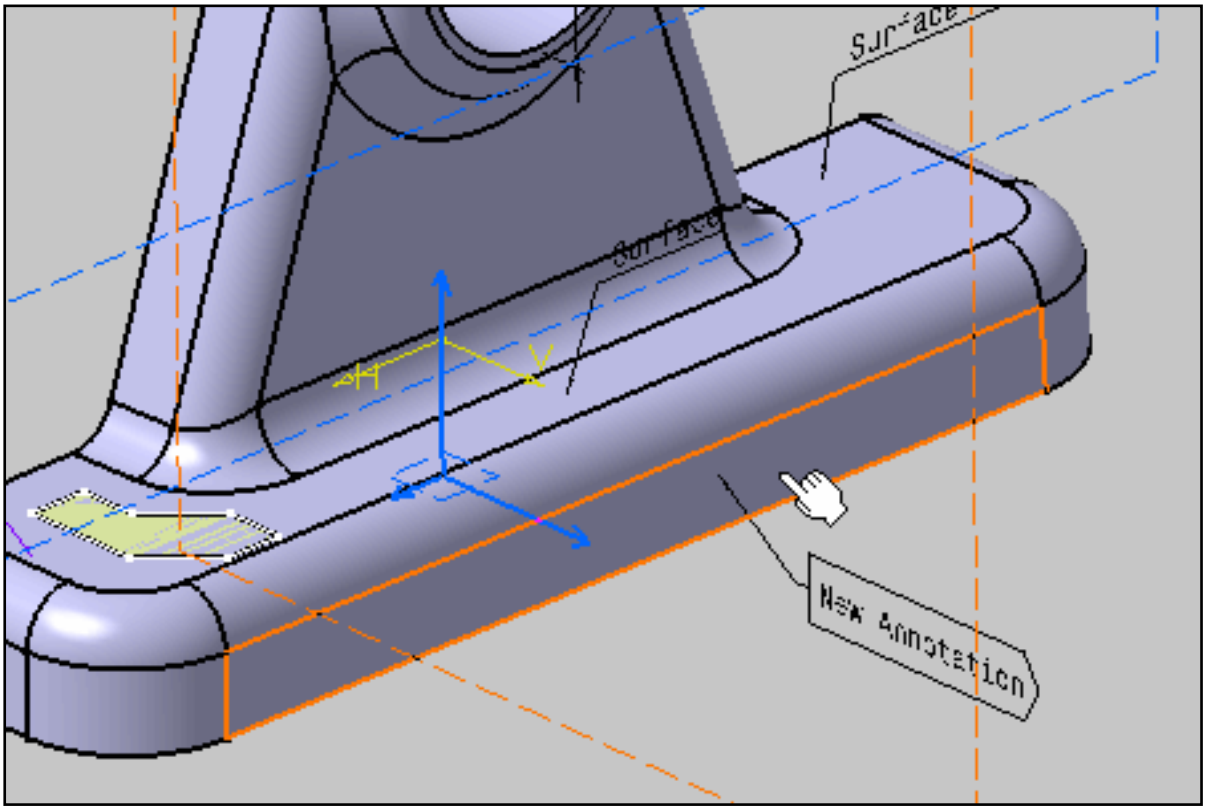
The leader is associated with the element you selected. If you move either the text or the element, the leader stretches to maintain its association with the element.

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.

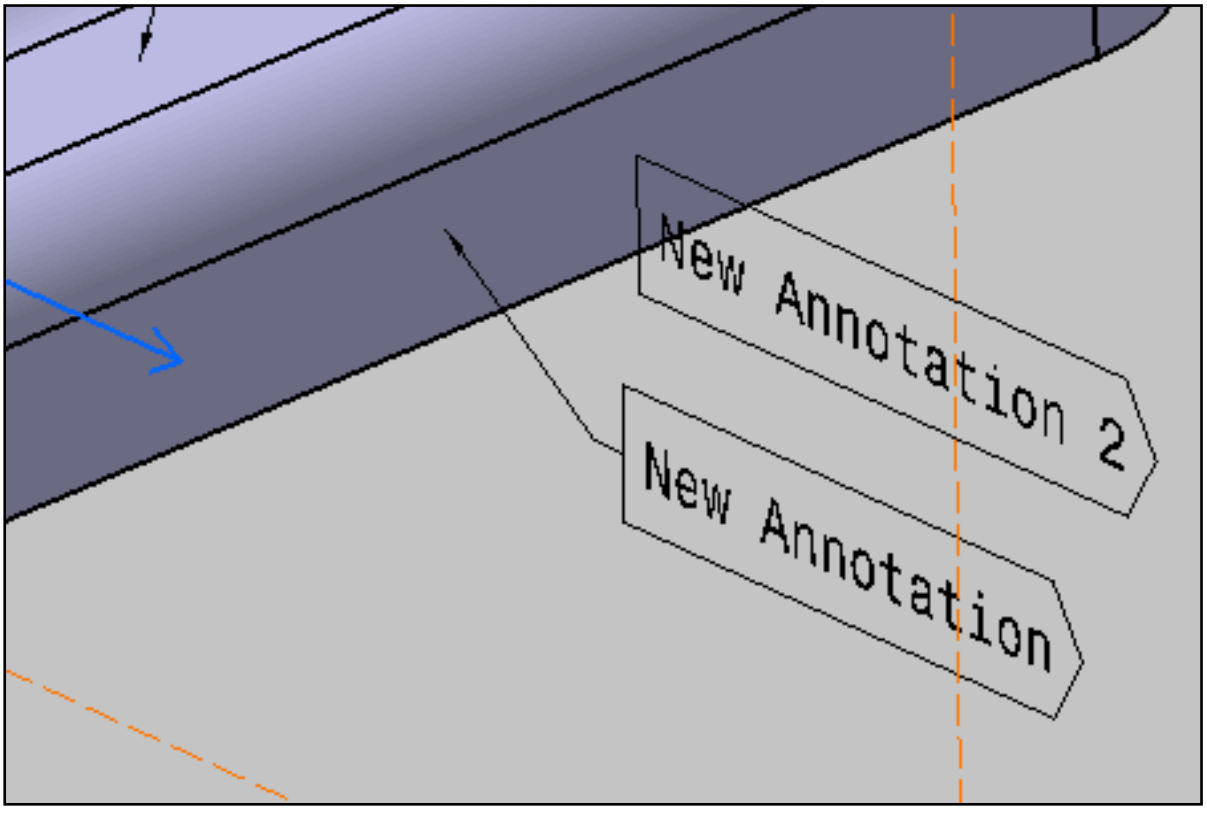
6. Click the **Flag Note** icon: 

7. Select the face as shown.

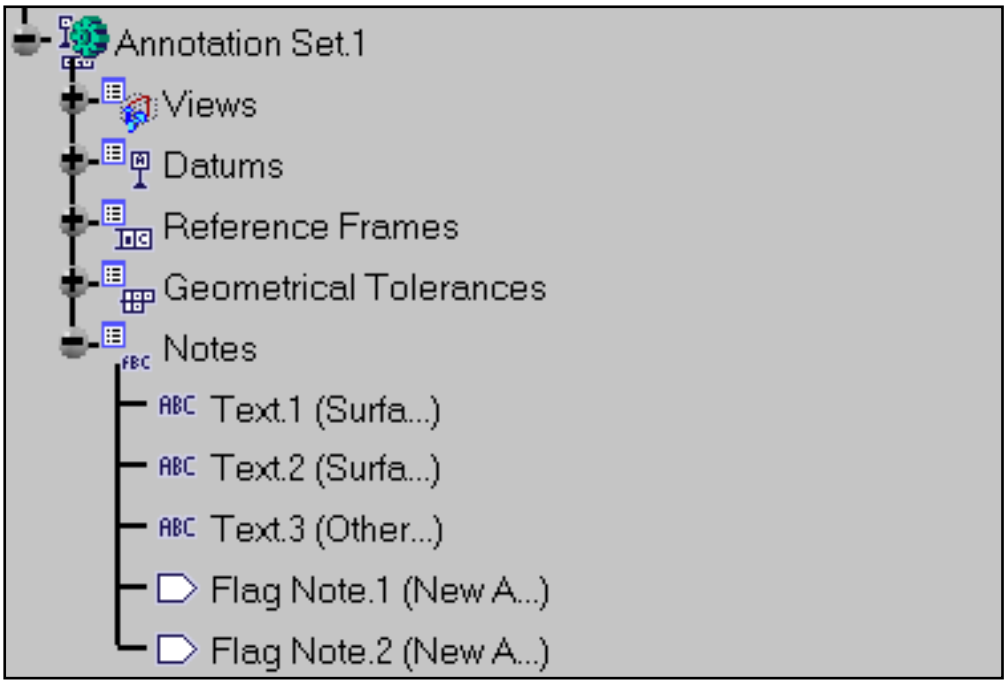


8. Enter your flag note name, for example "New Annotation 2" in the dialog box, specify a link and click **OK**.

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.

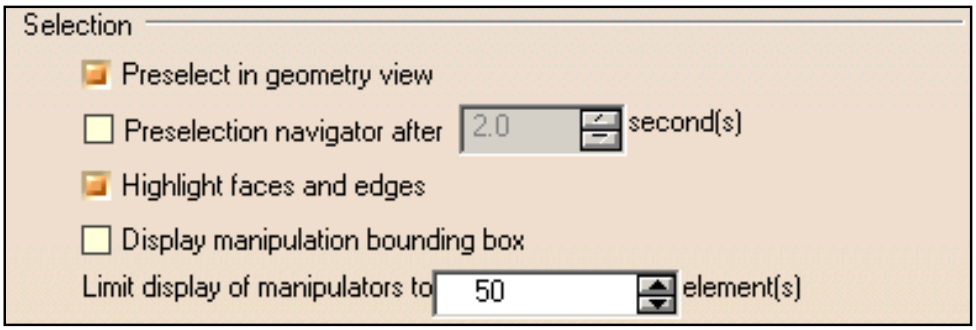
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Adding an Attribute Link

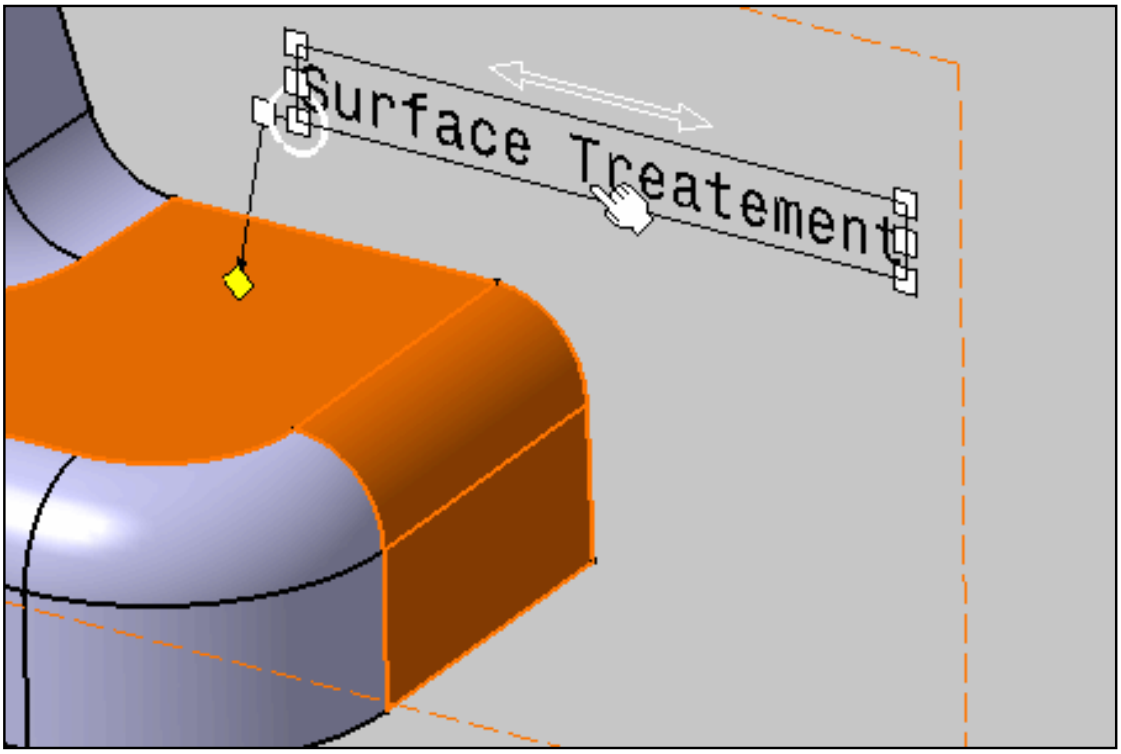
This task shows you how to add an attribute link parameter to a text while you are creating this annotation.
Note that the operating mode described here is valid for Text or a Flag Note.

Open the Tolerancing_Annotations_04 CATPart document.

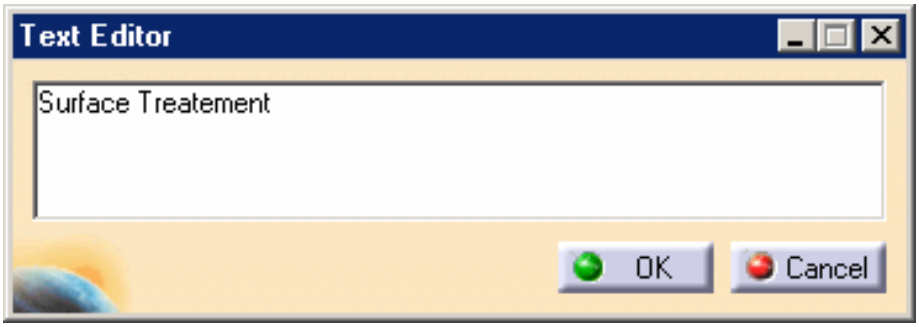
To improve the highlight of the geometry selection, select the **Tools->Options...** command. In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



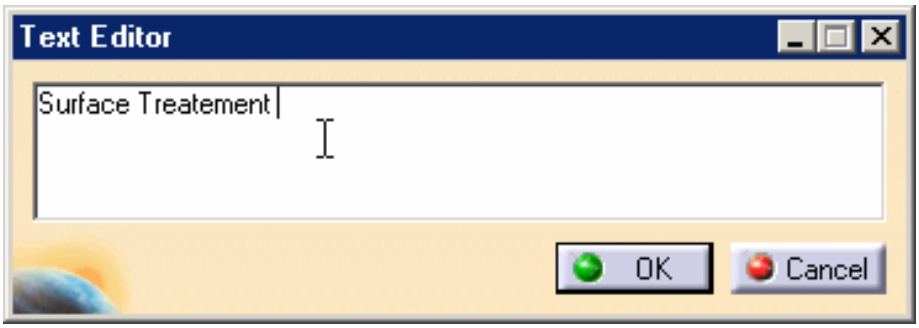
1. Double-click the annotation text to edit it.



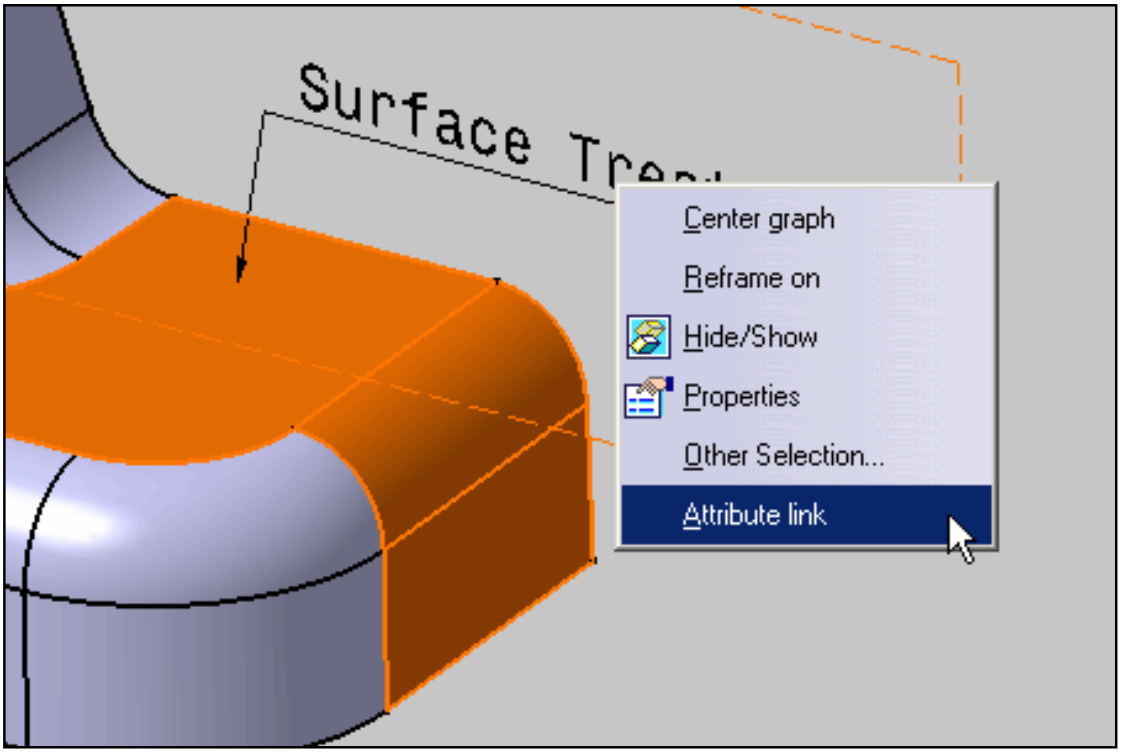
The **Text Editor** dialog box appears.



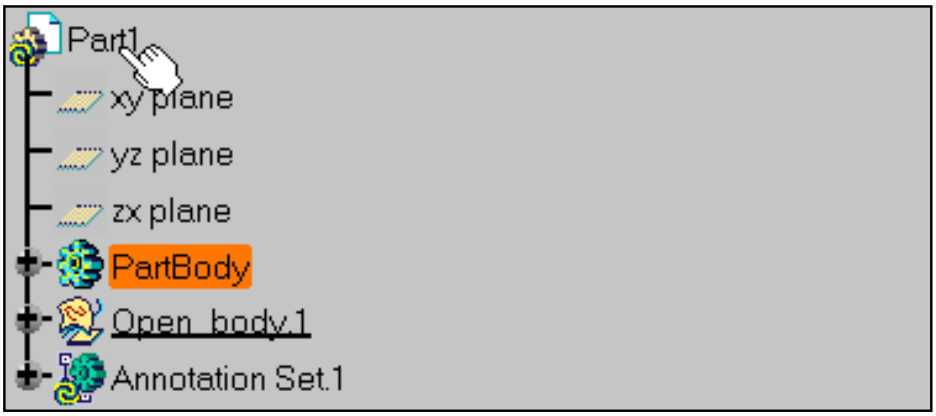
2. Take care to un-select the text into the **Text Editor** dialog box and put the cursor after the white space at the end of the text.



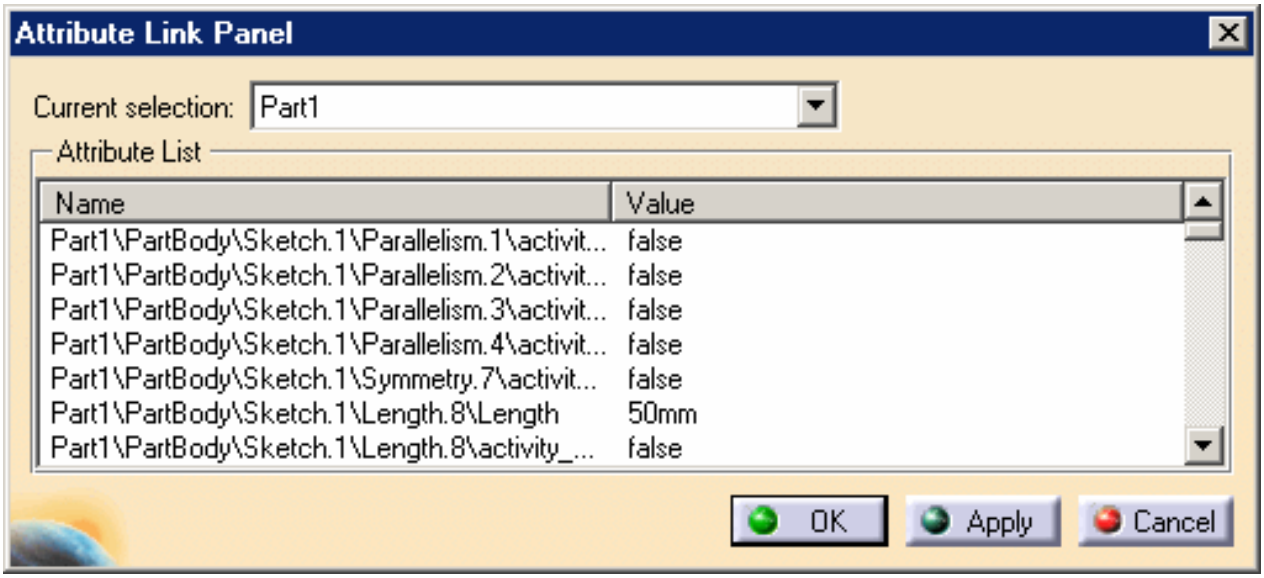
3. Right-click the annotation in the geometry area and select the **Attribute Link** contextual command (the annotation is not highlighted!).



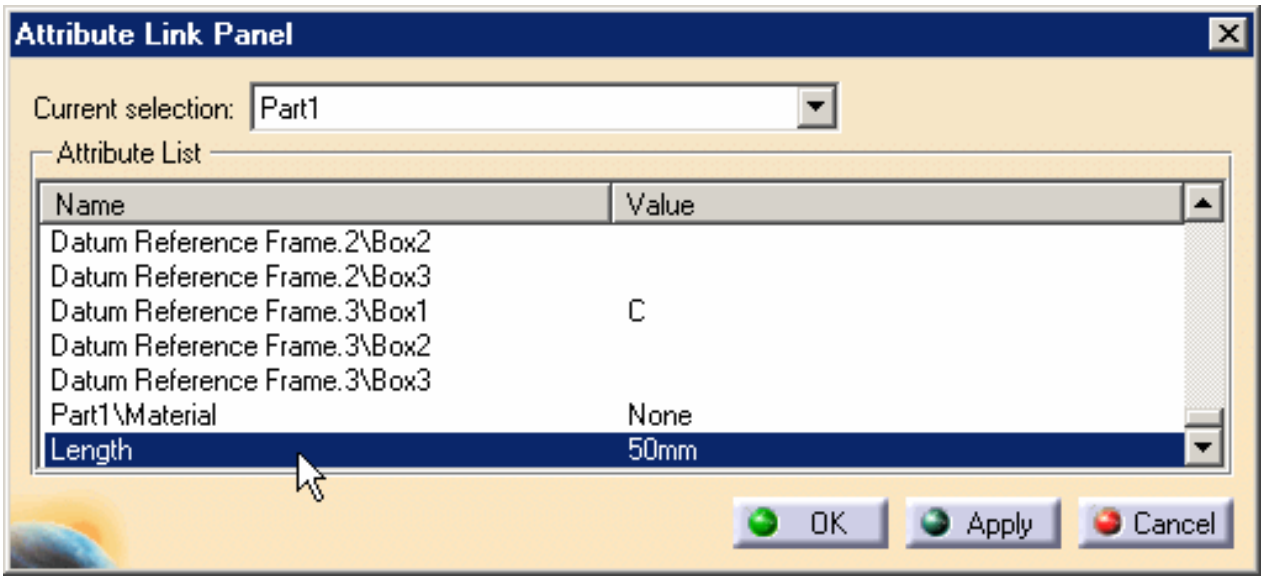
4. Select **Part 1** in the Specification Tree to access all parameters defined for the part.



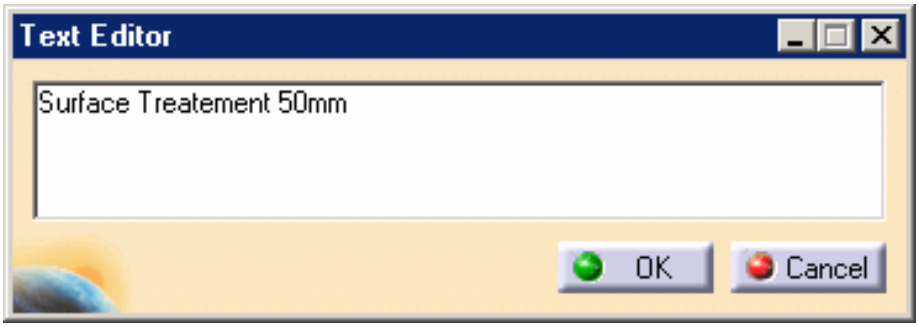
The **Attribute Link** dialog box appears.



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



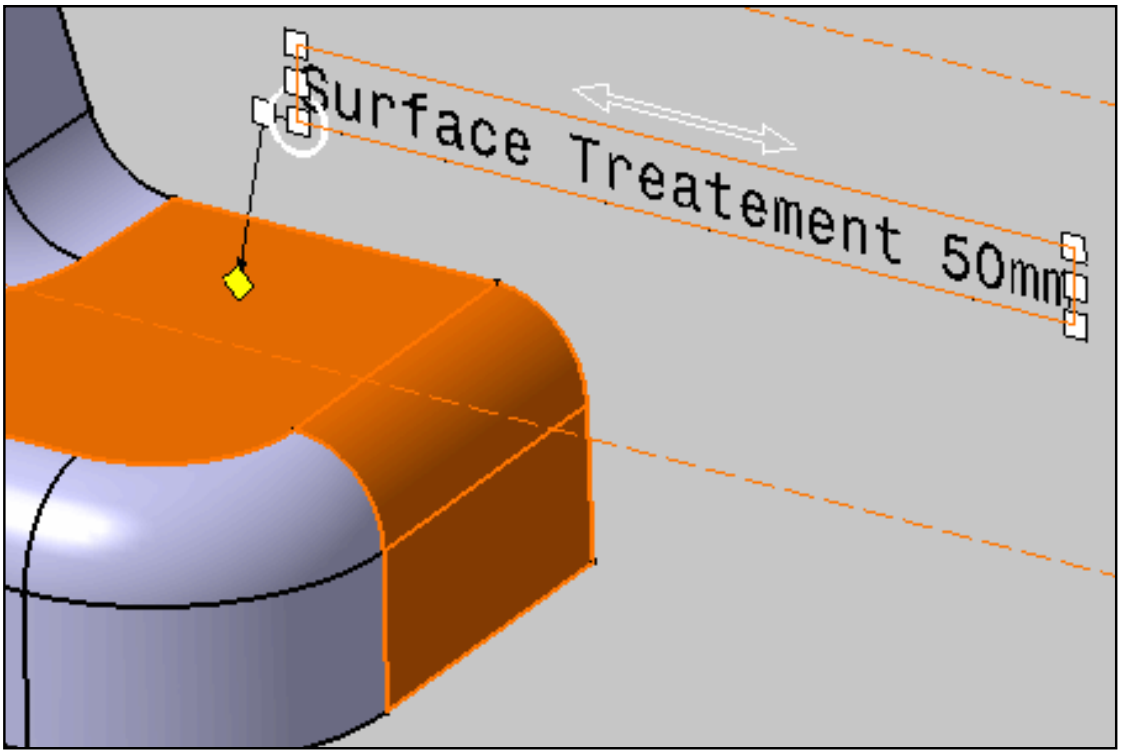
6. Click **OK** to close the **Attribute Link** dialog box.



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.

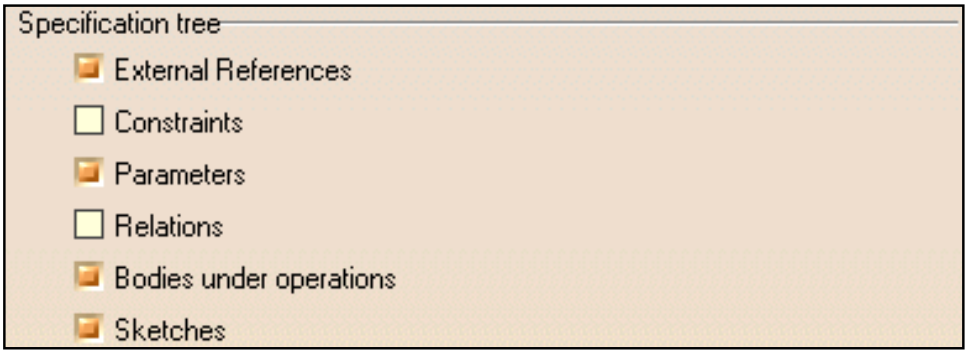
7. Click **OK** to confirm the operation and close the dialog box.

The parameter is attached to the textual annotation.

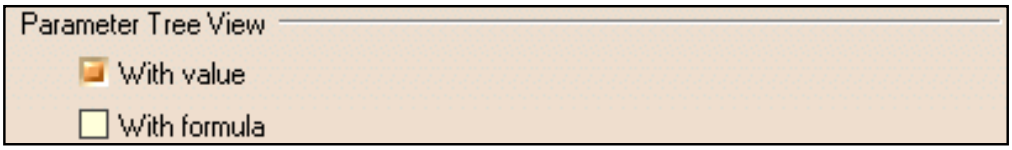


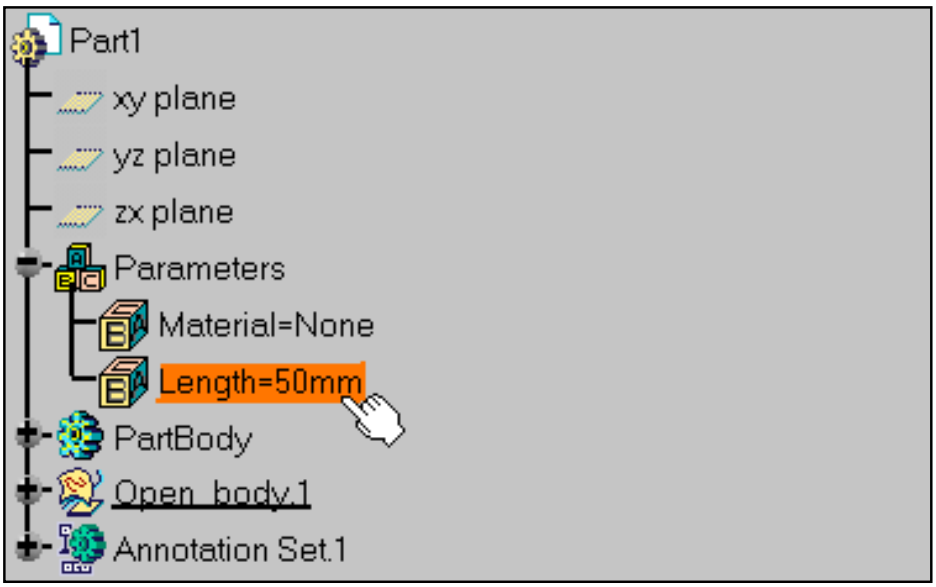
8. Double-click **Length** to edit the parameter: enter 70 mm in the **Edit Parameter** dialog box which appears and click **OK**.

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**.

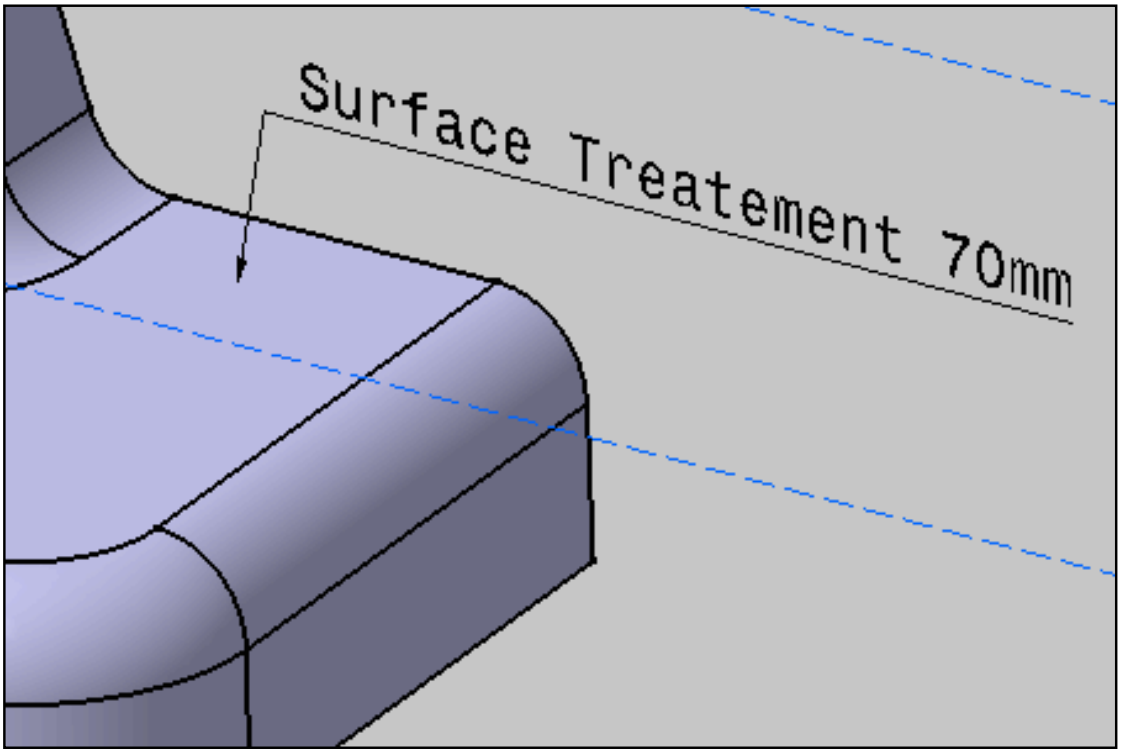






9. Un-select the annotation.

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.

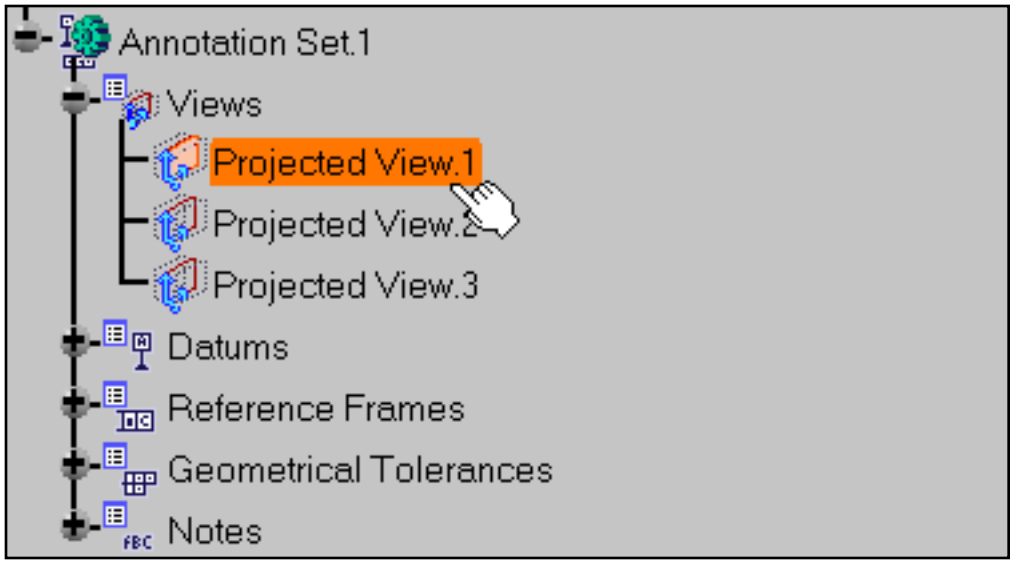
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Creating Datum

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

Open the Tolerancing_Annotations_04 CATPart document.

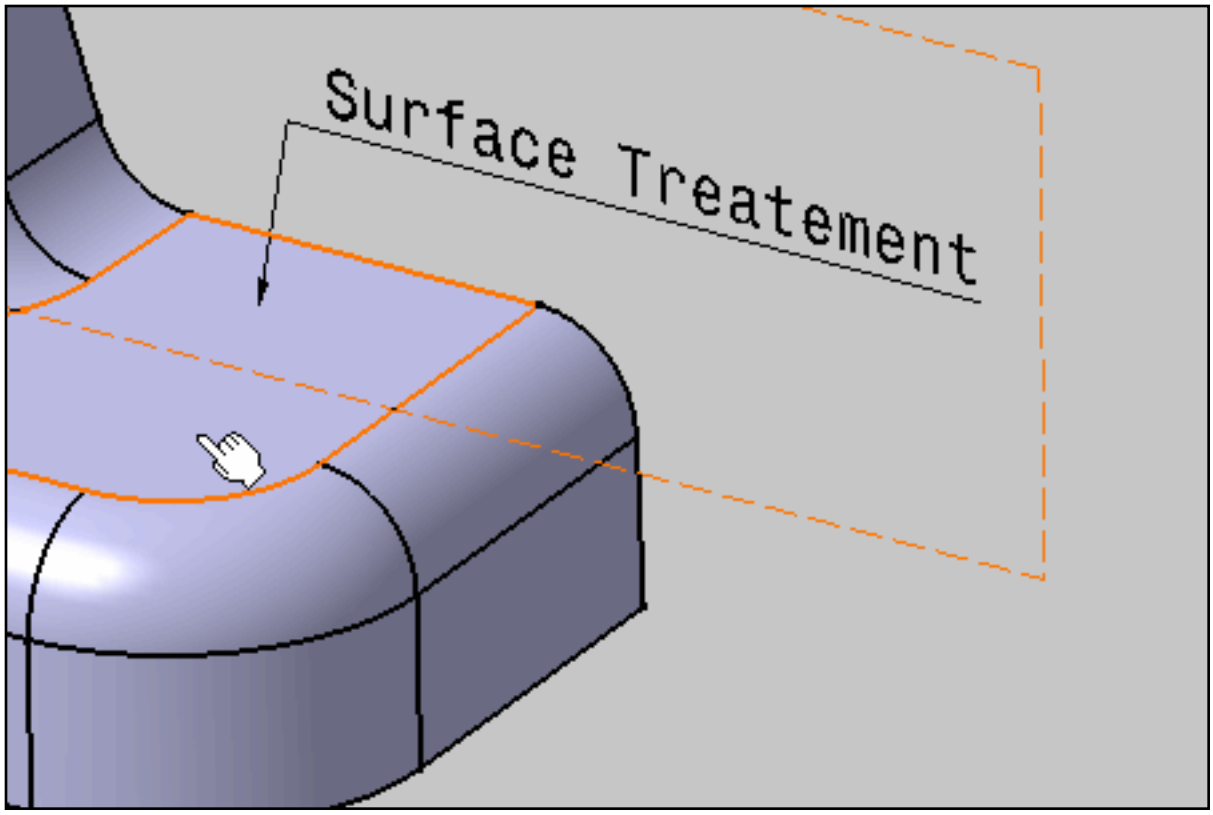
1. Activate the **Projected View.1** annotation plane.



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.

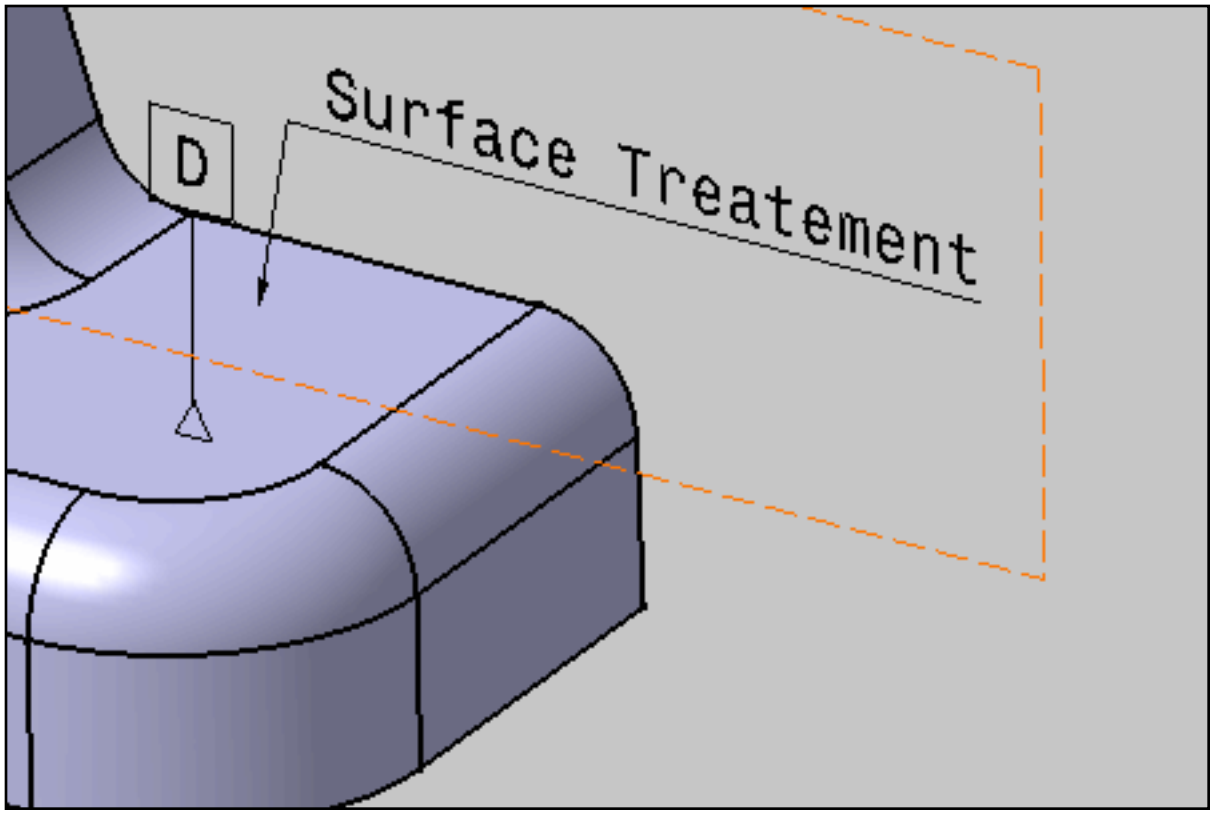


The **Datum Feature** dialog box displays with D as default identifier.



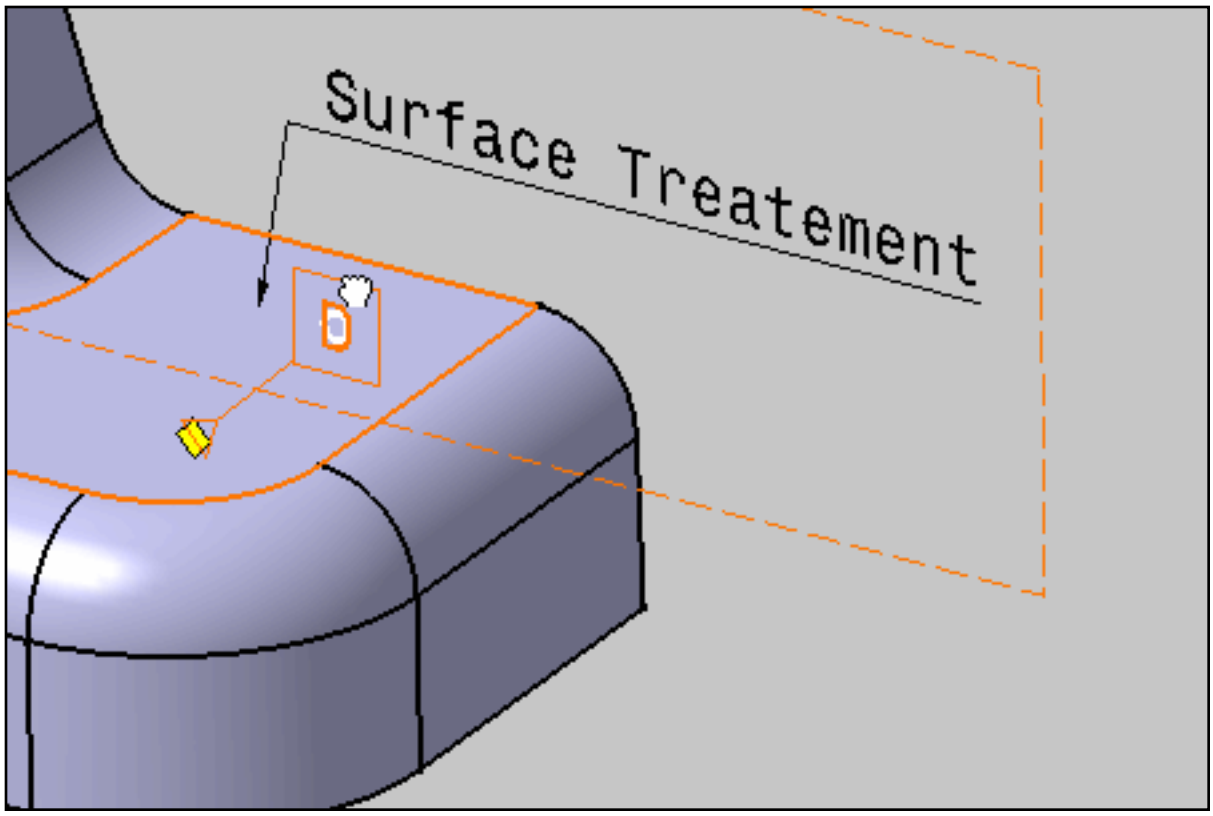
4. Click **OK** to create the datum if the identifier corresponds to your choice.

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

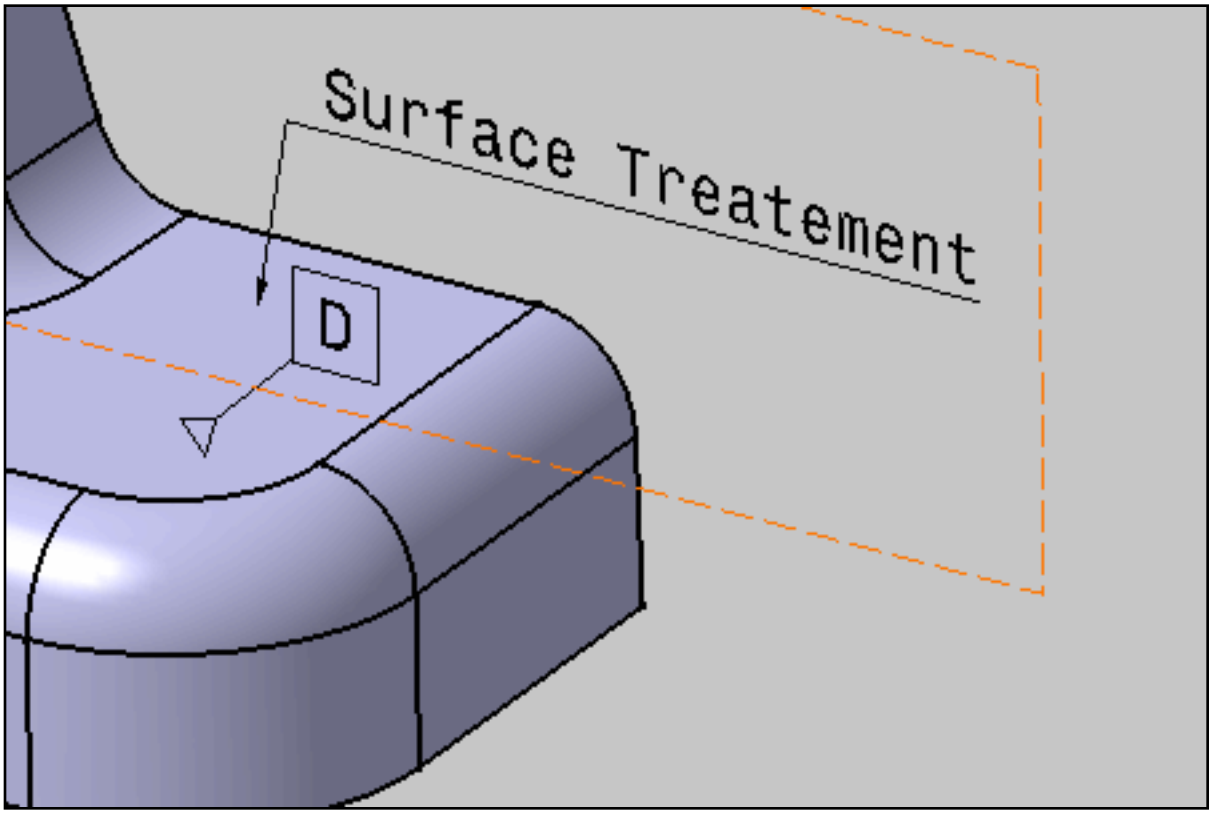


The datum is only a 3D annotation without any semantic link to the geometrical tolerancing.

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.

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Creating Geometrical Tolerances

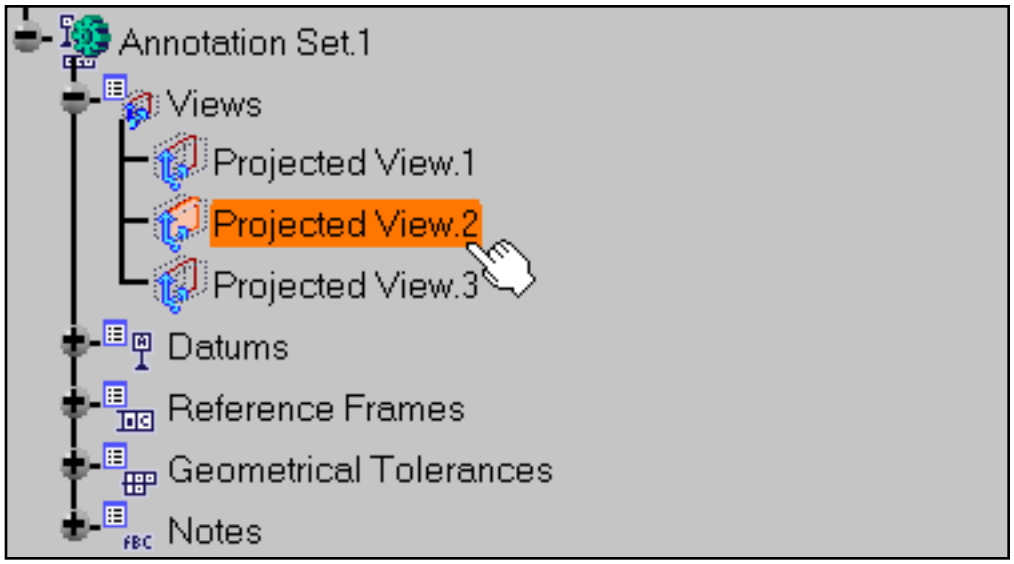
This task will show you how to create a geometrical tolerance annotation.

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 Tolerancing_Annotations_04 CATPart document.

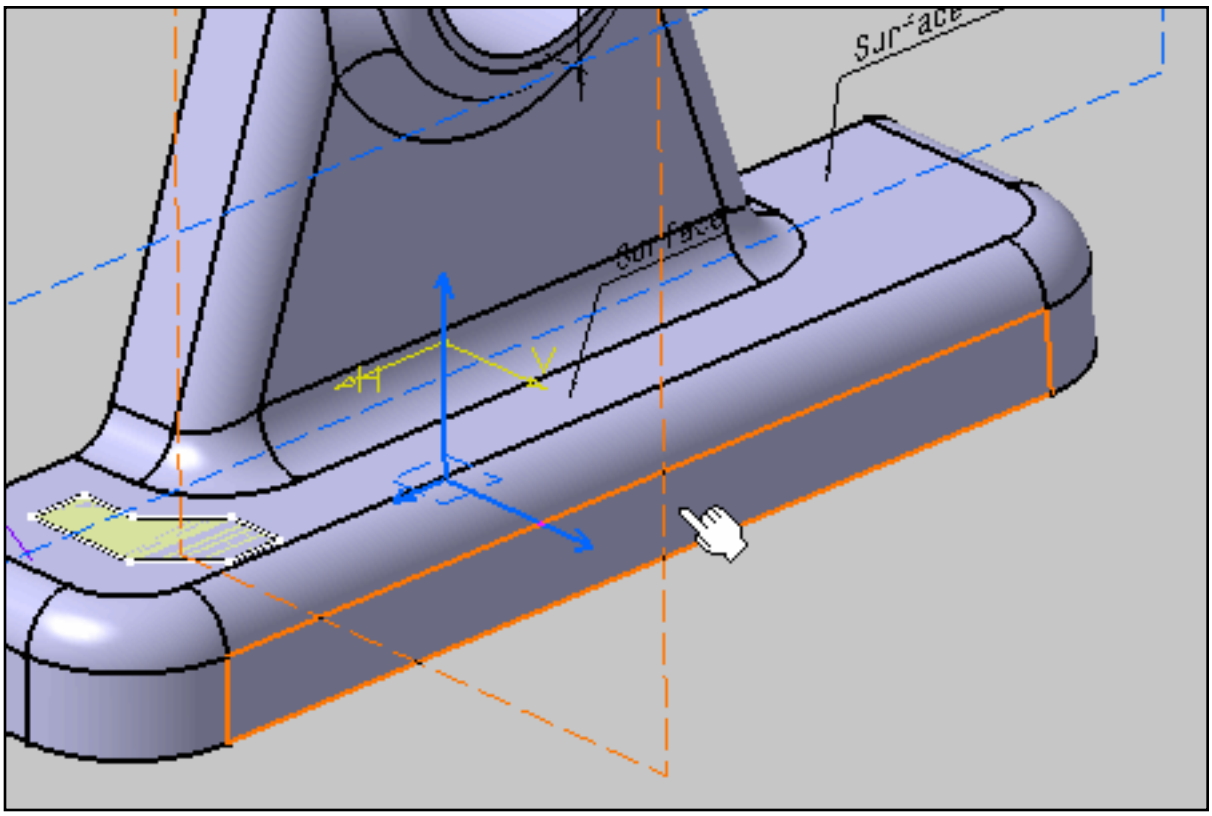
1. Select the **Projected View.2** annotation plane.



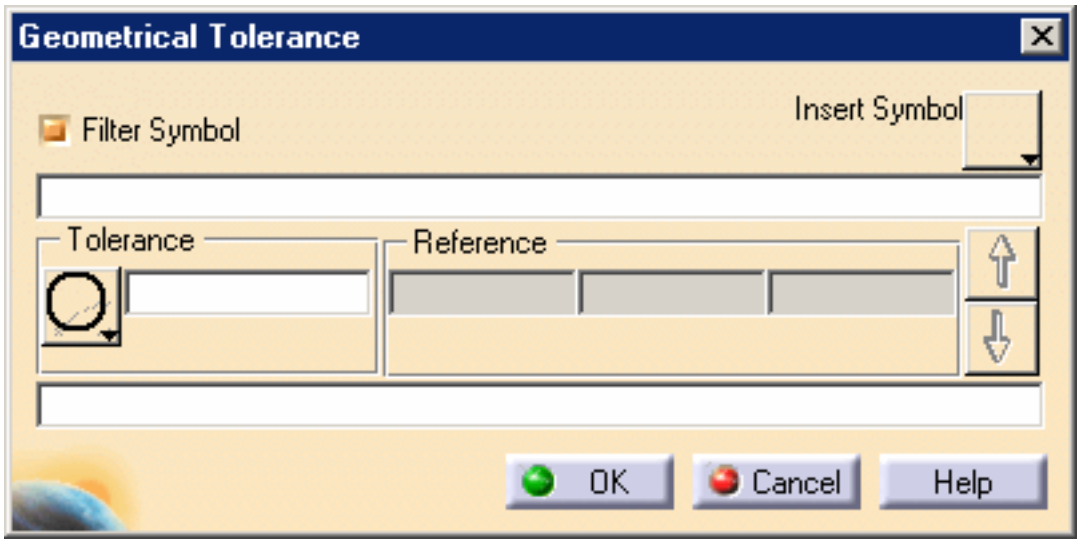
2. Click the **Geometrical Tolerance** icon: 

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:

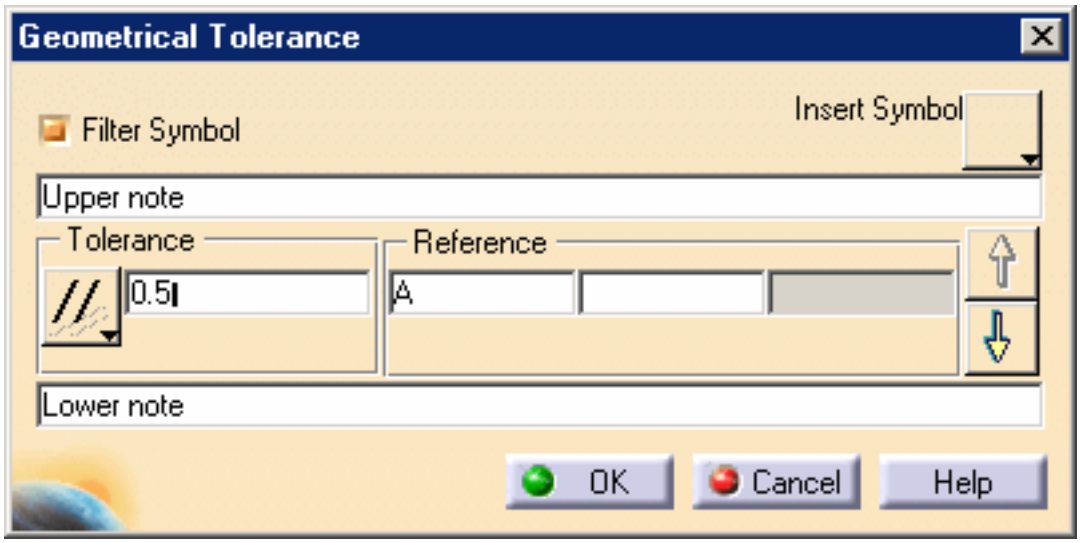
- Specify as many specification lines as you want (with the Up and Down arrows).
- Insert several modifiers anywhere in a tolerance or a reference.
- Add notes upper and lower the set of specification.

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.

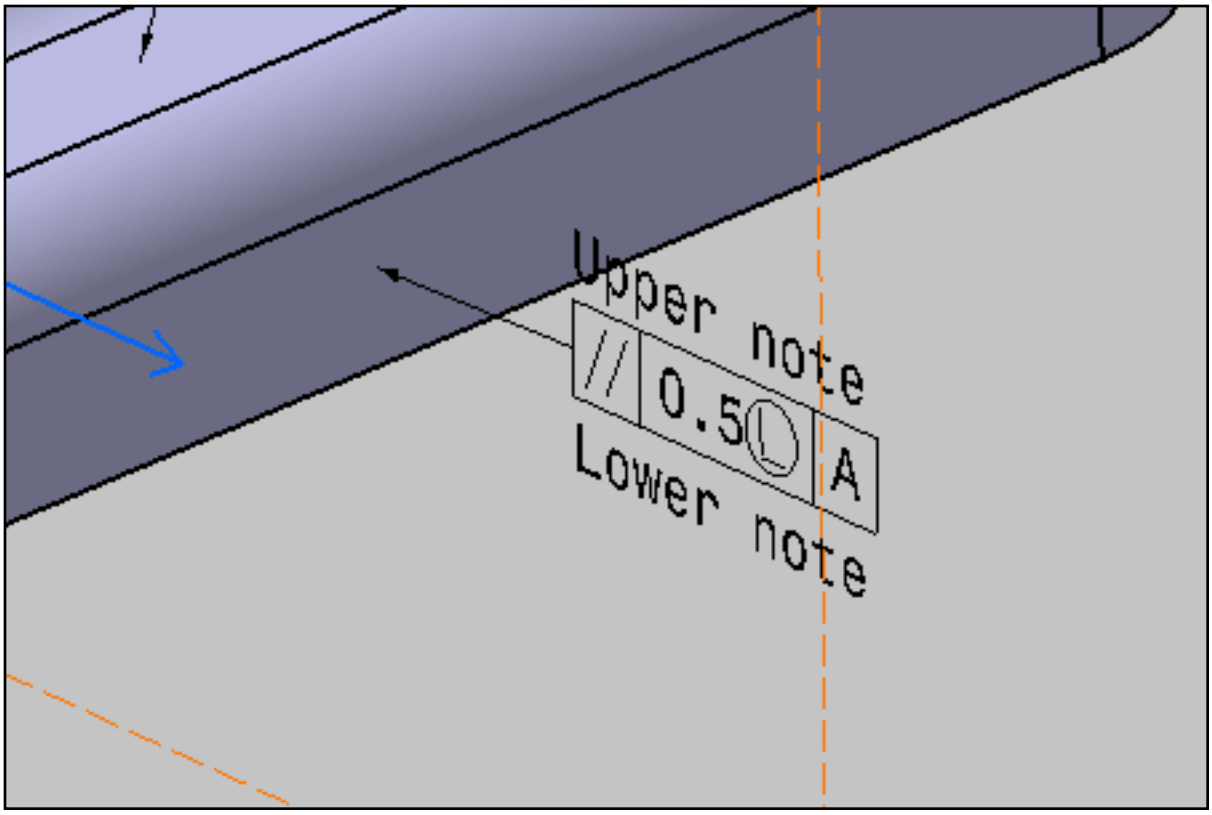
6. Specify the upper and lower notes.



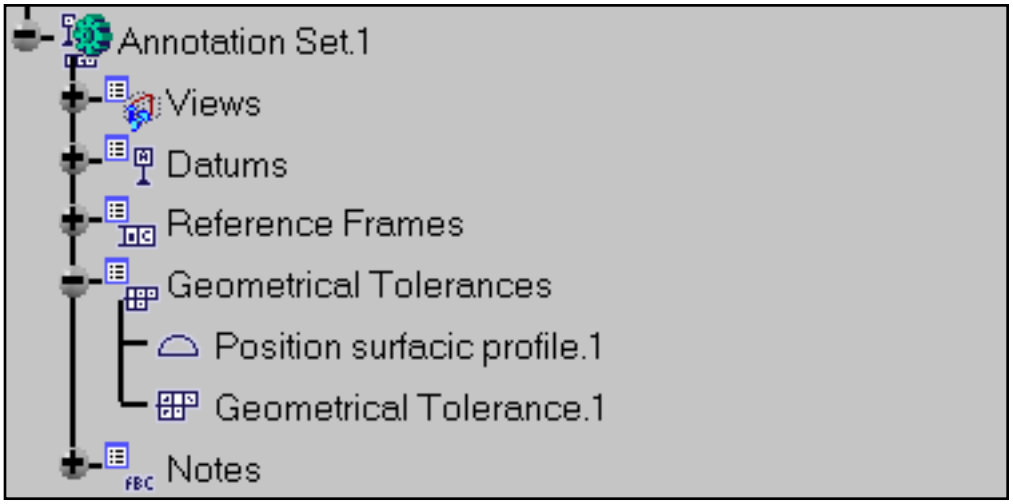
Modifiers are not displaying in tolerance and reference fields and appear with a "|" character.

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.



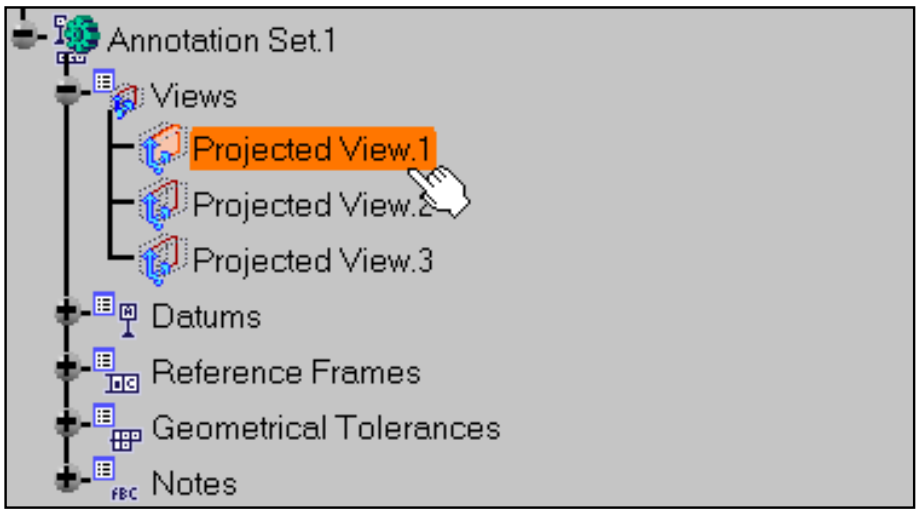
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Creating Roughness Symbols

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

Open the Tolerancing_Annotations_04 CATPart document.

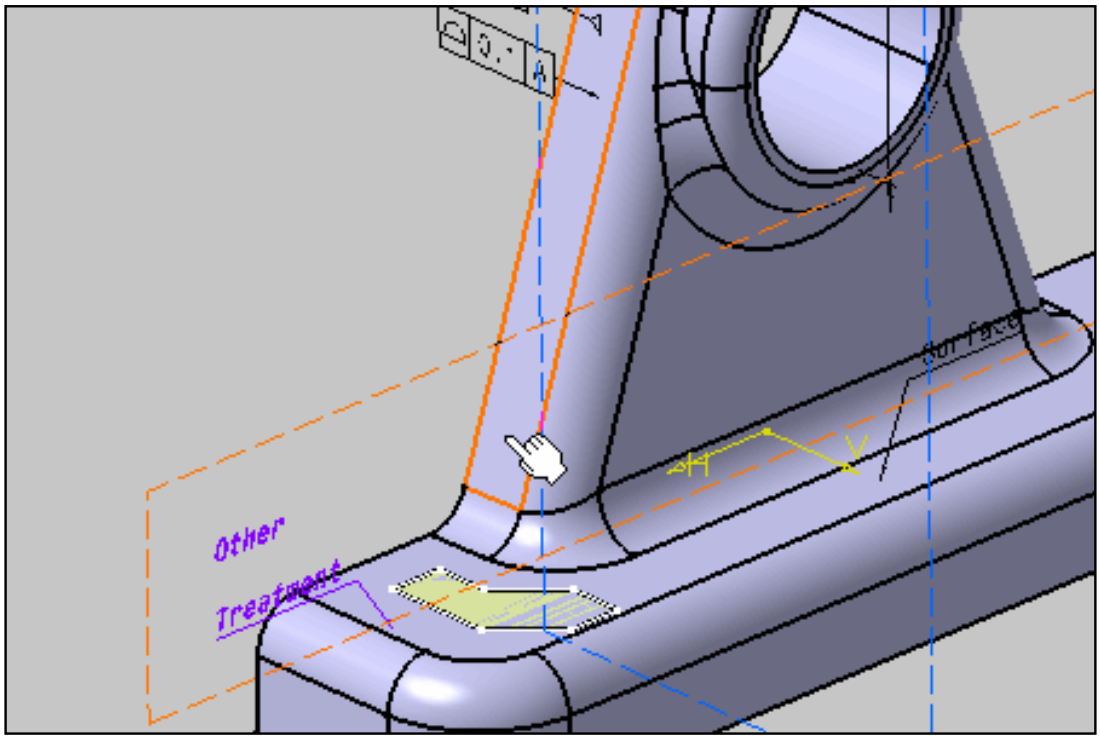
1. Select the **Projected View.1** annotation plane.



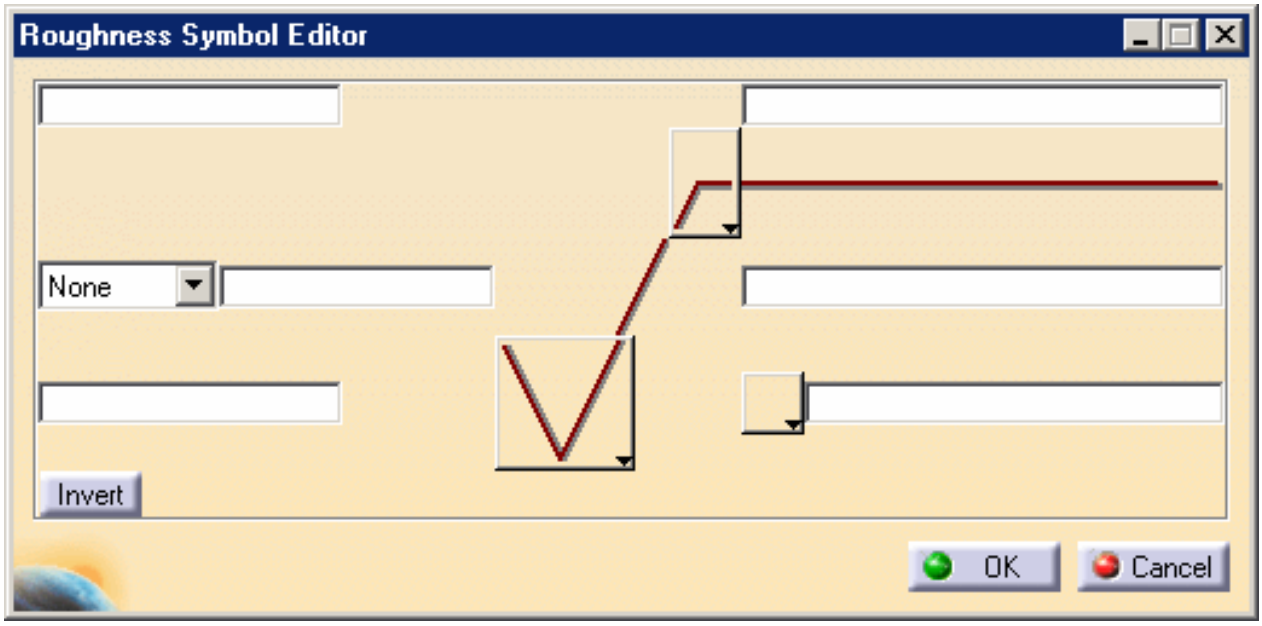
2. Click the **Roughness** icon:

3. Select the surface as shown on the part.

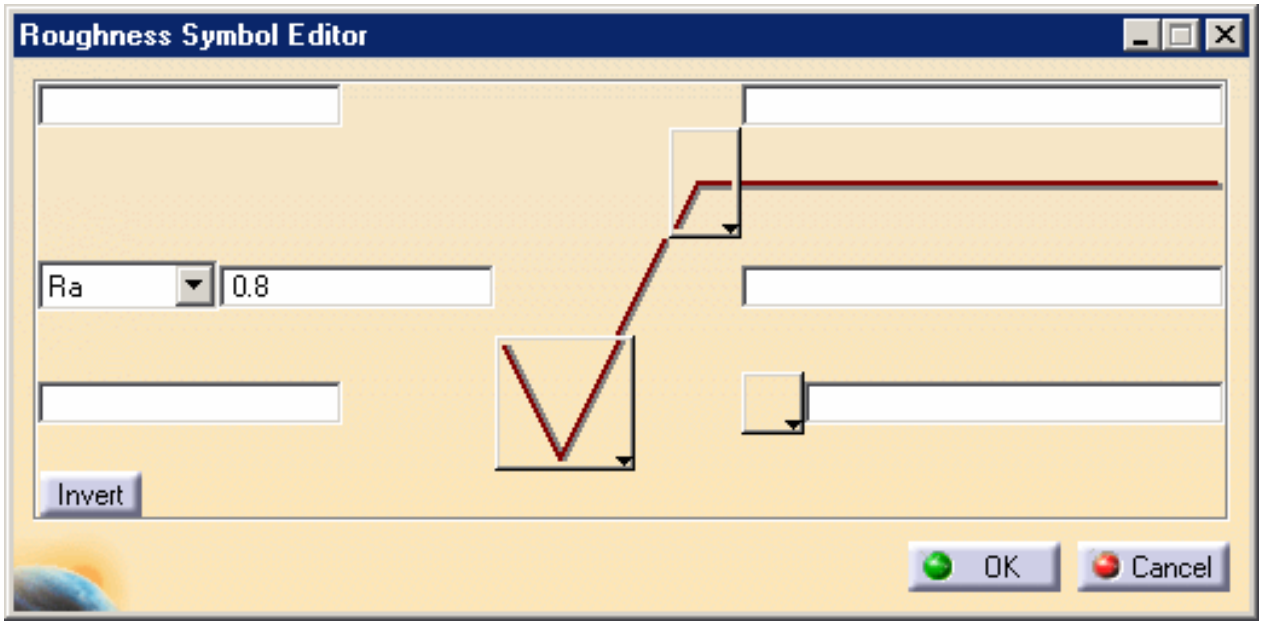
This scenario illustrates the creation of a roughness symbol by selecting geometry, but you can also select any Part Design or Generative Shape Design feature in the specification tree. In this case, the created annotation will not be attached to the selected feature, but to its geometrical elements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



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

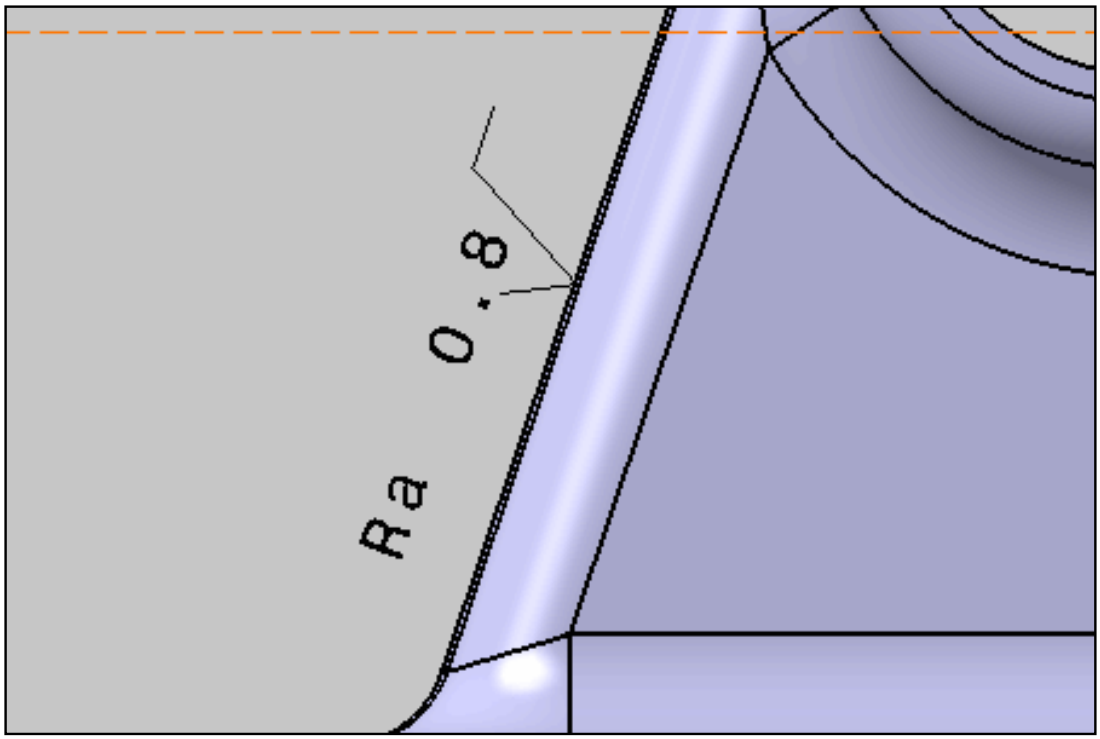


4. Select **Ra** symbol and enter a value: 0.8



5. Click **OK**.

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



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Creating Isolated Annotations

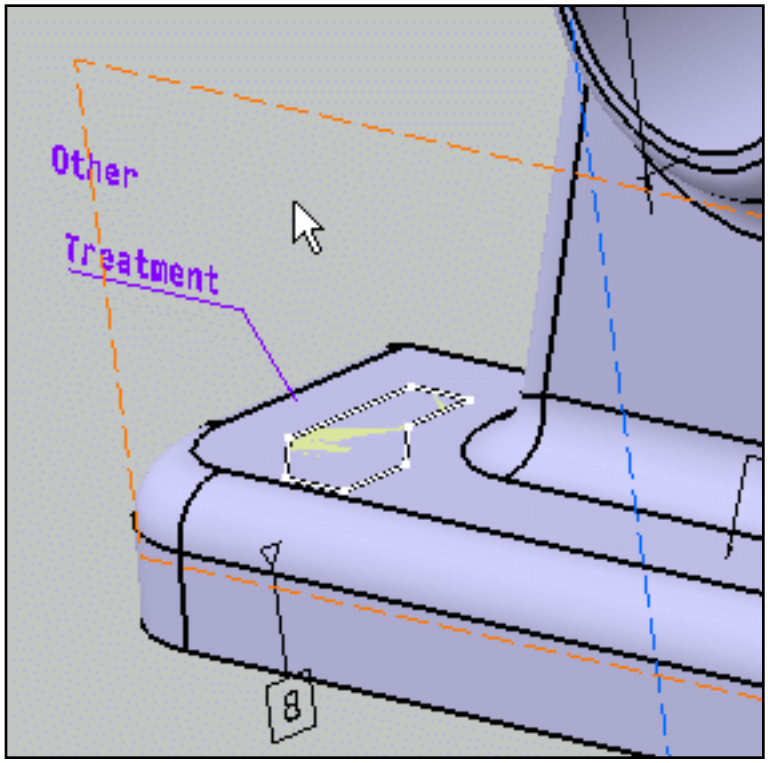
This task shows you how to create isolated annotations, i.e. annotations that are not linked to any geometry.

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)

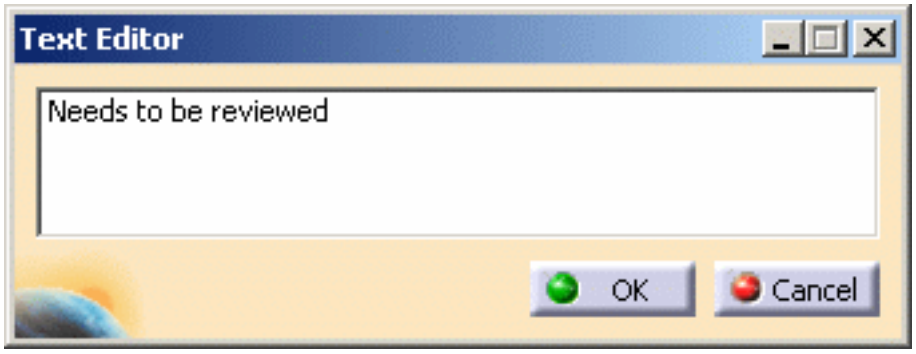
Open the Tolerancing_Annotations_04 CATPart document.

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).

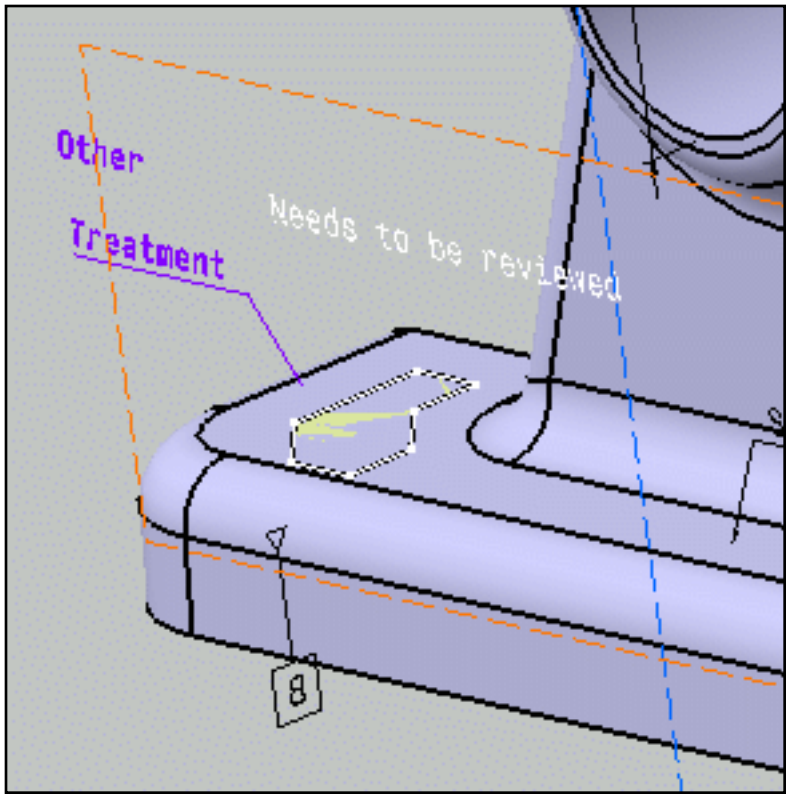


The **Text Editor** dialog box is displayed.

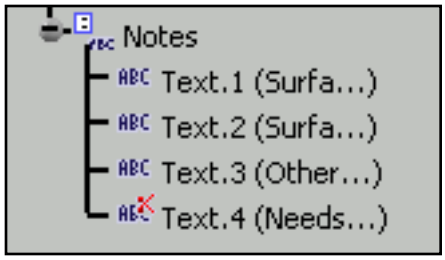
4. Type your text and then click **OK**.



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.

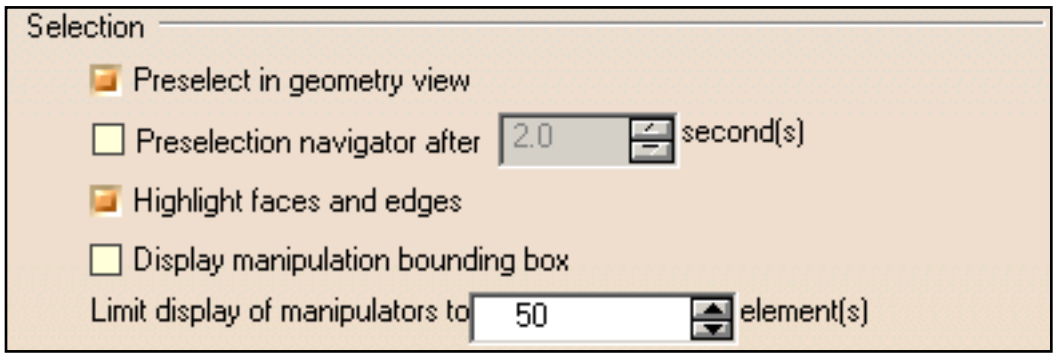
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Creating Dimensions

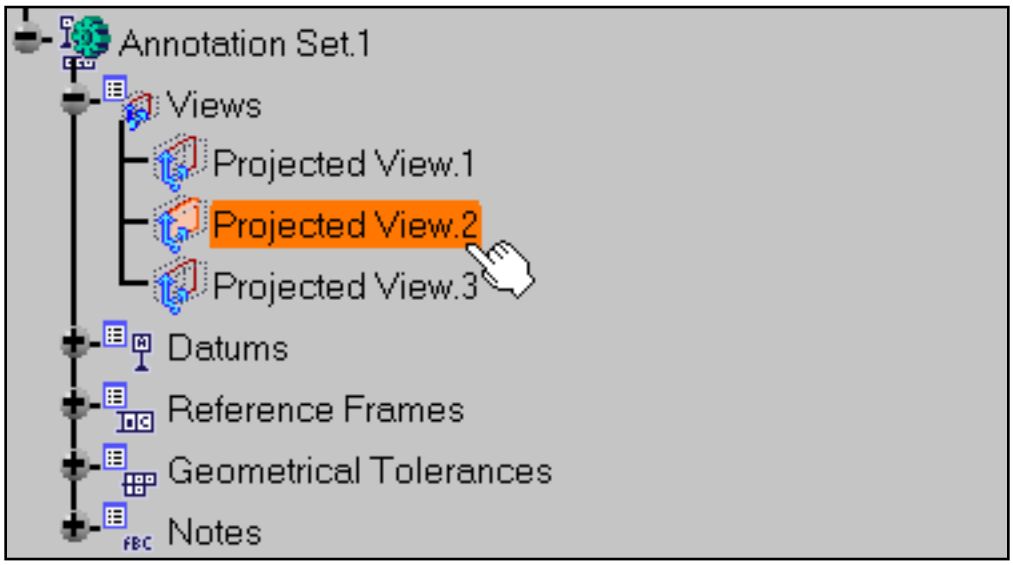
This task shows you how to create a dimension annotation. See Dimension Units reference for dimension's unit display.

Open the Tolerancing_Annotations_04 CATPart document.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

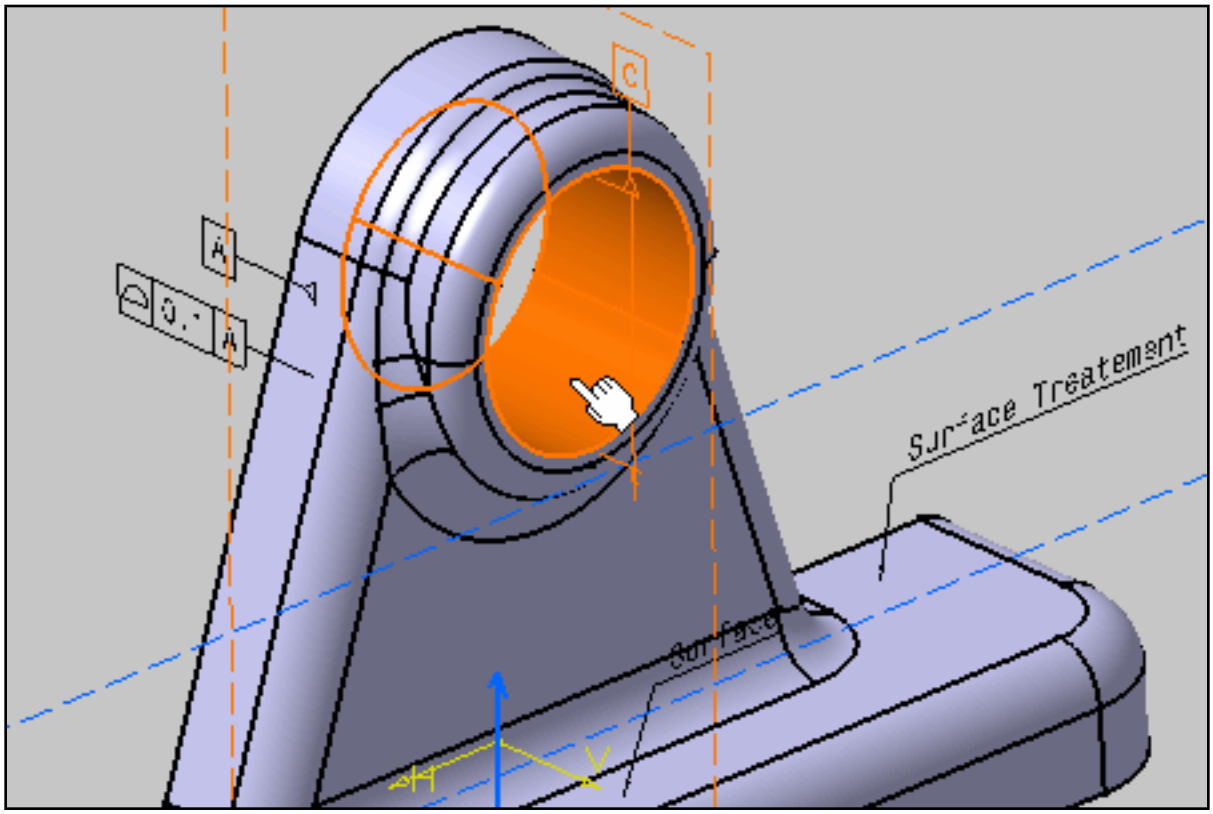


1. Activate the **Projected View.2** annotation plane.



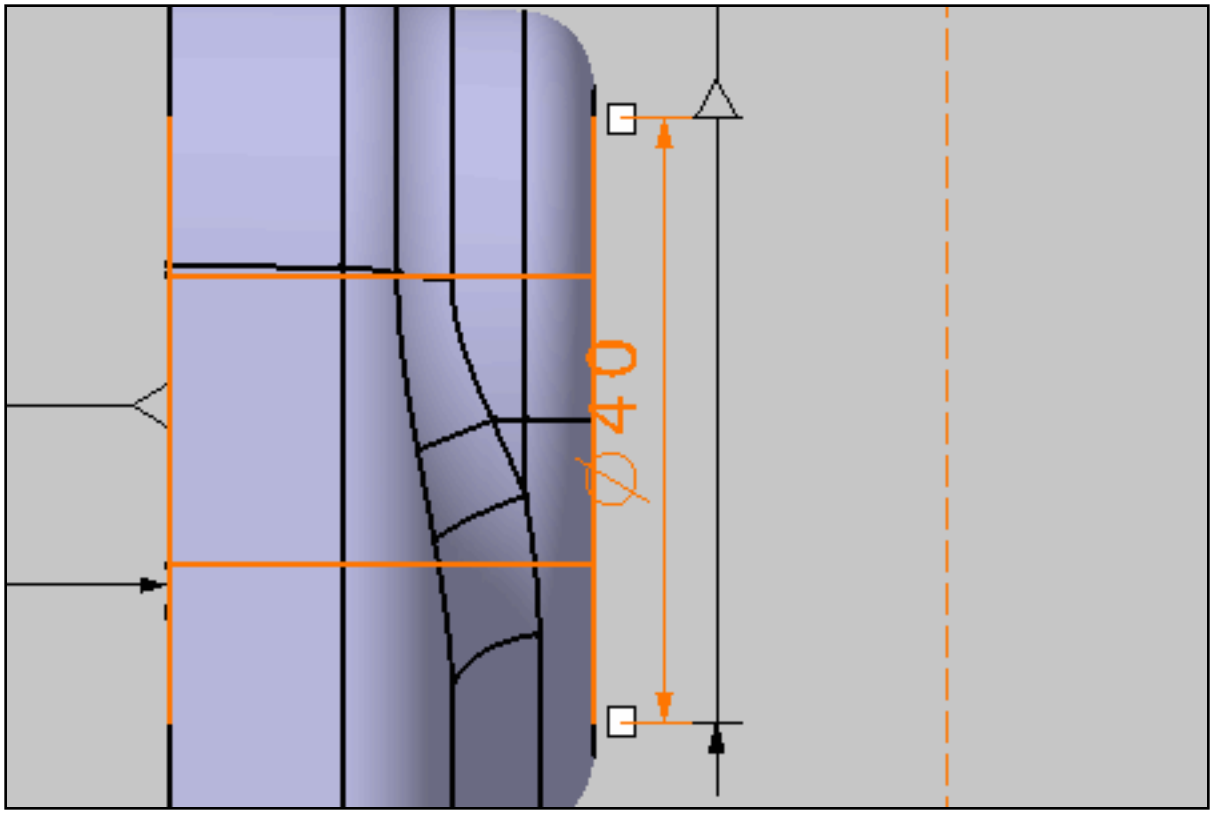
2. Click the **Dimensions** icon: 

3. Select the surface as shown on the part.

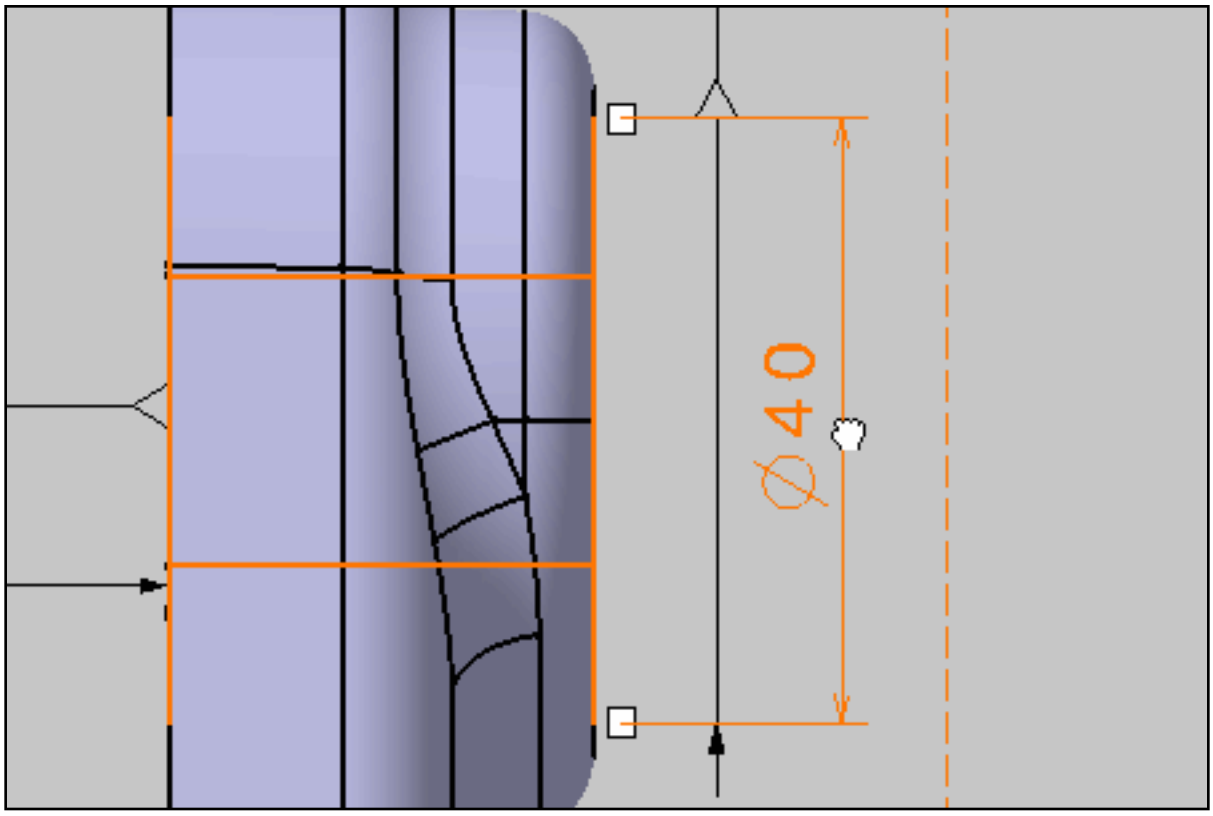


4. Click anywhere to create it.

The dimension is created.

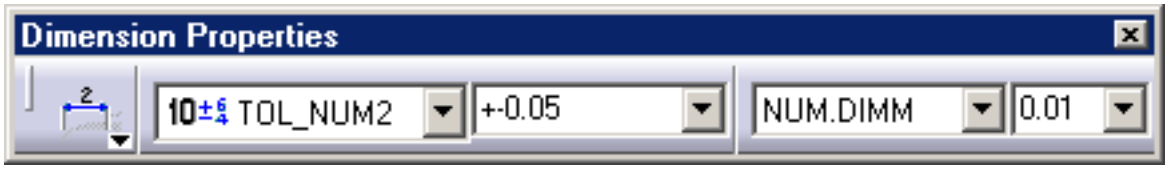


5. Drag the dimension.



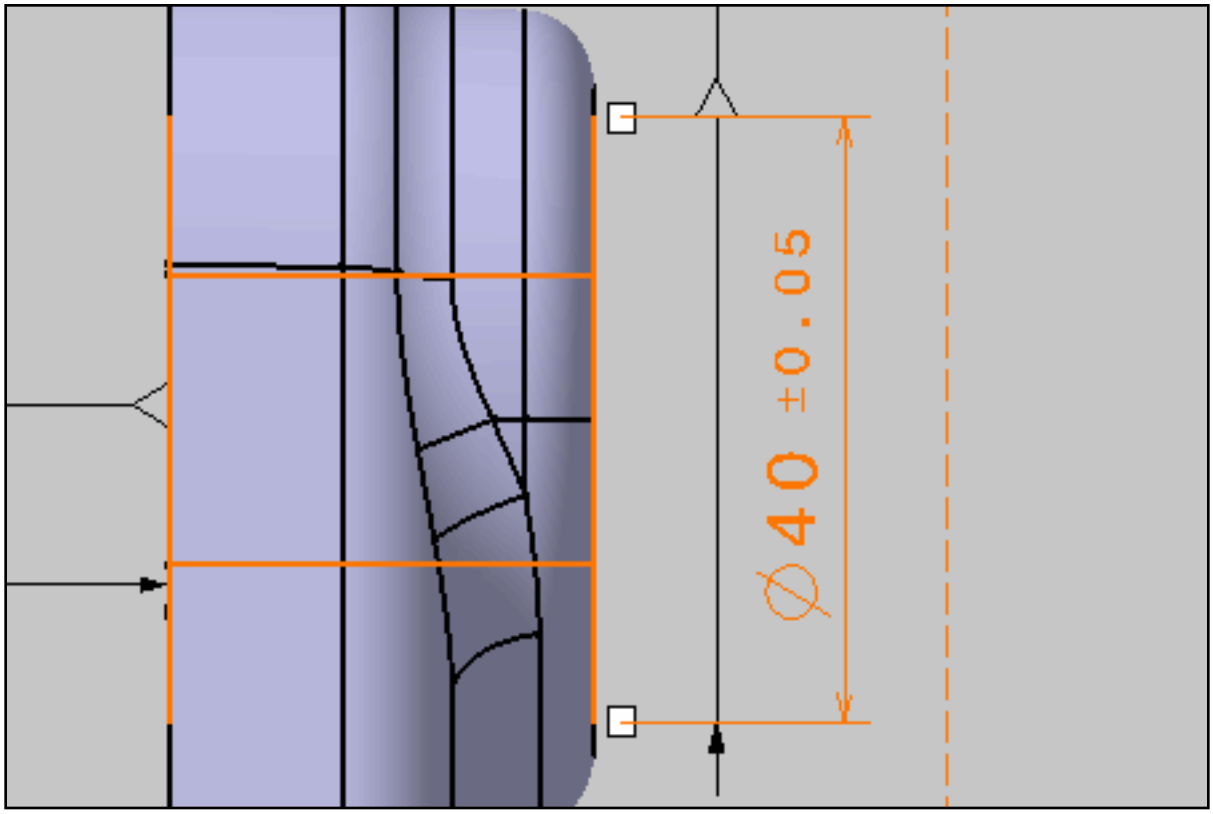
6. In the **Dimension Properties** toolbar, select **TOL_NUM2** in the **Tolerance Description** combo box, and **+-0.5** in the **Tolerance** combo box (the dimension is still selected).

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.

Tolerances are displayed with the dimension.



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Creating Basic Dimensions

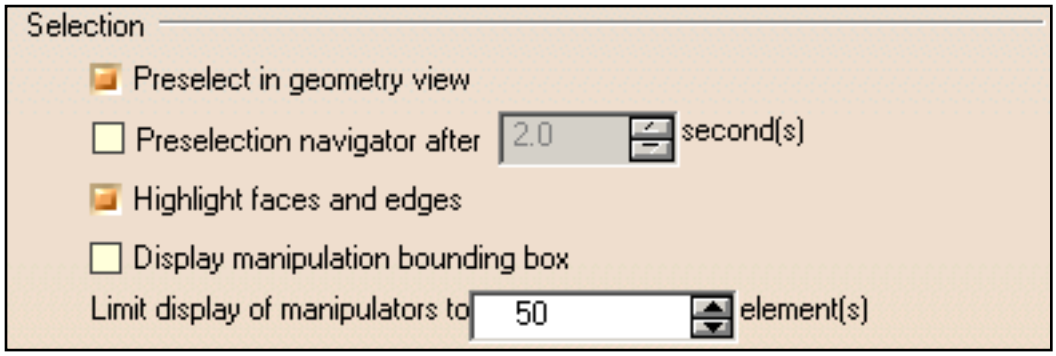
This task shows you how to create basic dimensions. These reference dimensions are used to define the location or the size of a geometrical element, from existing or new dimensions, and related to a specific context.

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

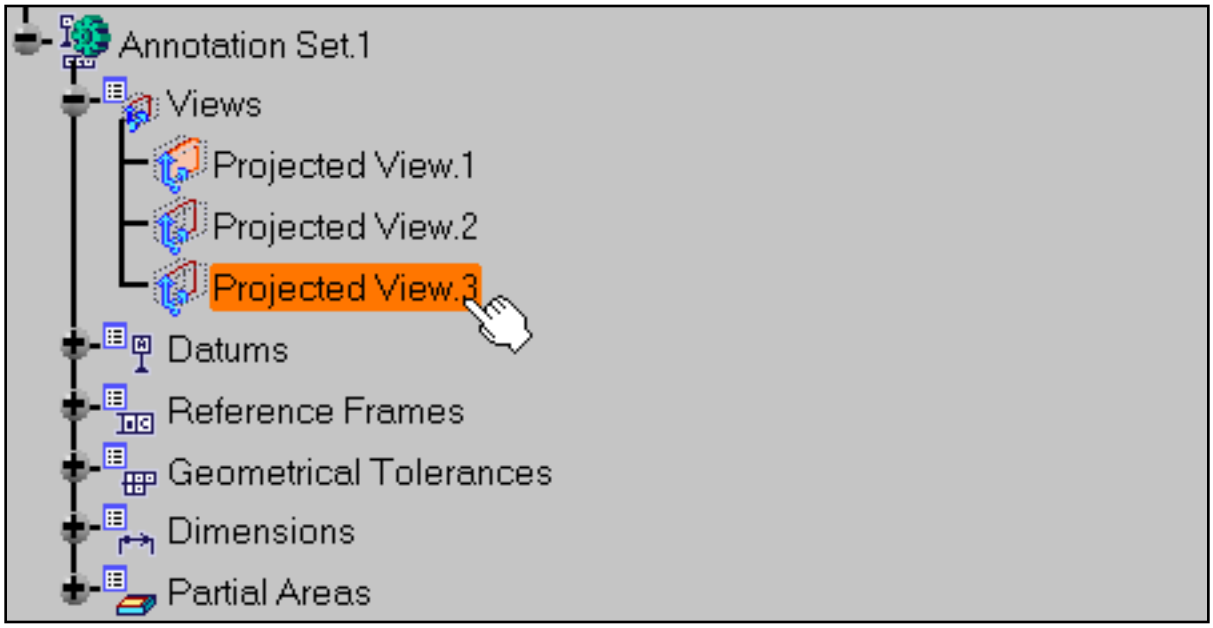
- Restricted areas, where basic dimensions define the restricted area dimensions and location.
- Datum targets, where basic dimensions define the datum target location from another datum target or geometrical elements.
- Datum reference frames, where basic dimensions define the datum reference frame 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.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



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

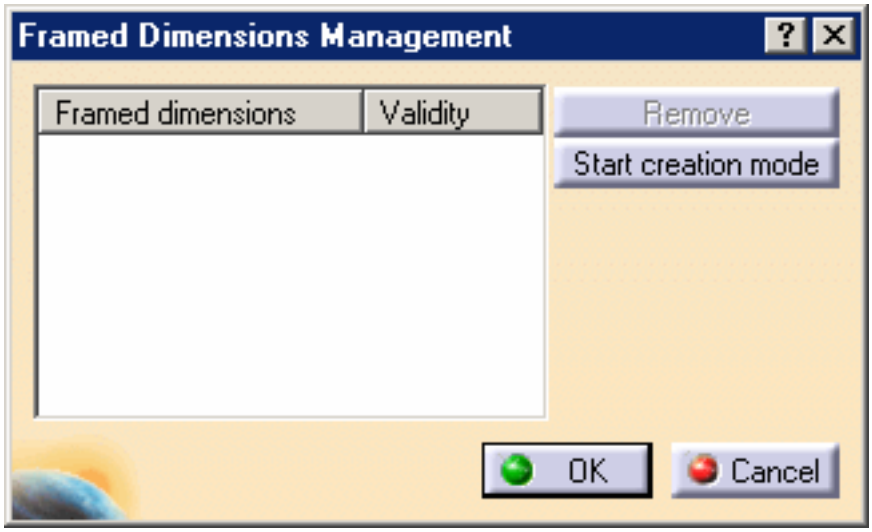


2. Click the **Basic Dimension** icon: 

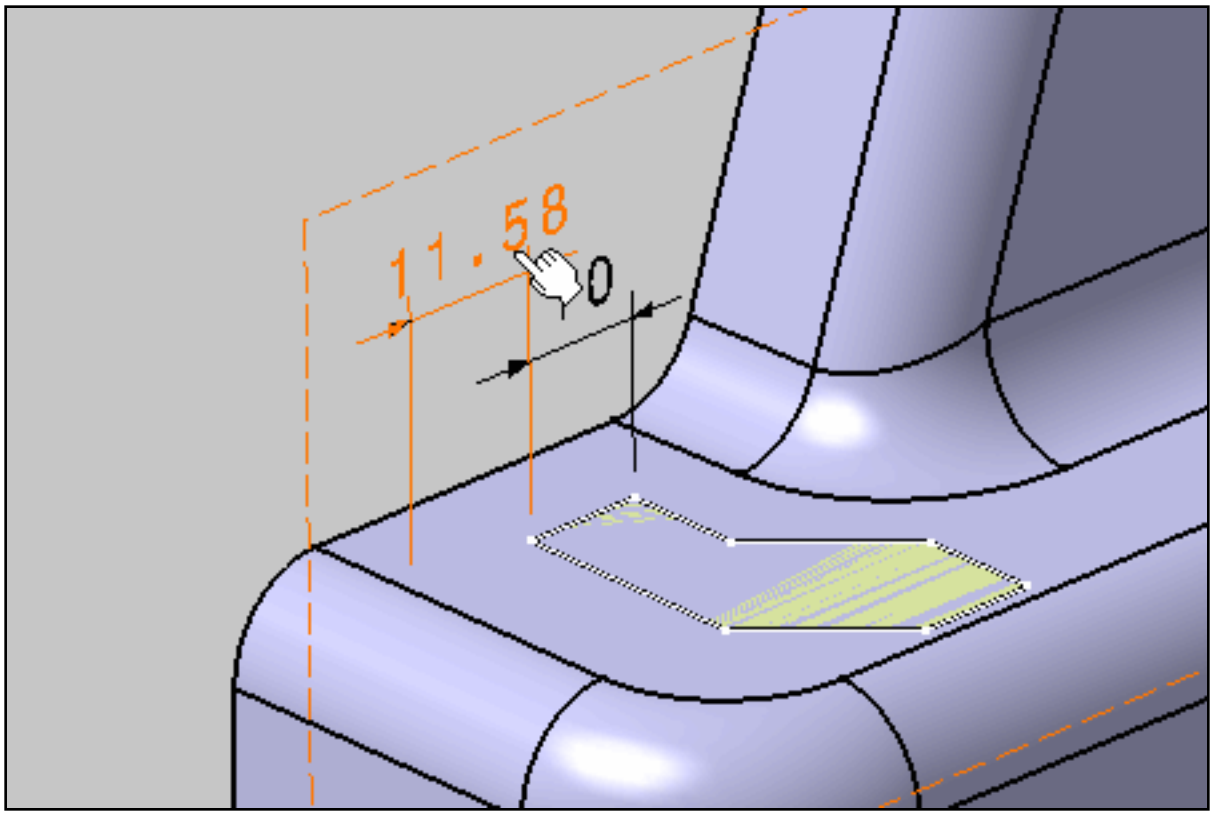
3. Select the **Restricted Area.1** partial area.



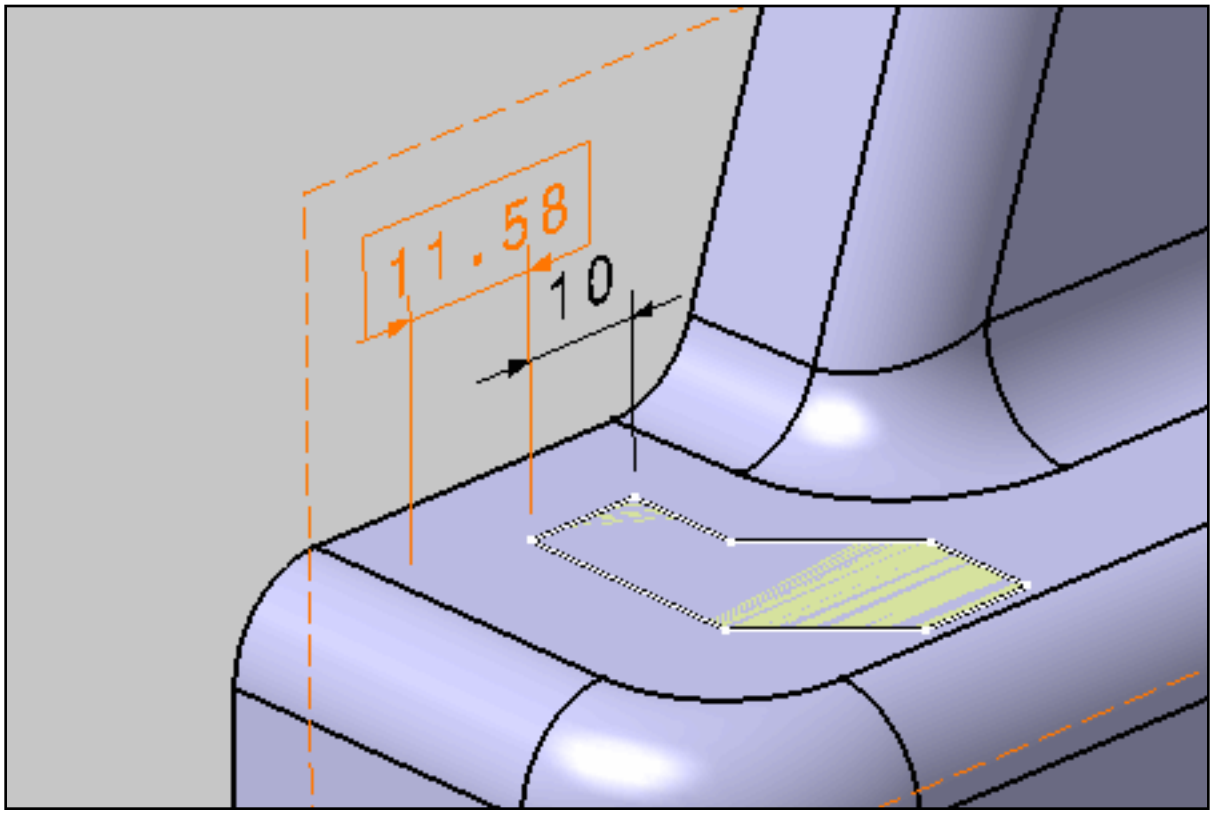
The **Framed Dimensions Management** dialog box appears.

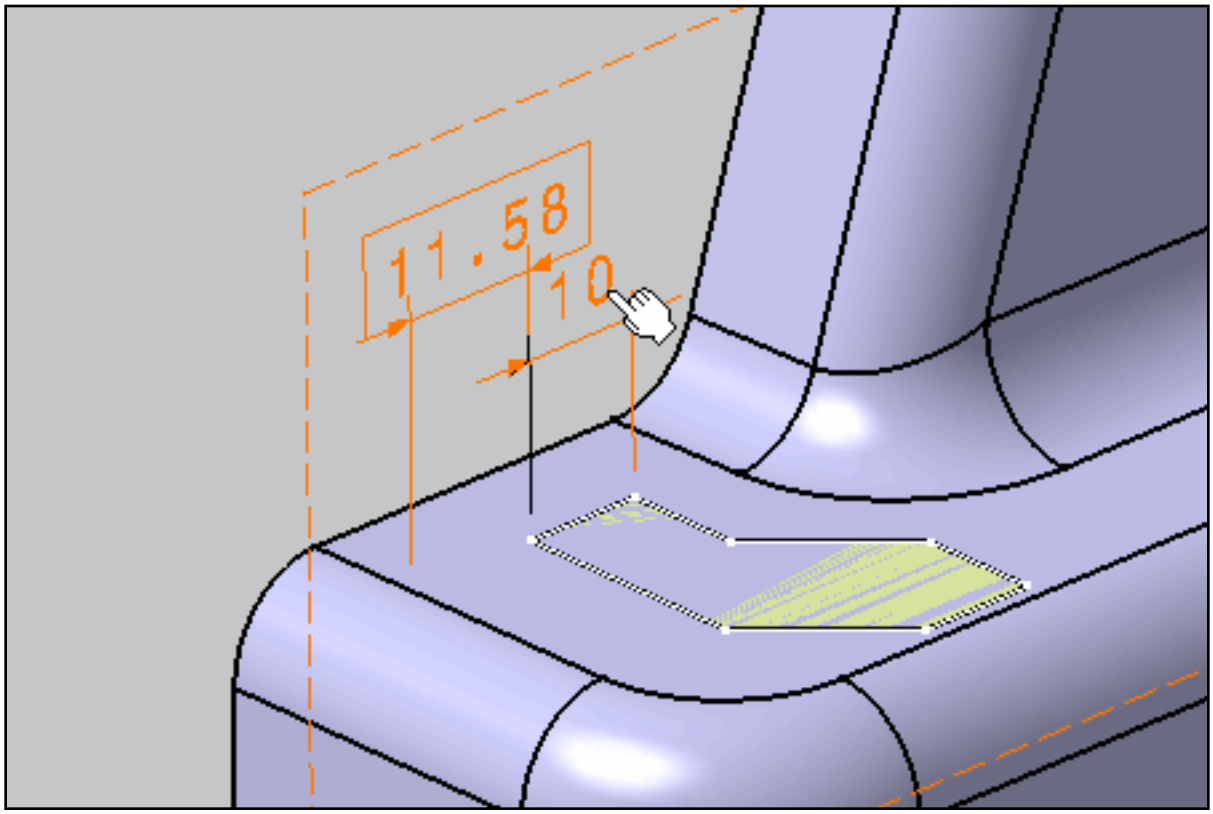


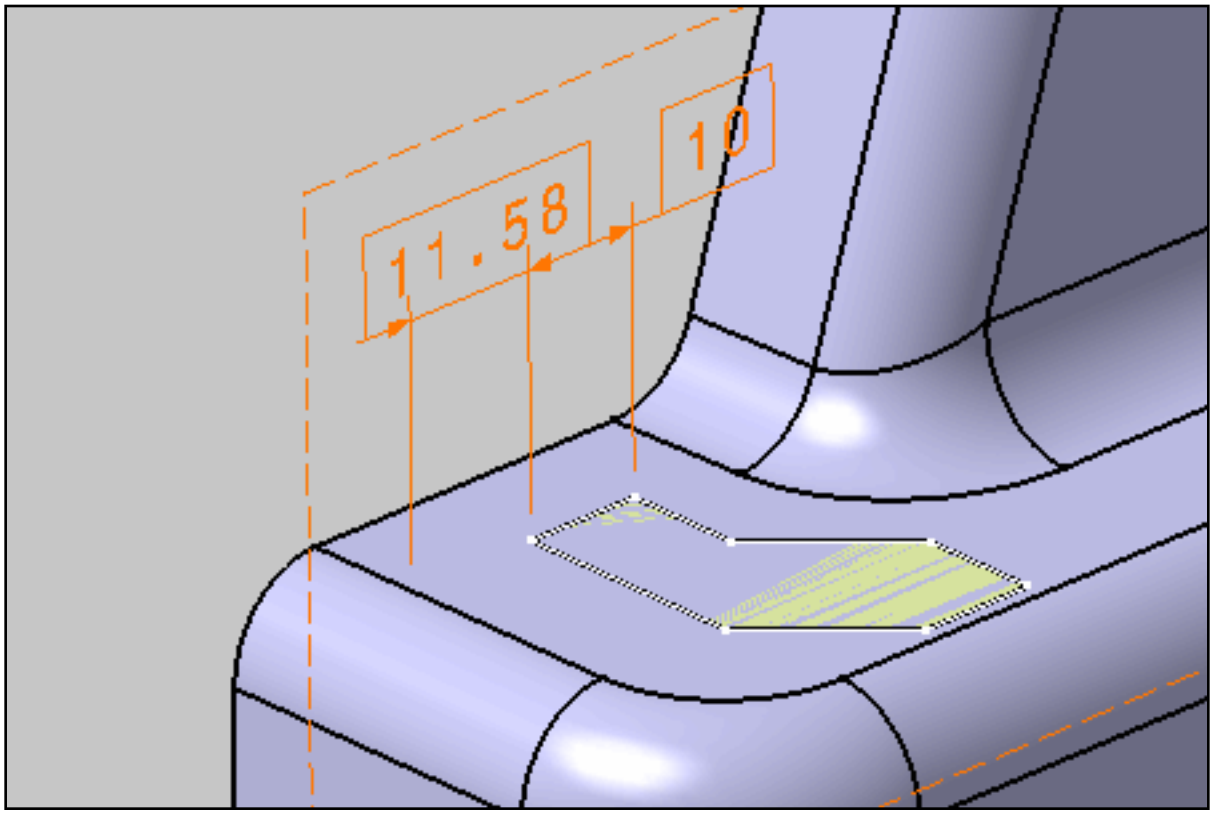
4. Select dimensions as shown on the part.



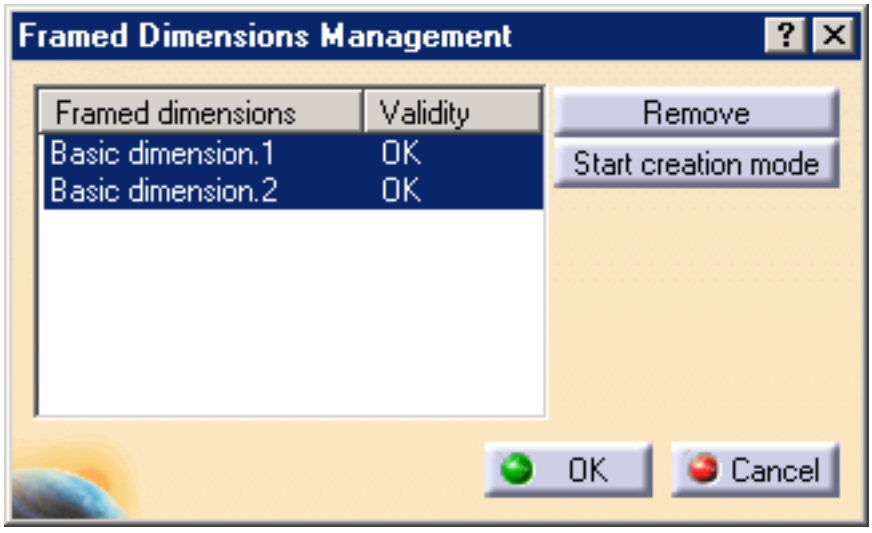
The dimension is converted into a basic dimension when clicking.


11.58
10

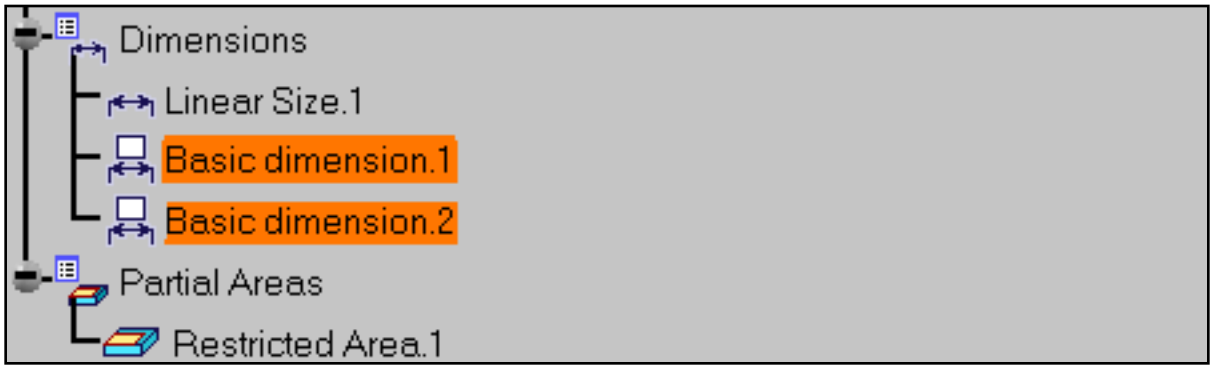

11.58
10



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

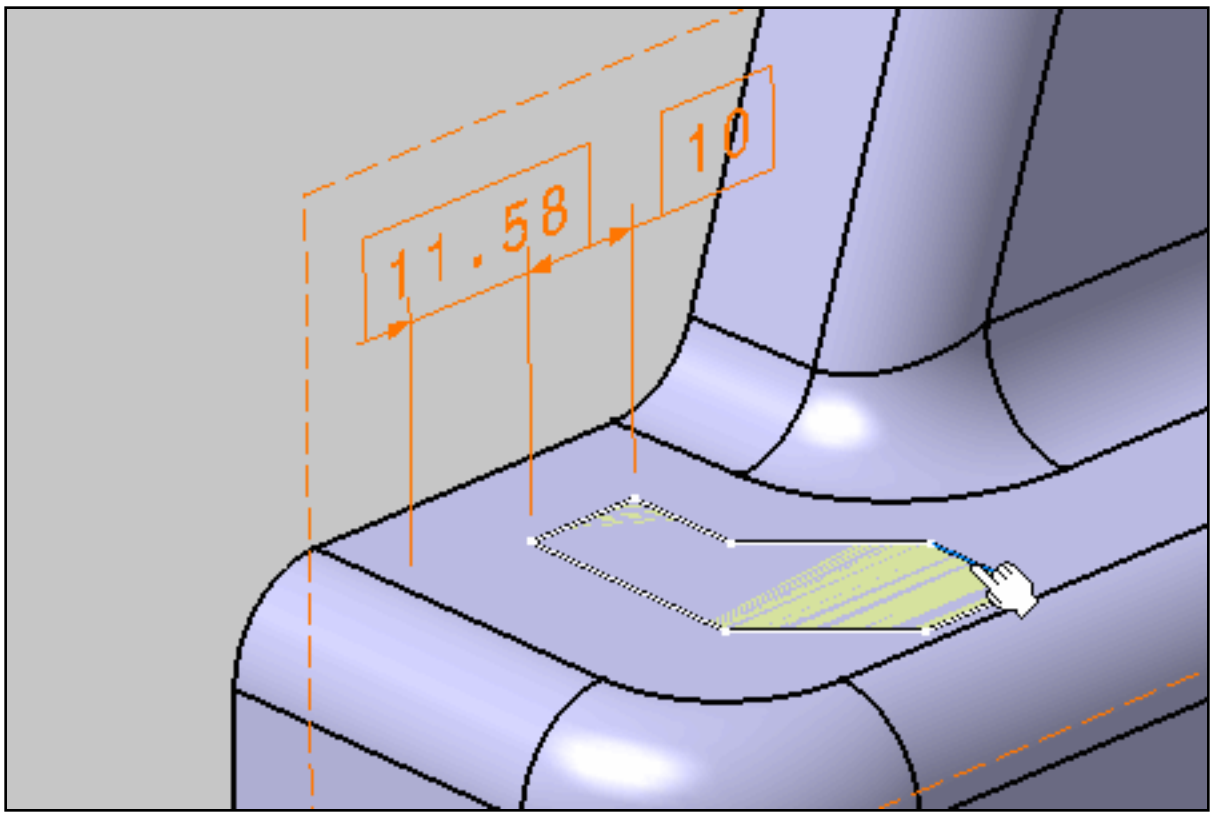


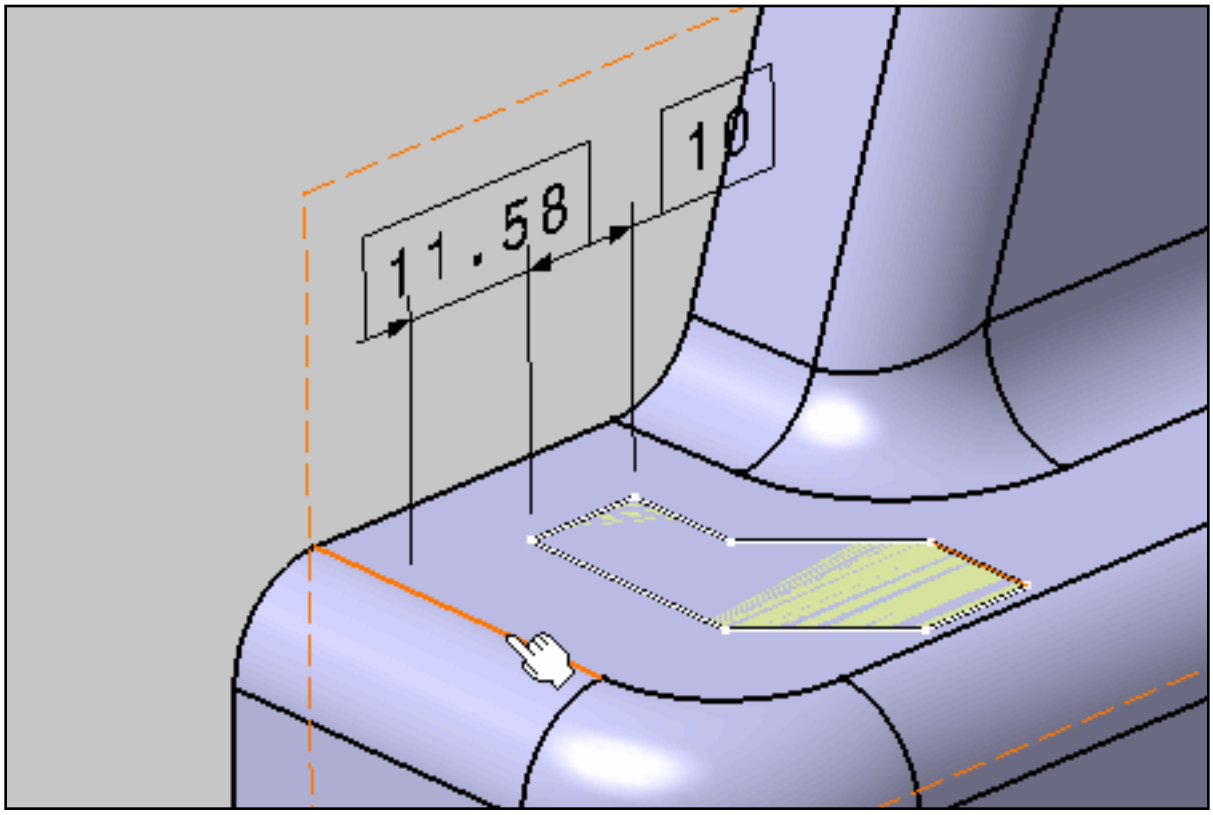
During the conversion, converted annotations remain selected in the specification tree, the geometry area and the dialog box.



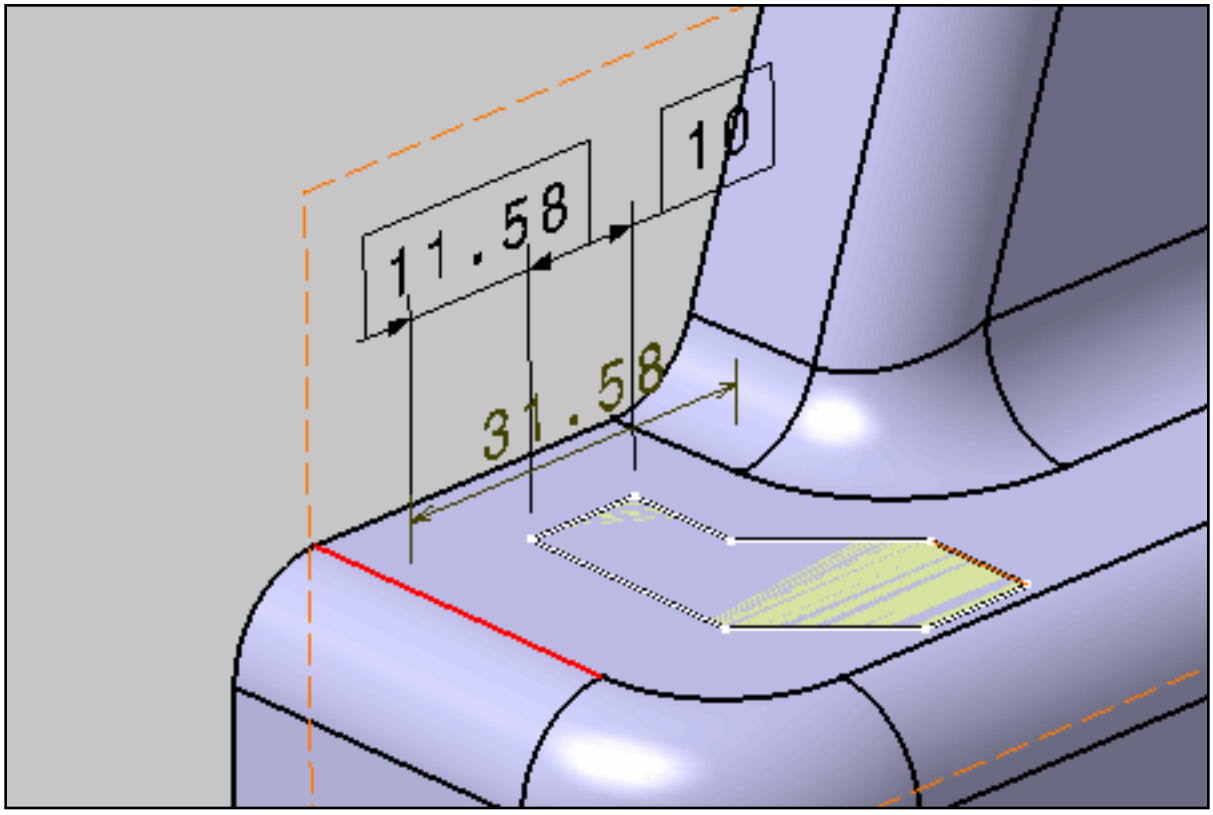
5. Click **Start creation mode** in the **Framed Dimensions Management** dialog box.

6. Select edges as shown on the part.

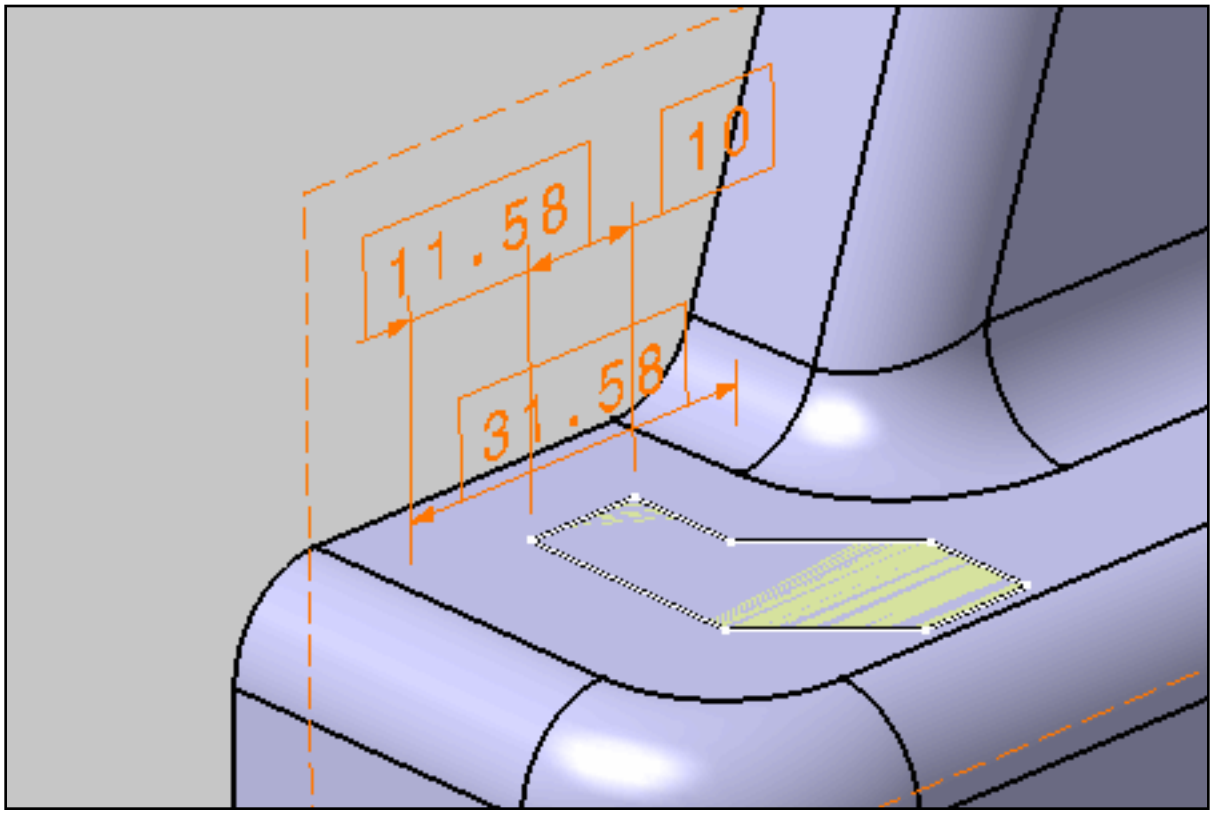




The basic dimension is pre-created.

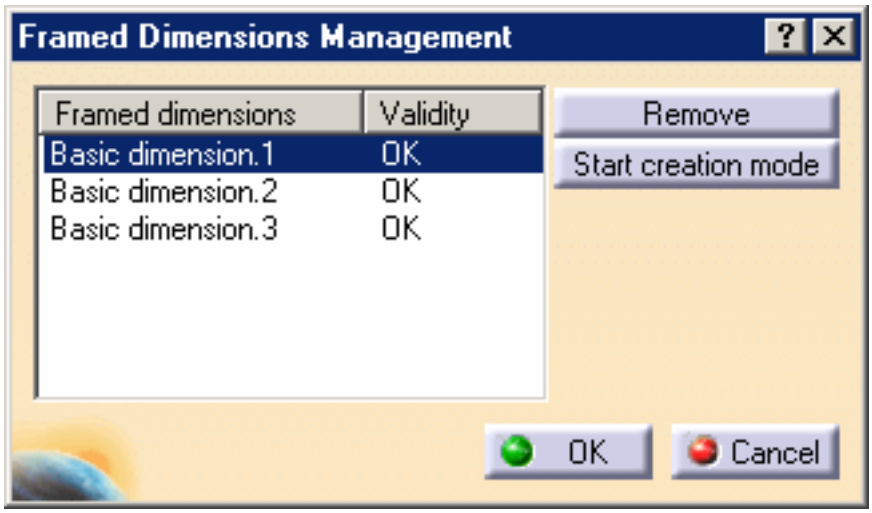


7. Click in the free space to create the basic dimension.

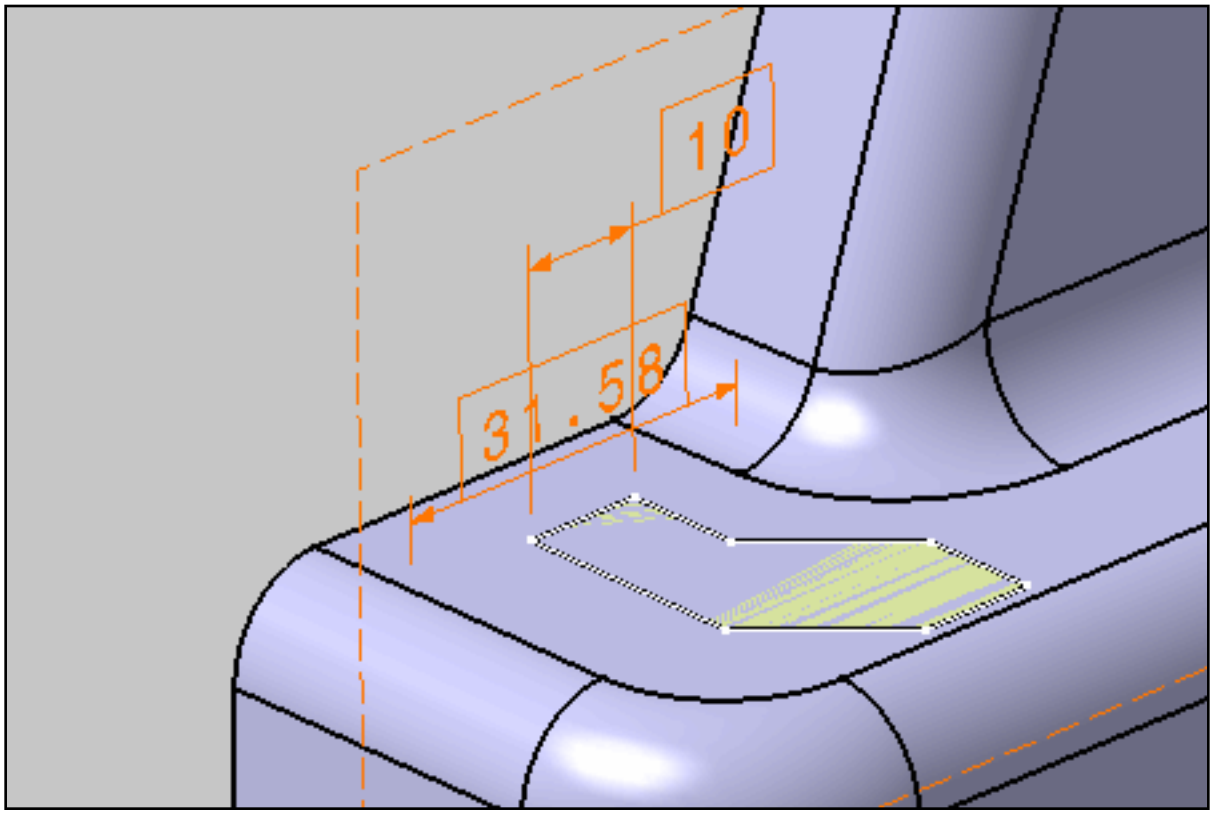


8. Click **End creation mode** in the **Framed Dimensions Management** dialog box.

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.

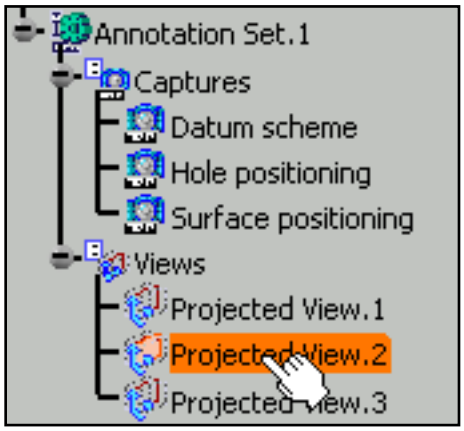
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Creating Coordinate Dimensions

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.

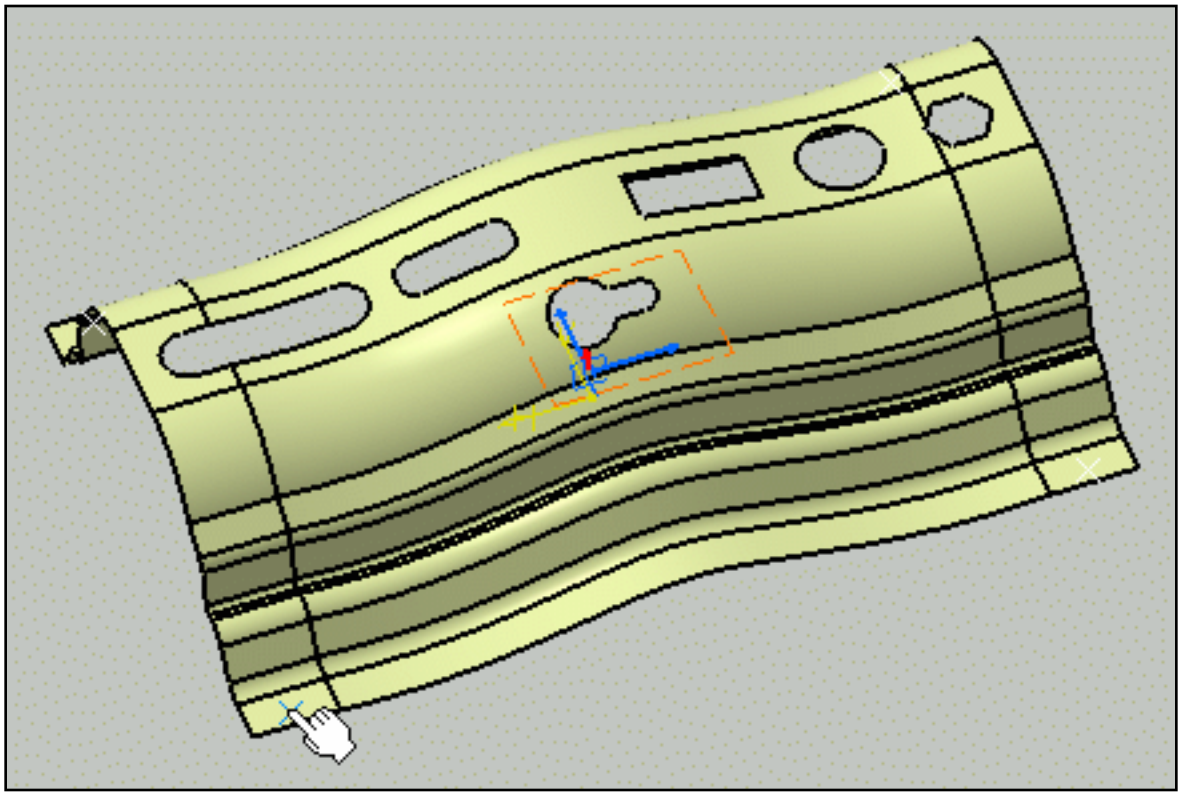


2. Click the **Coordinate Dimensions** icon. 
3. Right-click anywhere to display the contextual menu.
   - **2D Coordinates** lets you create 2D (x, y) coordinate dimensions in the active view axis system.
   - **3D Coordinates** lets you create 3D (x, y, z) coordinate dimensions in the part/product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

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



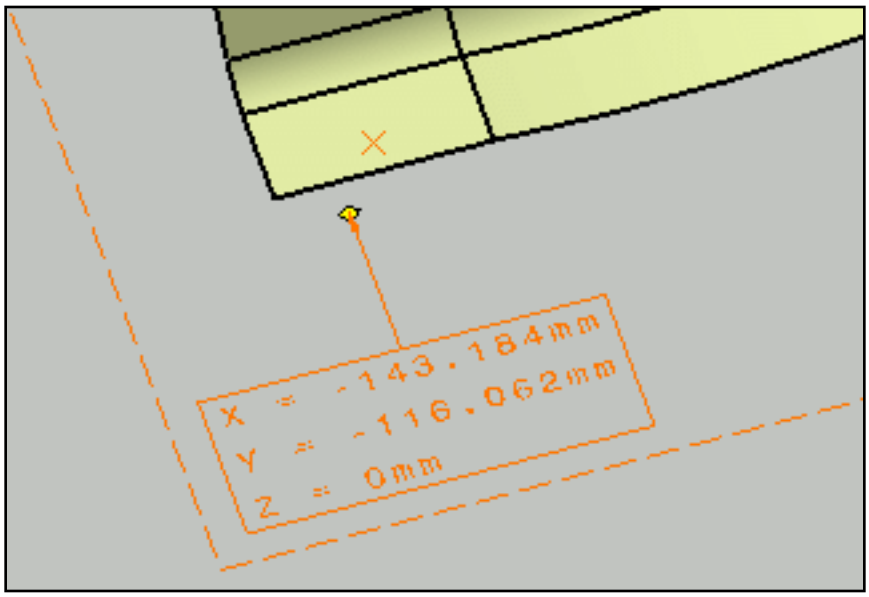
4. Select a point as shown below.



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).



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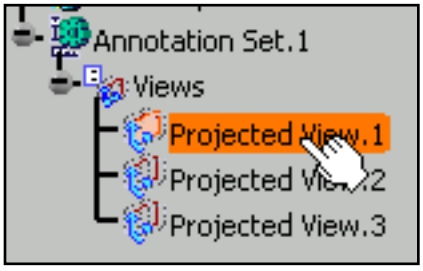
Creating Stacked Dimensions

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

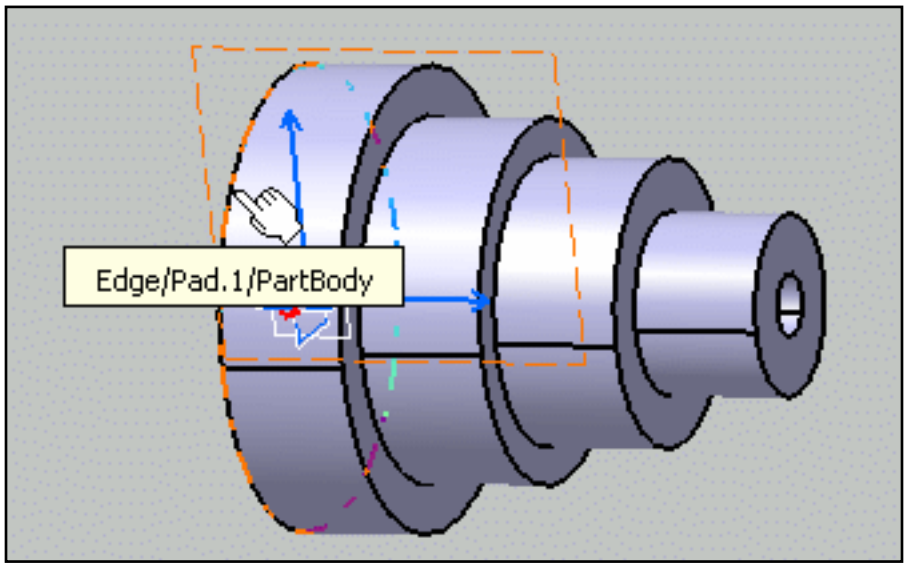
Open the Tolerancing_Annotations_10 CATPart document.

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**.

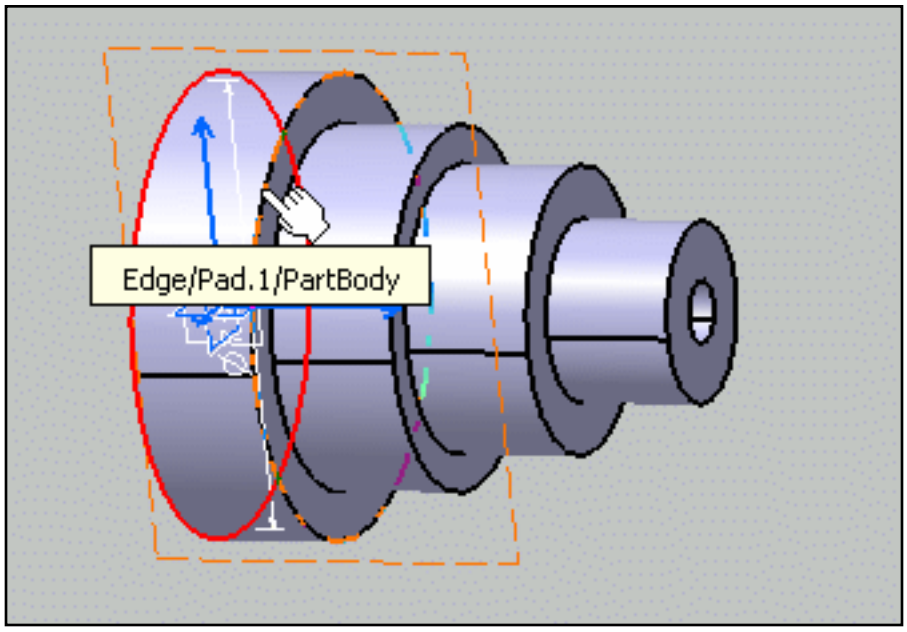
1. Activate the **Projected View.1** annotation plane.



2. Click the **Stacked Dimensions** icon. 
3. Select the first element.

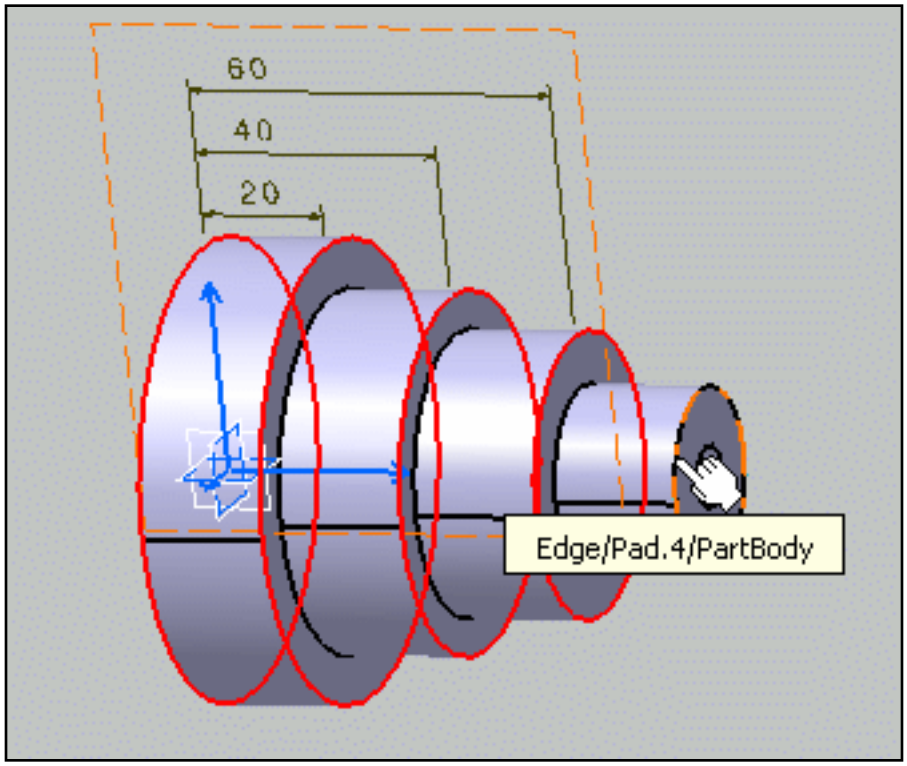


4. Select the second element.

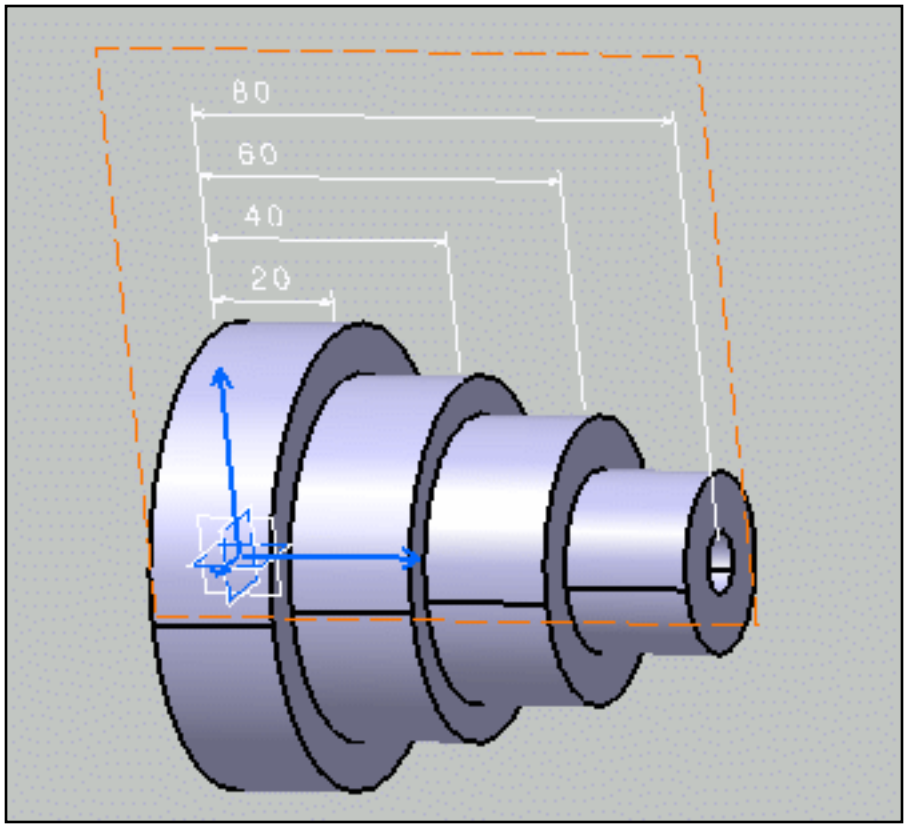


You just created a first dimension within the stacked dimensions system. You can then proceed to create as many dimensions as necessary.

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.



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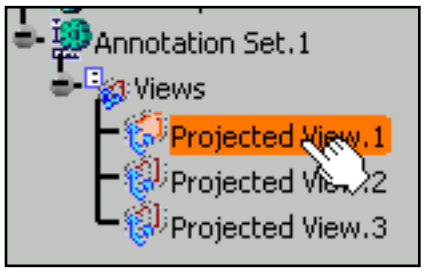
Creating Cumulated Dimensions

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

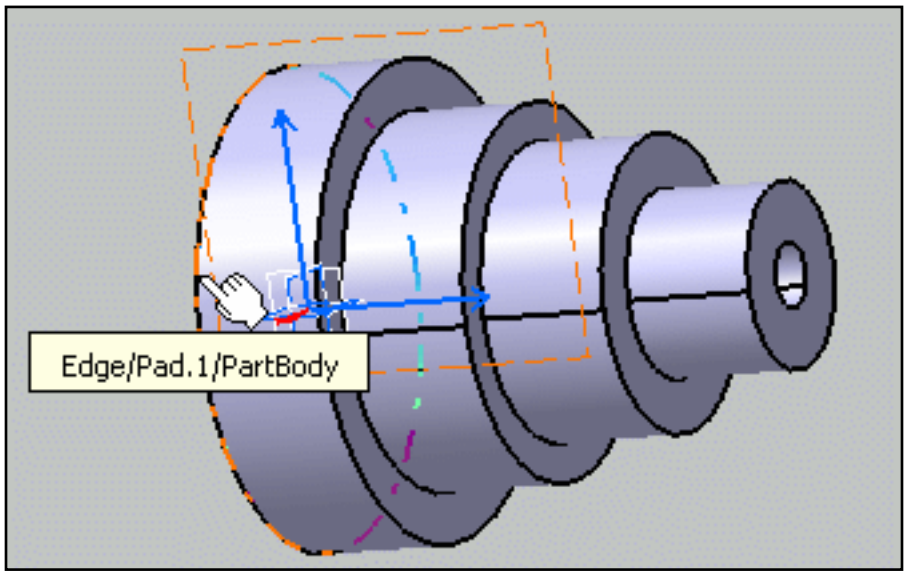
Open the Tolerancing_Annotations_10 CATPart document.

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**.

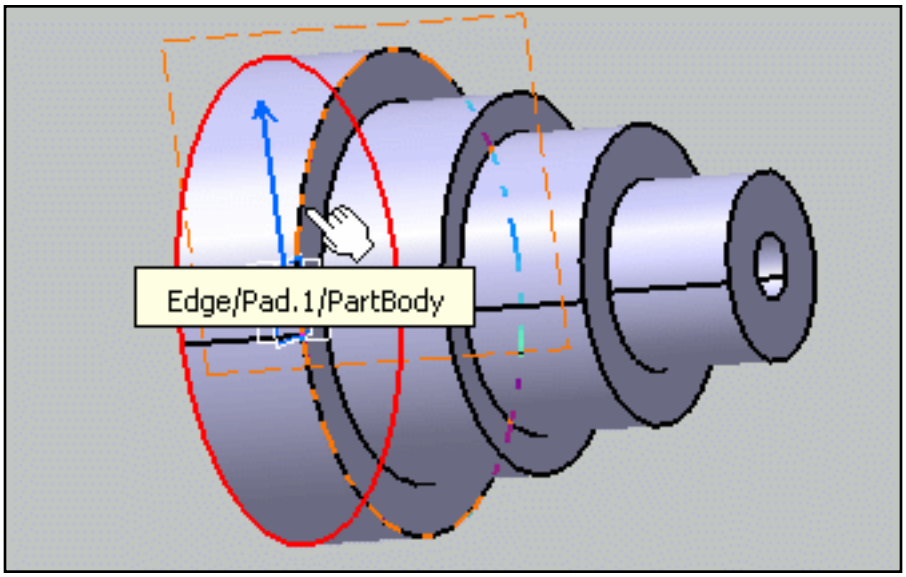
1. Activate the **Projected View.1** annotation plane.



2. Click the **Cumulated Dimensions** icon. 
3. Select the first element.

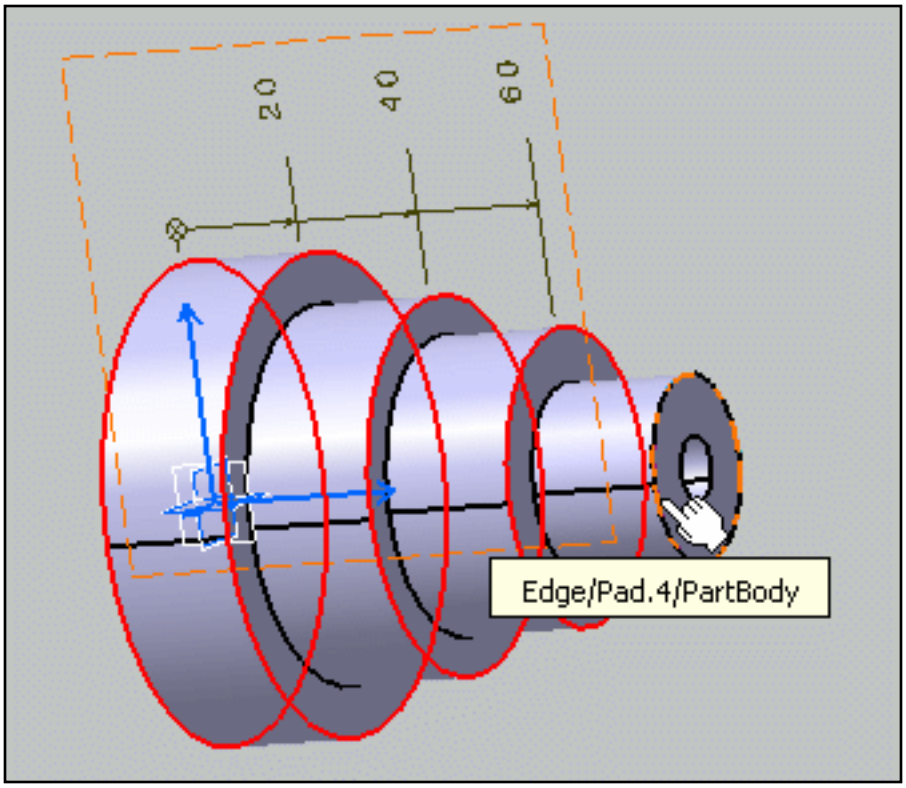


4. Select the second element.

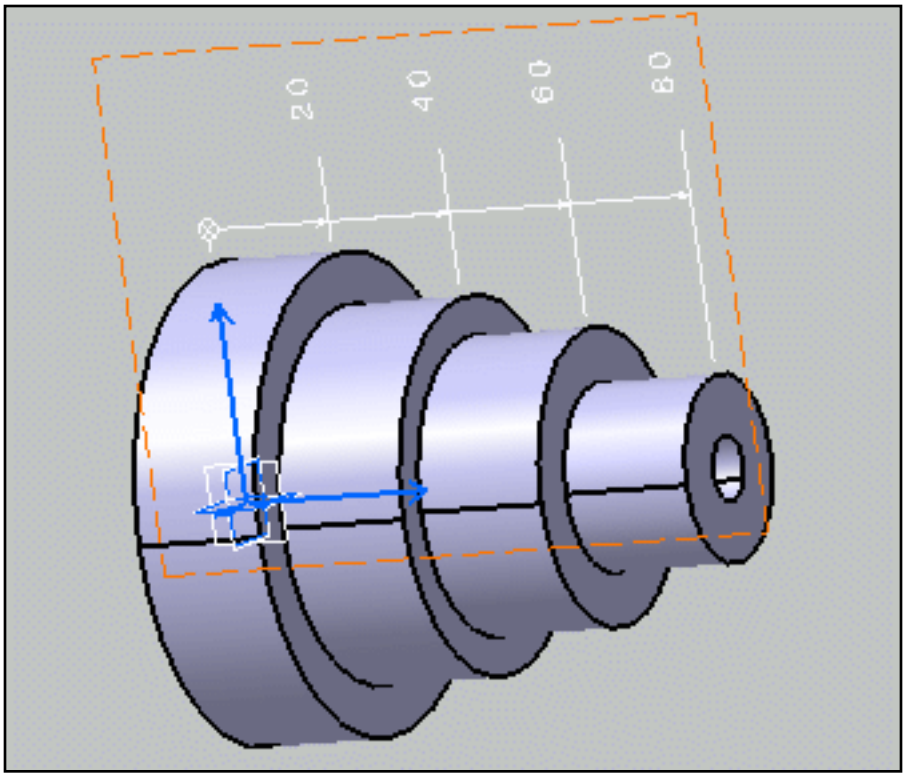


You just created a first dimension within the cumulated dimensions system. You can then proceed to create as many dimensions as necessary.

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.

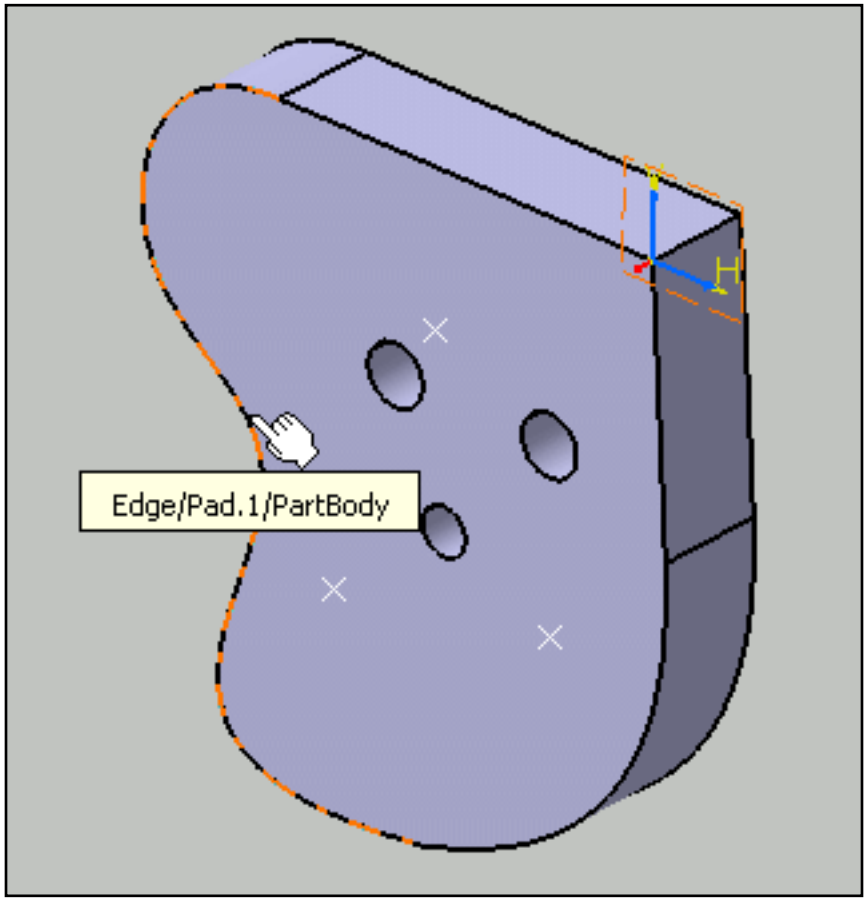
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Creating Curvilinear Dimensions

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.

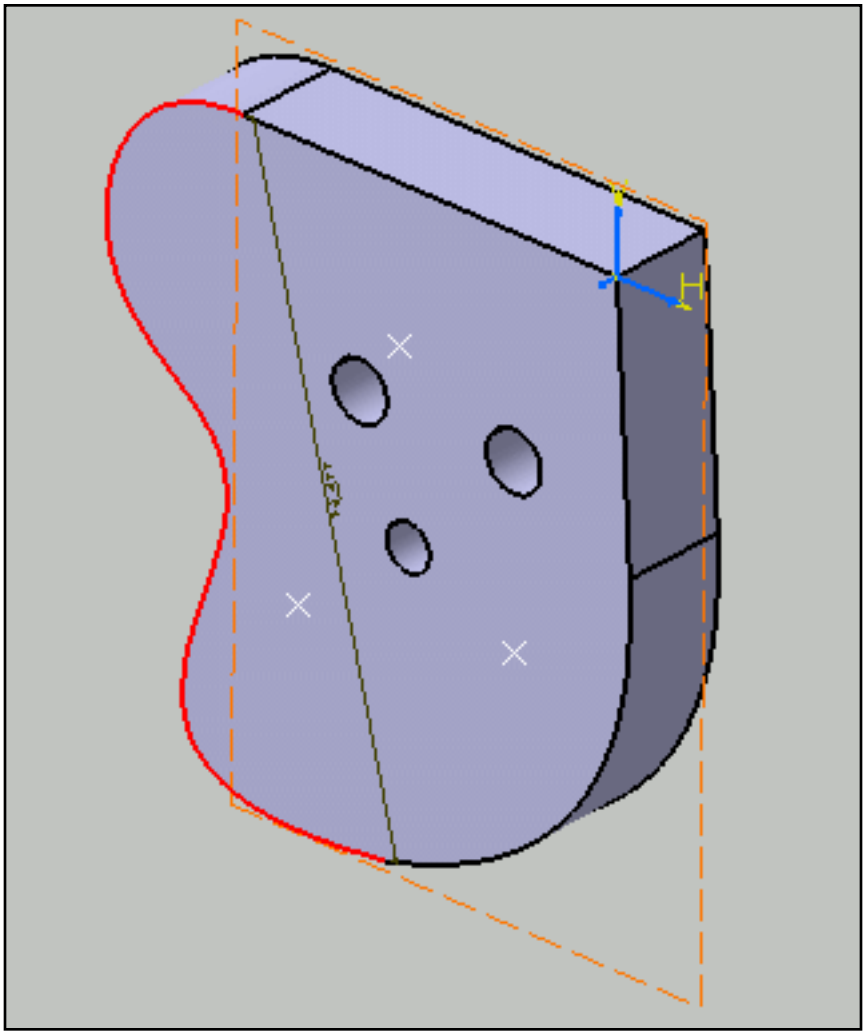
Open the Tolerancing_Annotations_11 CATPart document.

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*.

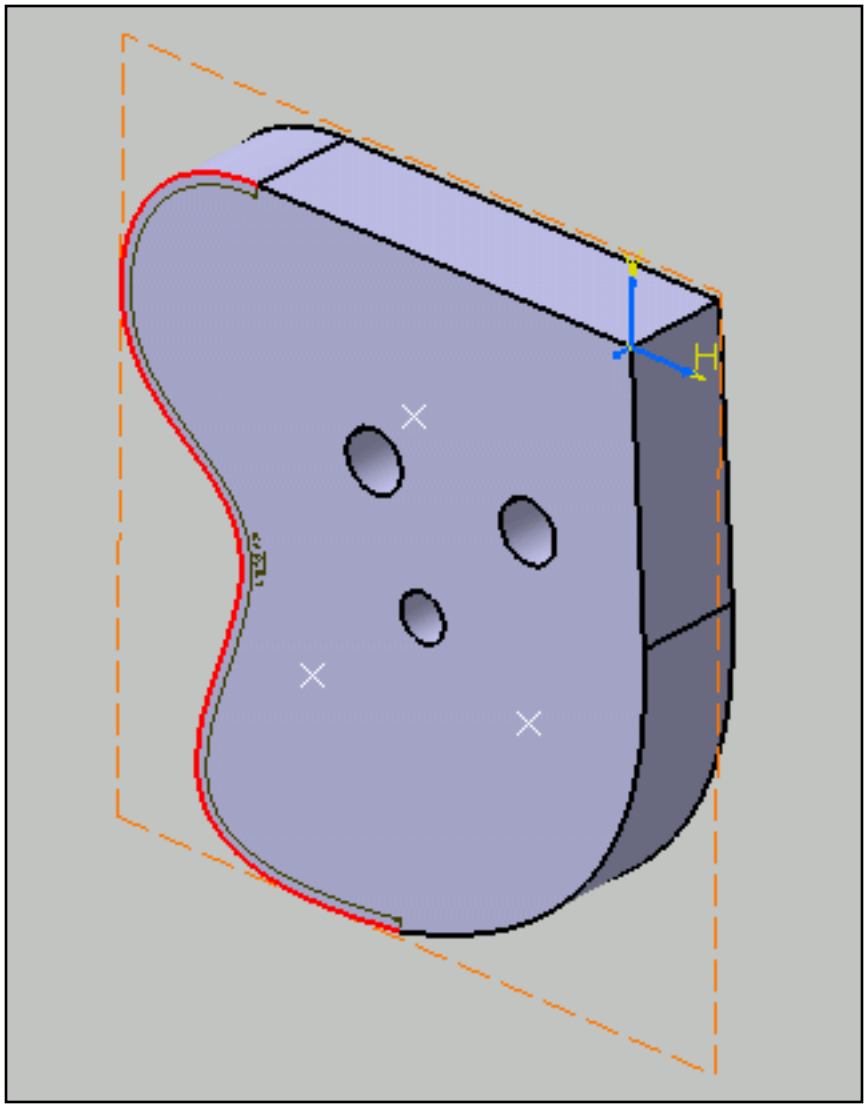
The dimension is previewed. By default, the dimension line is linear.



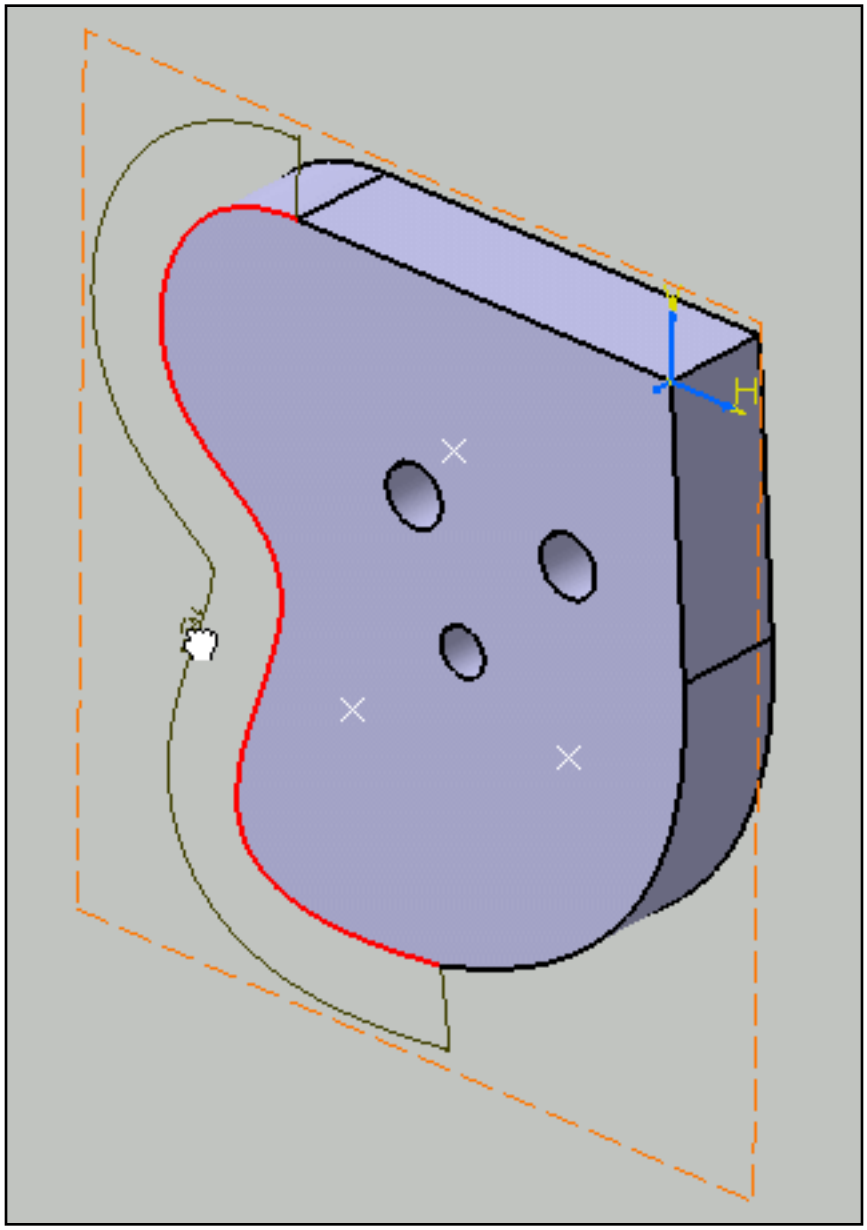
4. Right-click to display the contextual menu.
5. Select a representation mode for the dimension line:

   - **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.

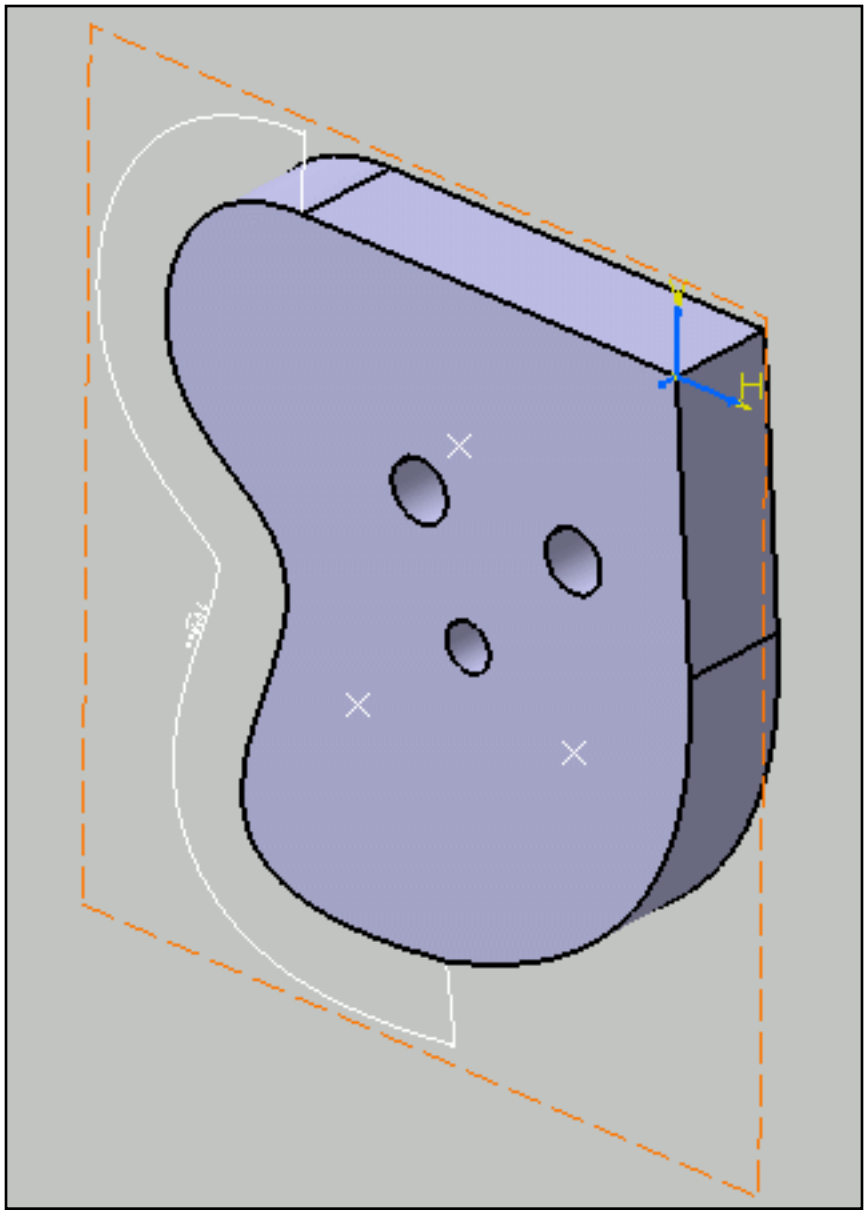
   Select **Offset** for example. The dimension line is modified accordingly.



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.



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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.

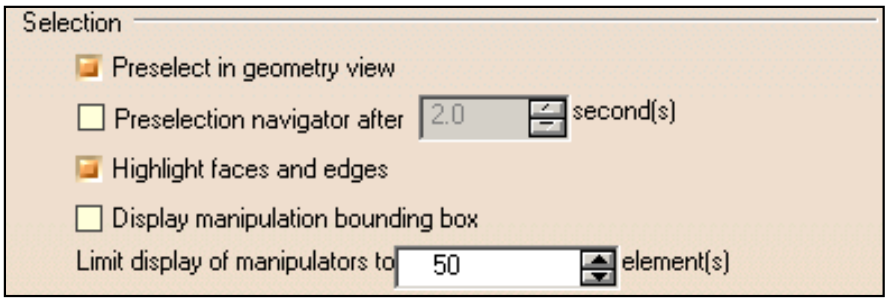
- Part Design Chamfer features; dimensions are generated according to the Part Design chamfer feature definition: for a chamfer with a length x length definition, the dimension format will be distance x distance, whereas for a chamfer with a length x angle definition, the dimension format will be distance 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.

Remember that:

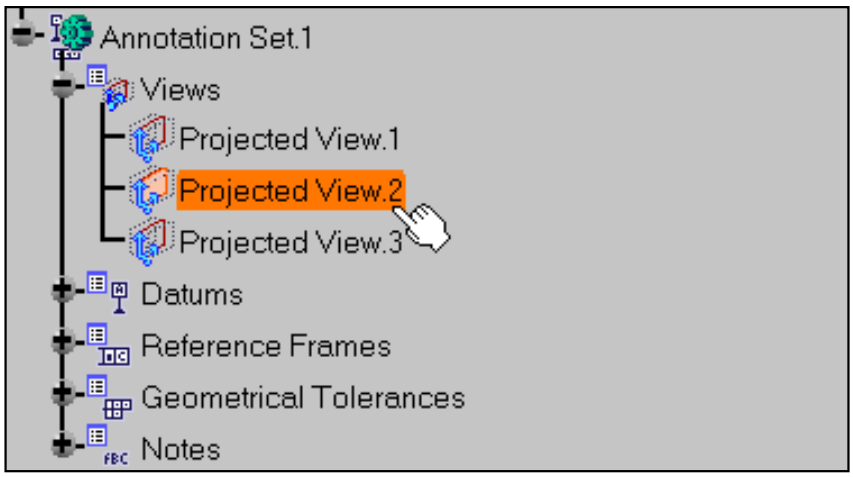
- Dimensions are associated with the design of a part, including the Mean Dimension behavior.
- When parameter tolerances are still defined, they are set to the dimension tolerances.
- Modifying dimension tolerances modifies the parameter tolerances and vice-versa.

Open the Tolerancing_Annotations_04 CATPart document.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

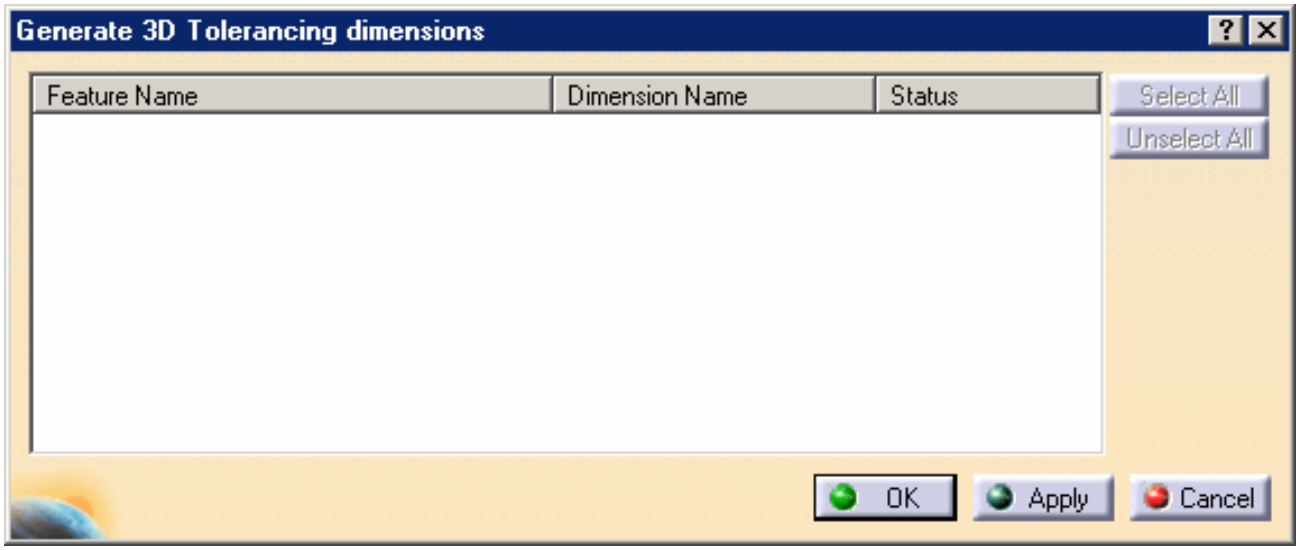


1. Activate the **Projected View.2** annotation plane.

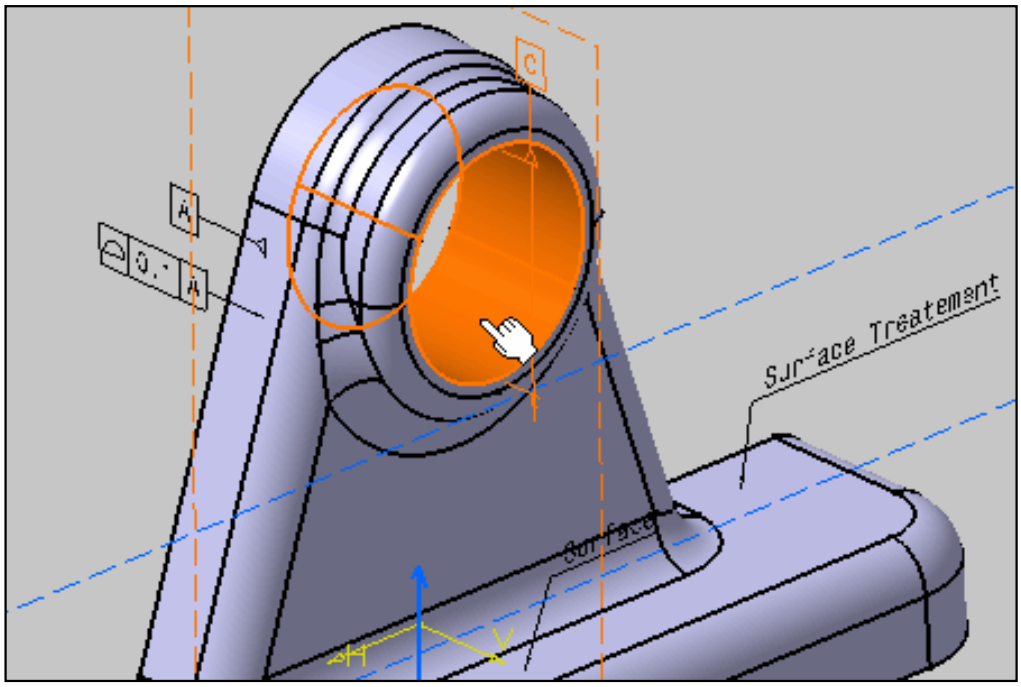


2. Click the **Generative Dimension** icon: 

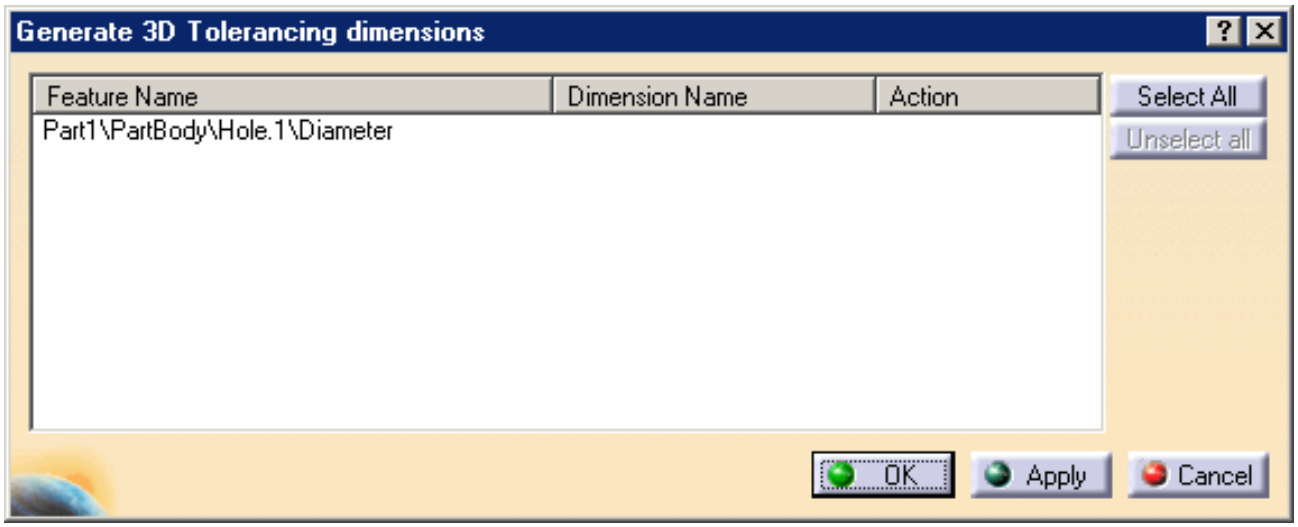
   The **Generate 3D Tolerancing Dimensions** dialog box appears.



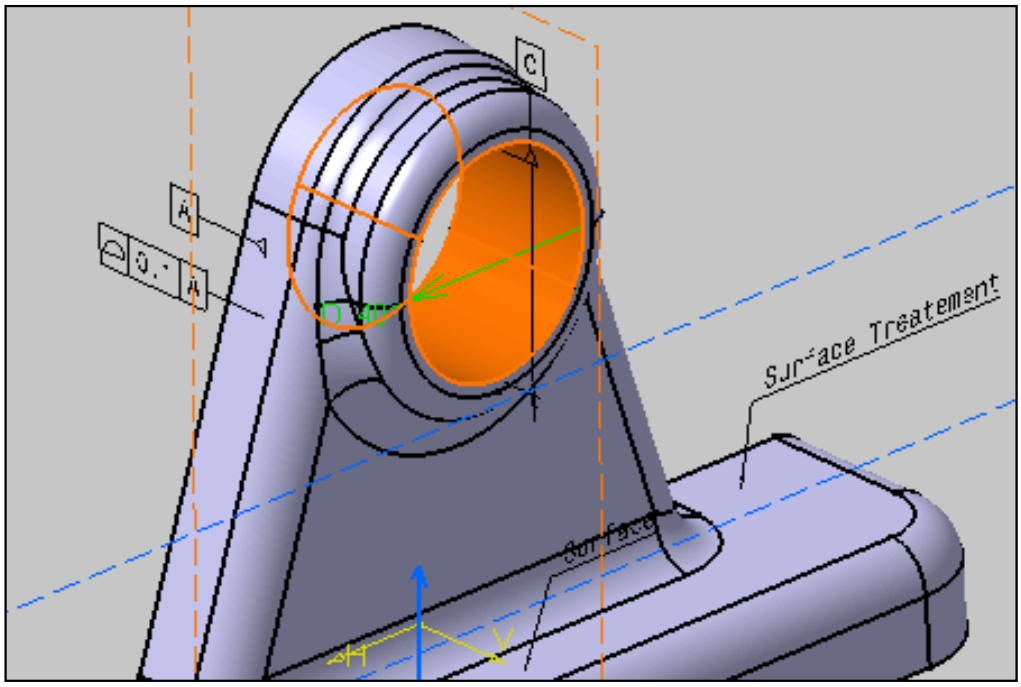
3. Select the hole as shown on the part.



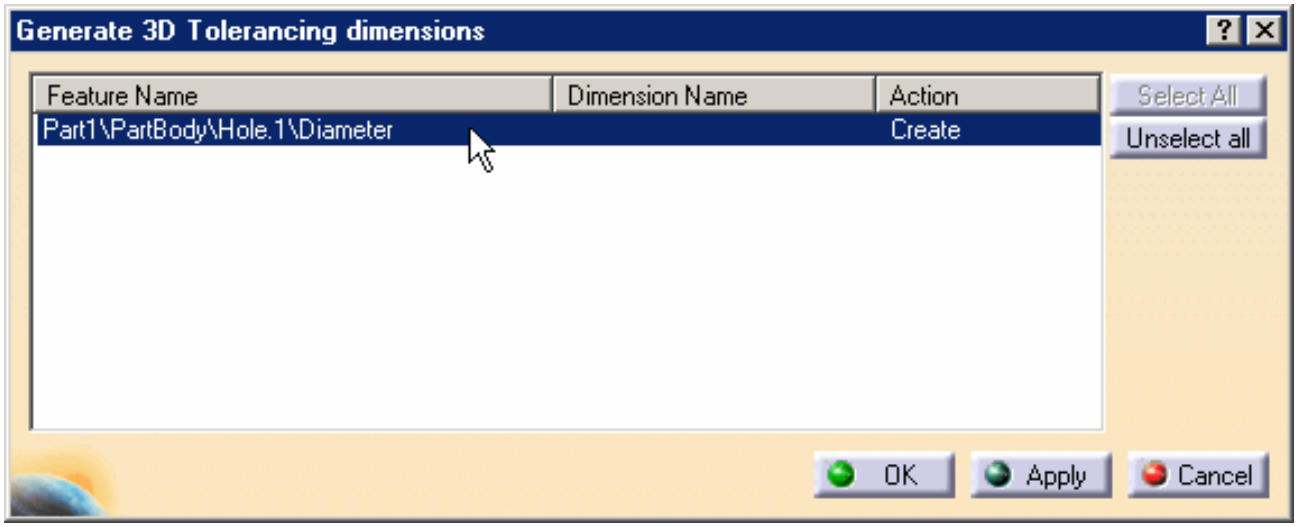
The hole parameter is displayed in the dialog box.



Also, the hole parameter is shown on the part.

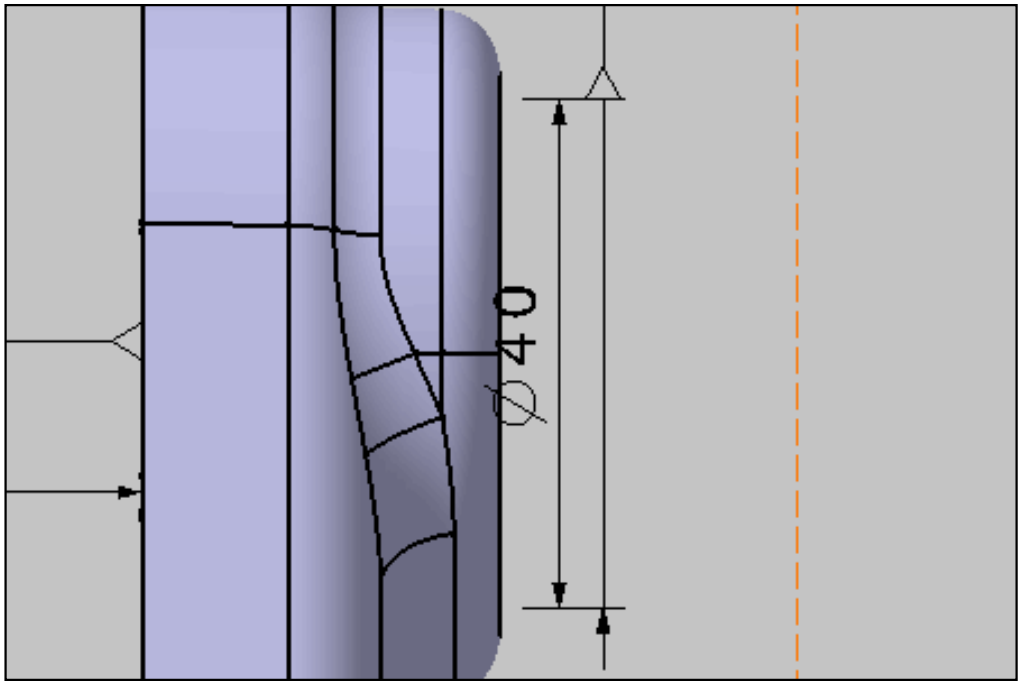


4. Select the **Diameter** parameter in the dialog box. You can also select the parameter on the geometry.

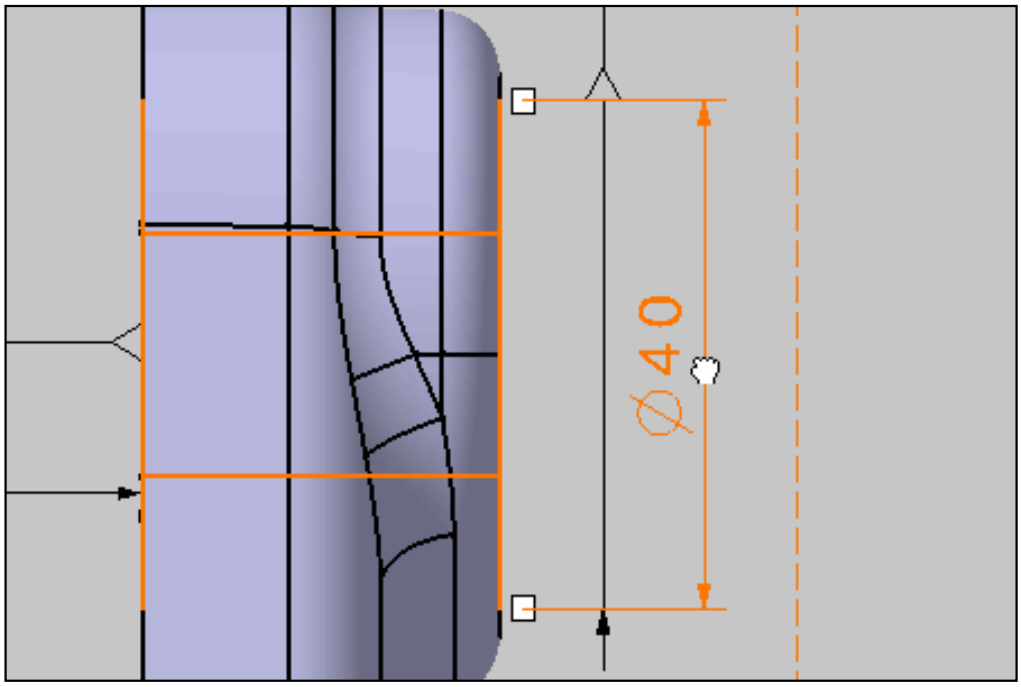


5. Click **OK** in the dialog box.

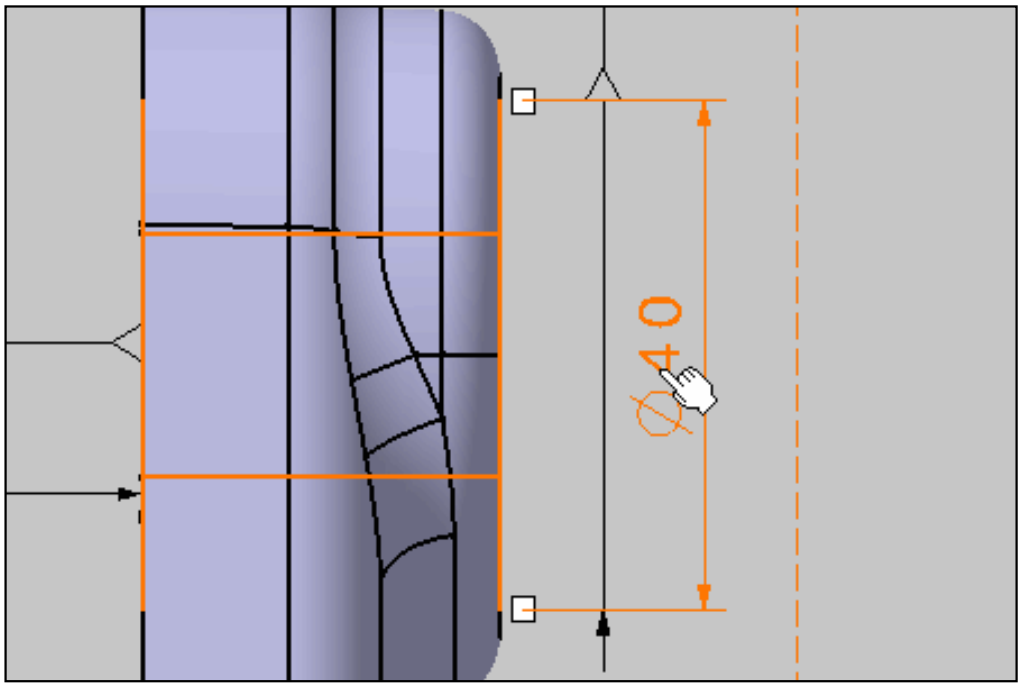
The diameter dimension is created.



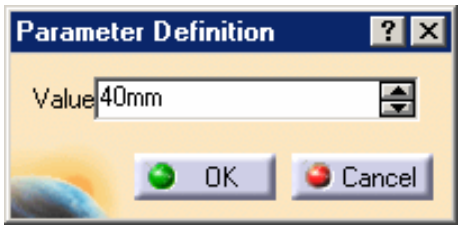
6. Drag the dimension.



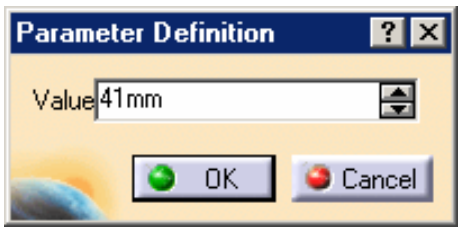
7. Right-click the dimension and select **Edit Generative Parameter** from the contextual menu.



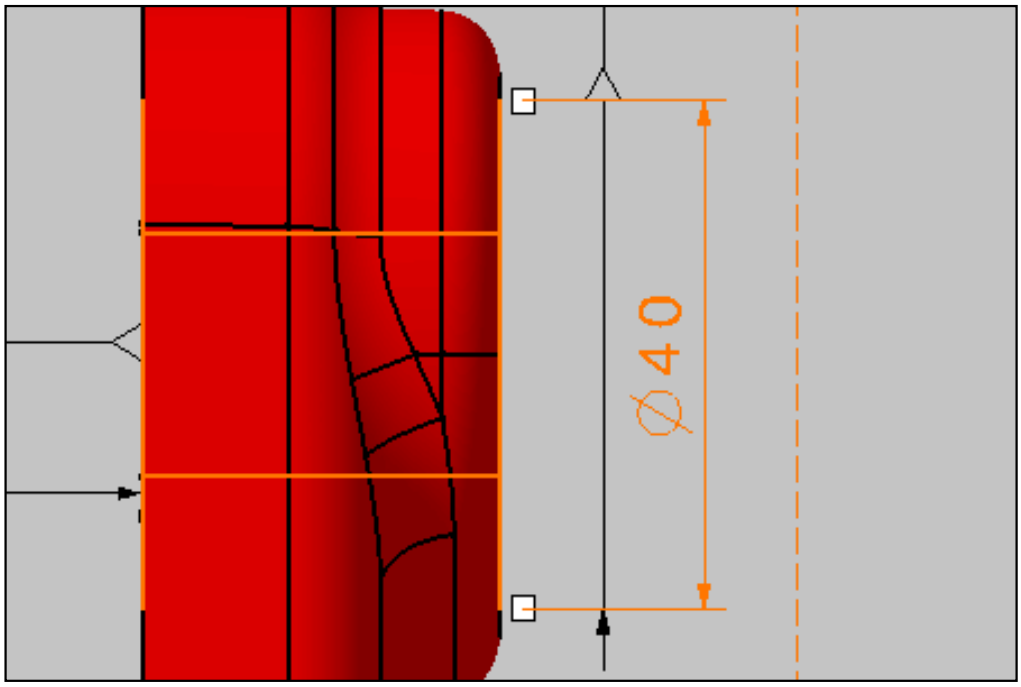
The **Parameter Definition** dialog box appears.
This dialog box allows you to modify the hole parameter and edit the part.



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

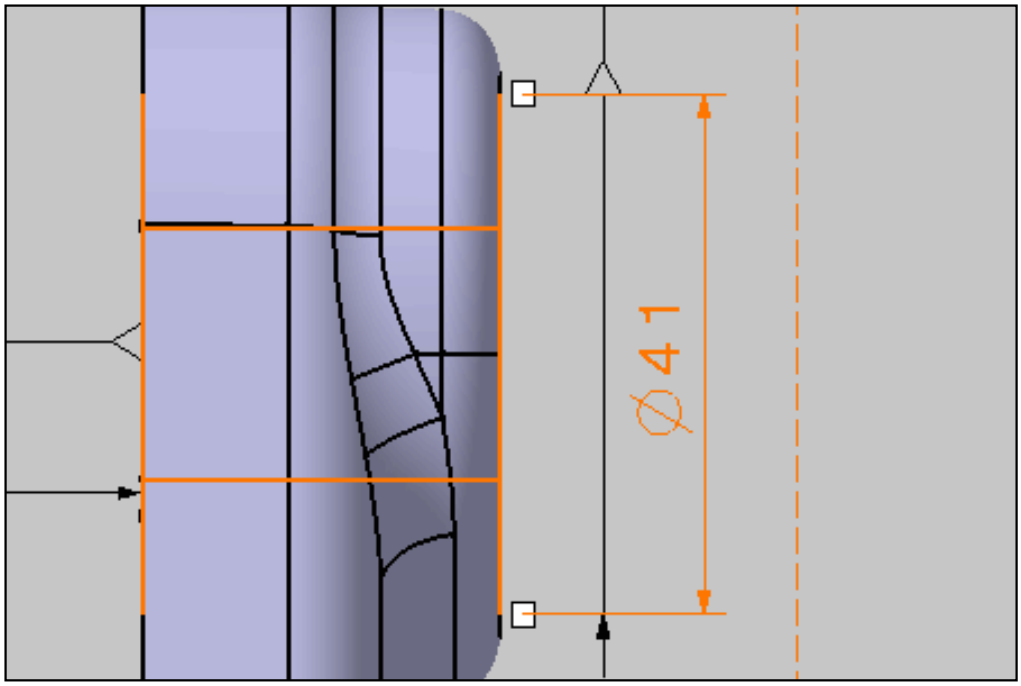


The part turns red because it has been modified but not updated.



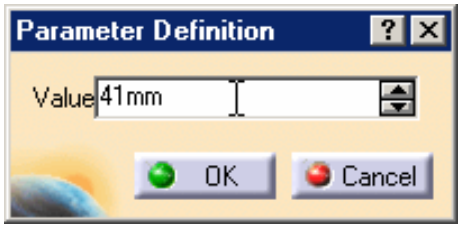
8. Click the **Update** icon: 

   The part's hole and diameter dimension are modified.

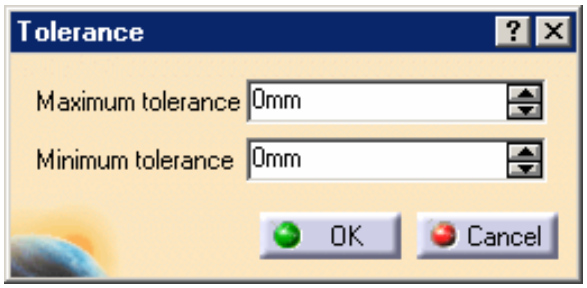


9. Right-click the dimension and select **Edit Generative Parameter** from the contextual menu again.

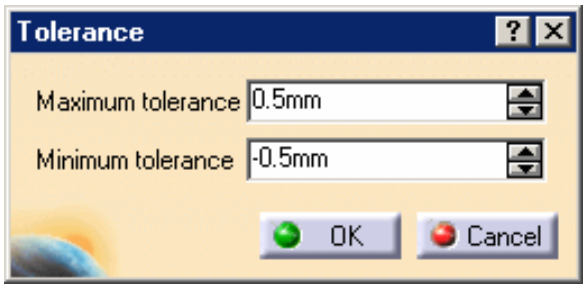
10. Right-click in the value field and select the **Add Tolerance...** from contextual menu.



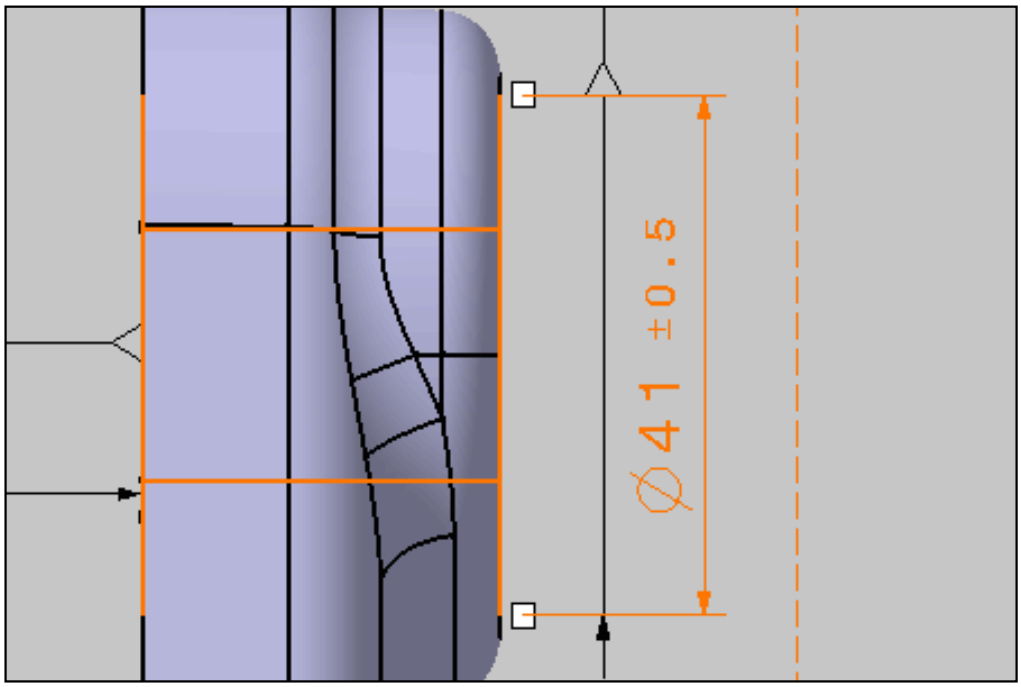
The **Tolerance** dialog box appears.
This dialog box allows you to modify the hole parameter tolerances.



11. Set the tolerances to 0.5 and -0.5

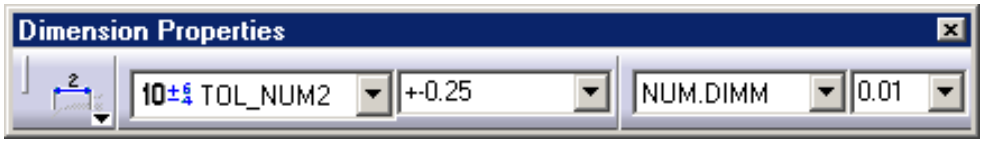


Tolerances are displayed on the dimension.

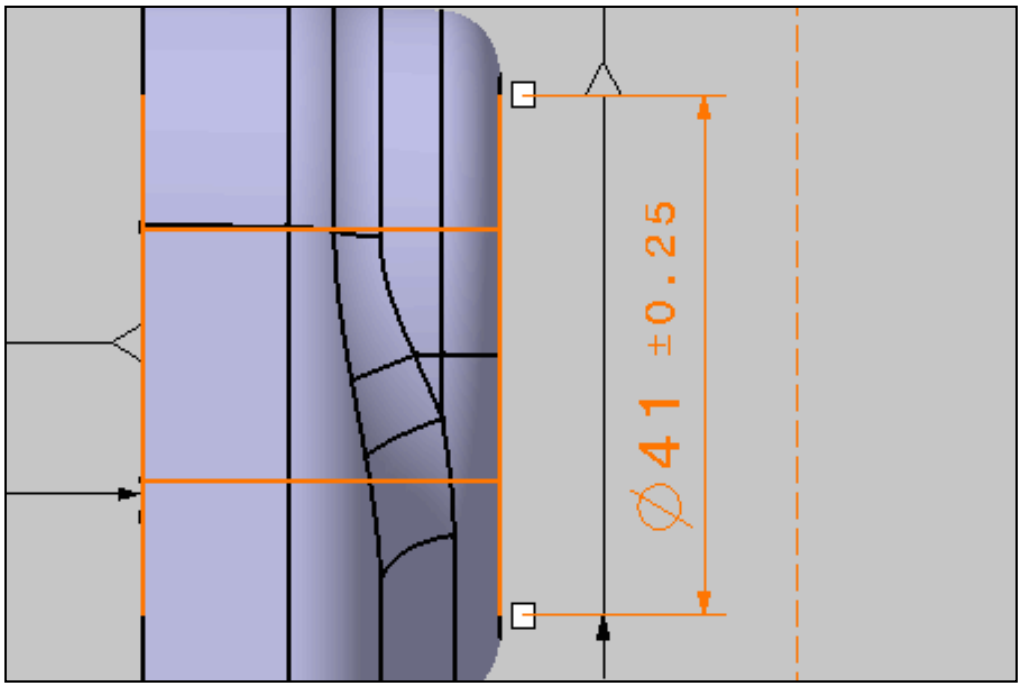


12. Click **OK** in the **Tolerance** and **Parameter Definition** dialog boxes.

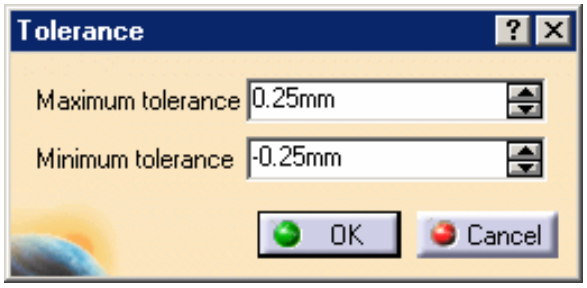
13. Modify the tolerances in the **Dimension Properties** toolbar to +-0.25.



Tolerances are updated on the dimension.



The parameter's tolerances are also updated.



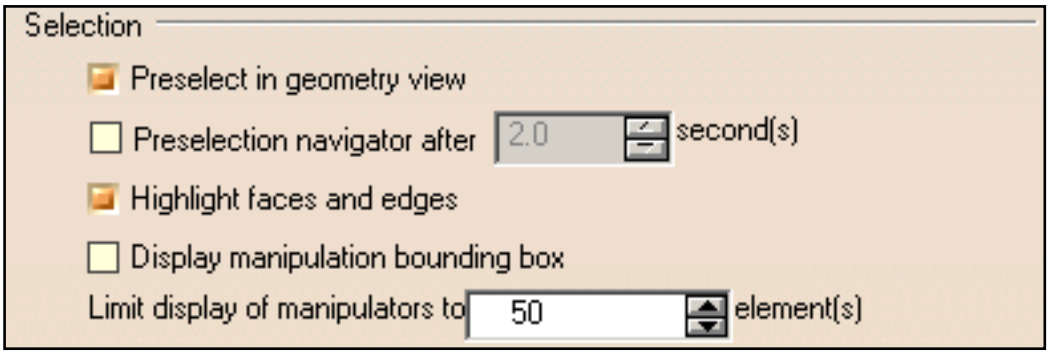
Back Up Next

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.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



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.

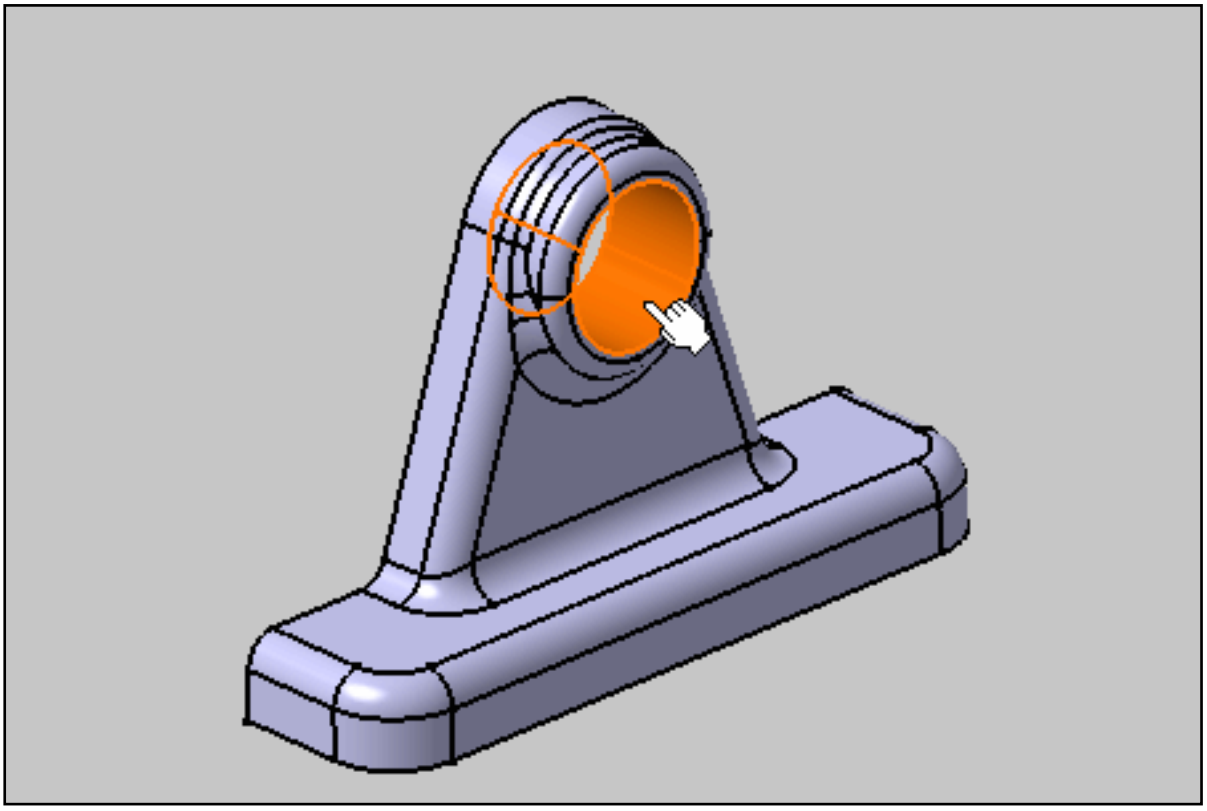


3. Double-click the **Note Object Attribute** component family item.

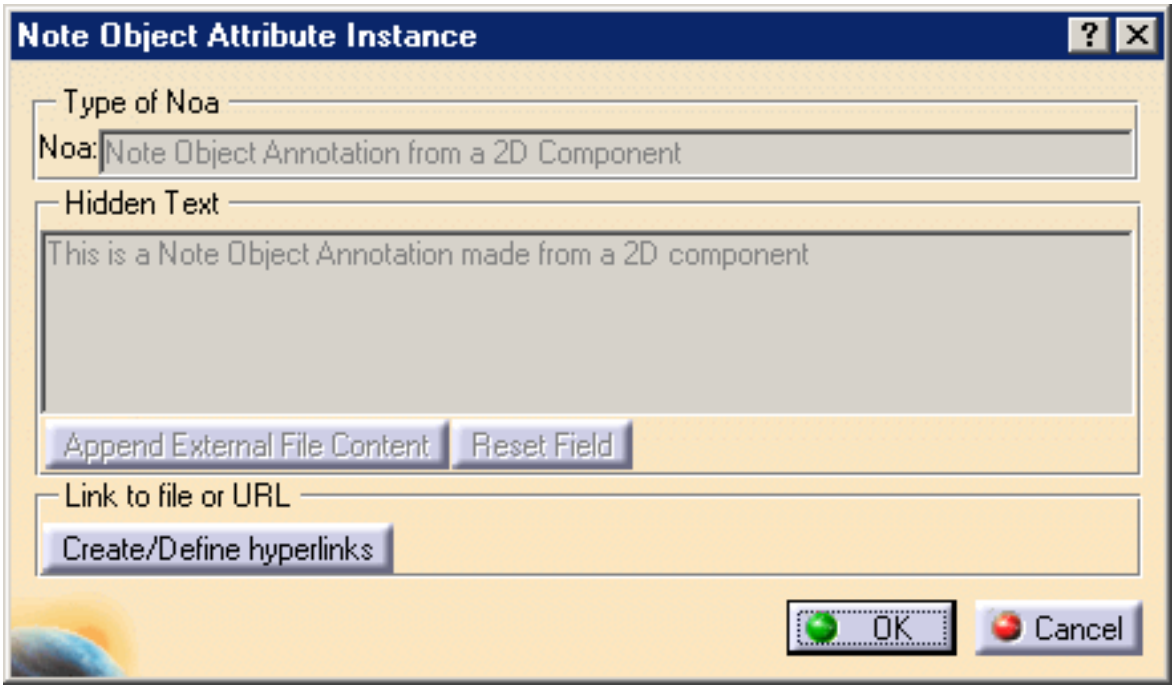
4. Double-click the **Note Object Attribute Using 2D Component** component item.

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.

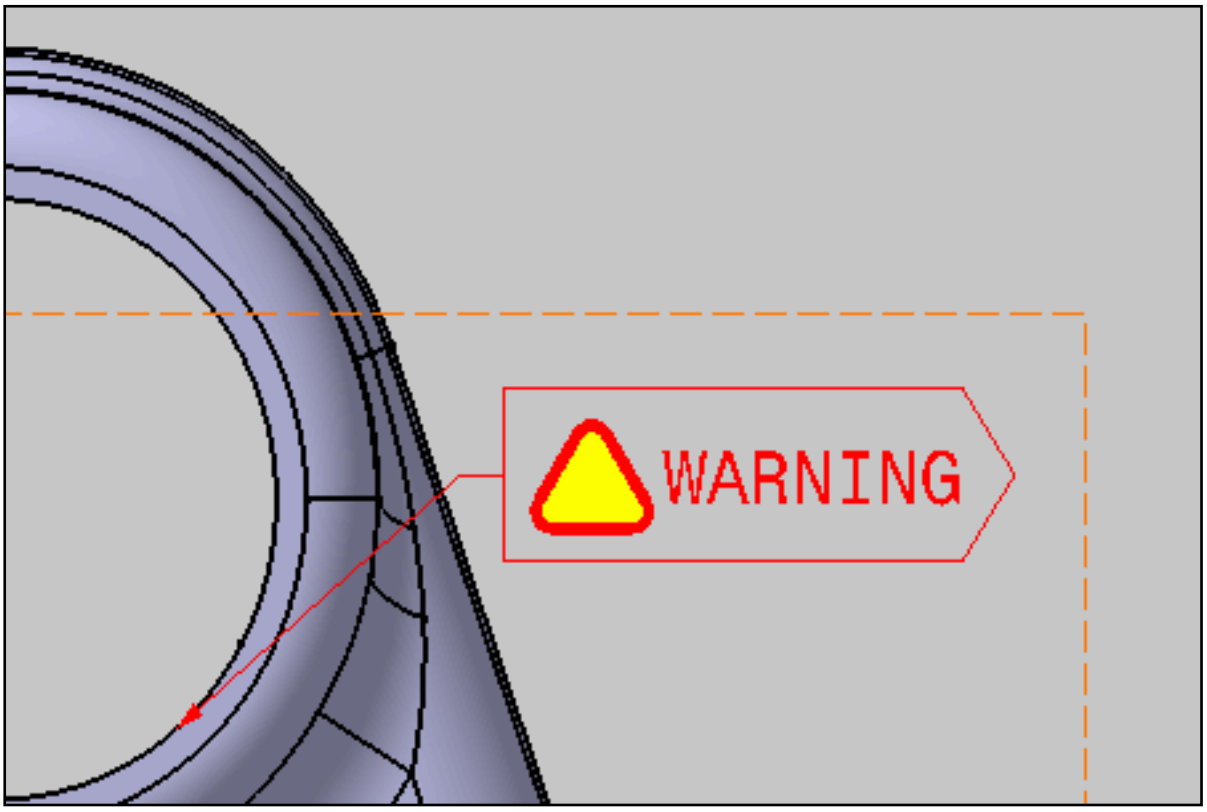


The **Note Object Attribute Instance** dialog box appears. The hidden text specified with the note object attribute is not modifiable. To unlock it, see Creating from a 2D Component; in this case, the dialog box is enabled.



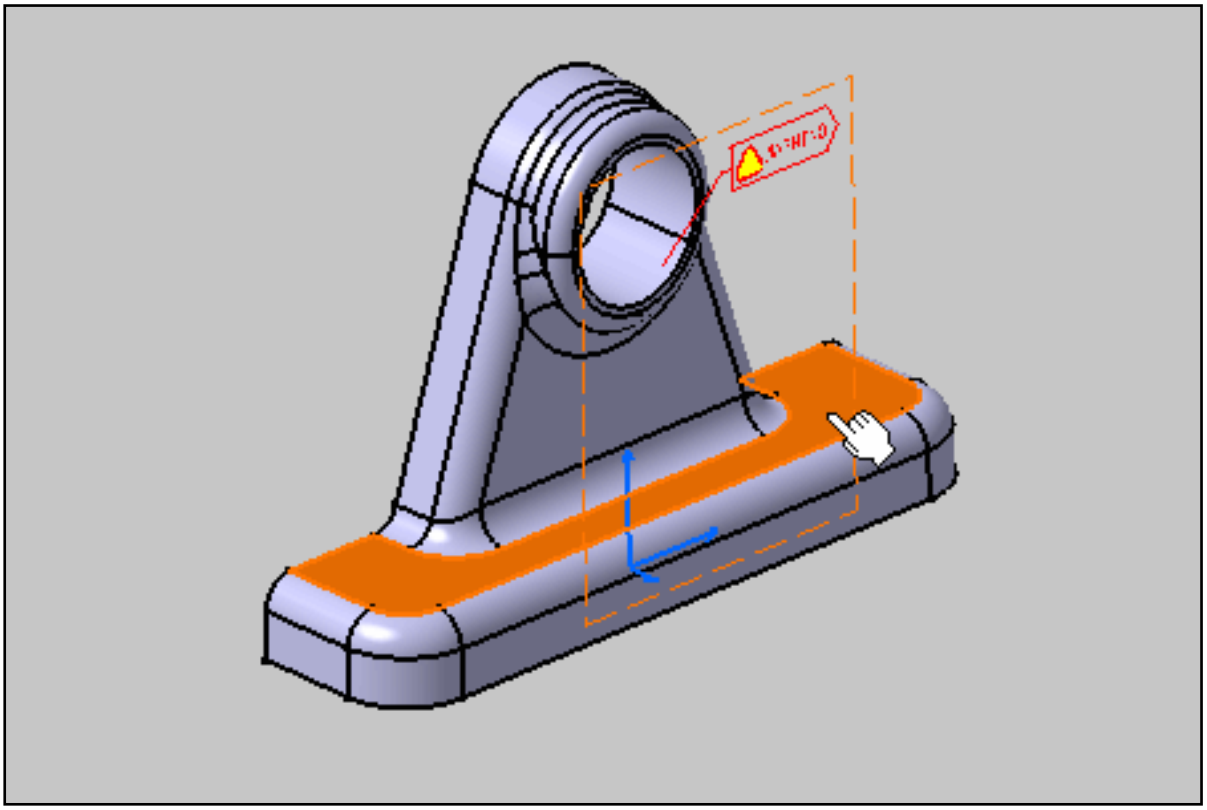
6. Click **OK**.

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

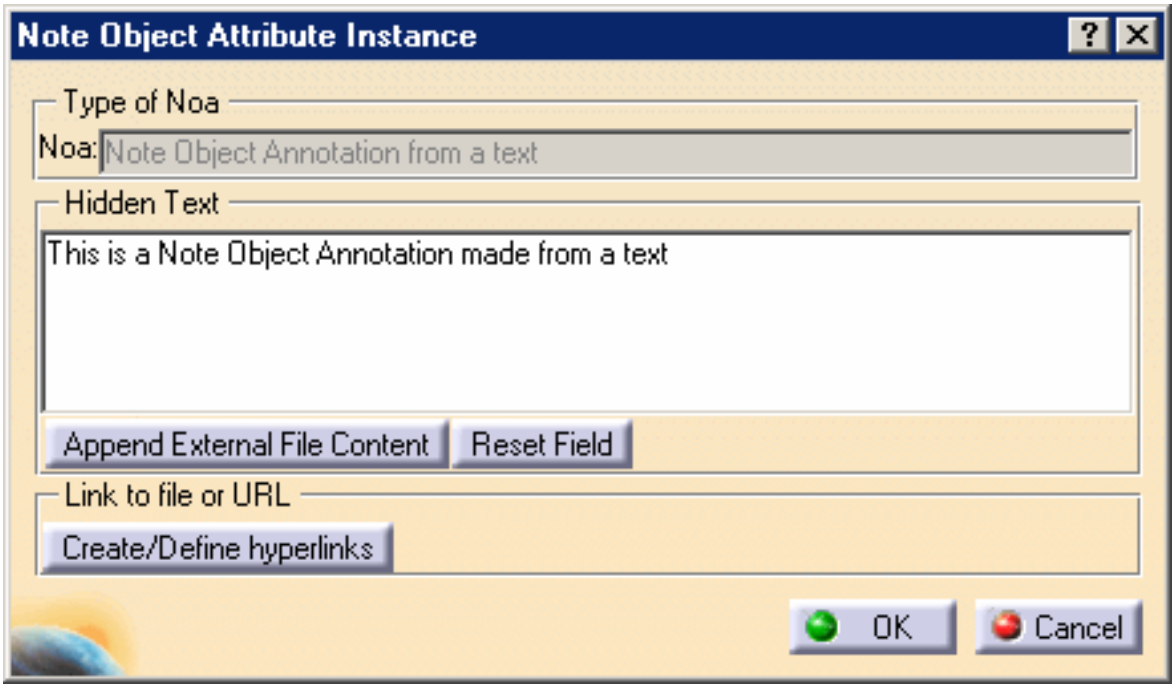


7. Double-click the **Note Object Attribute Using Text** component item in the **Catalog Browser** dialog box.

8. Select the surface as shown on the part.

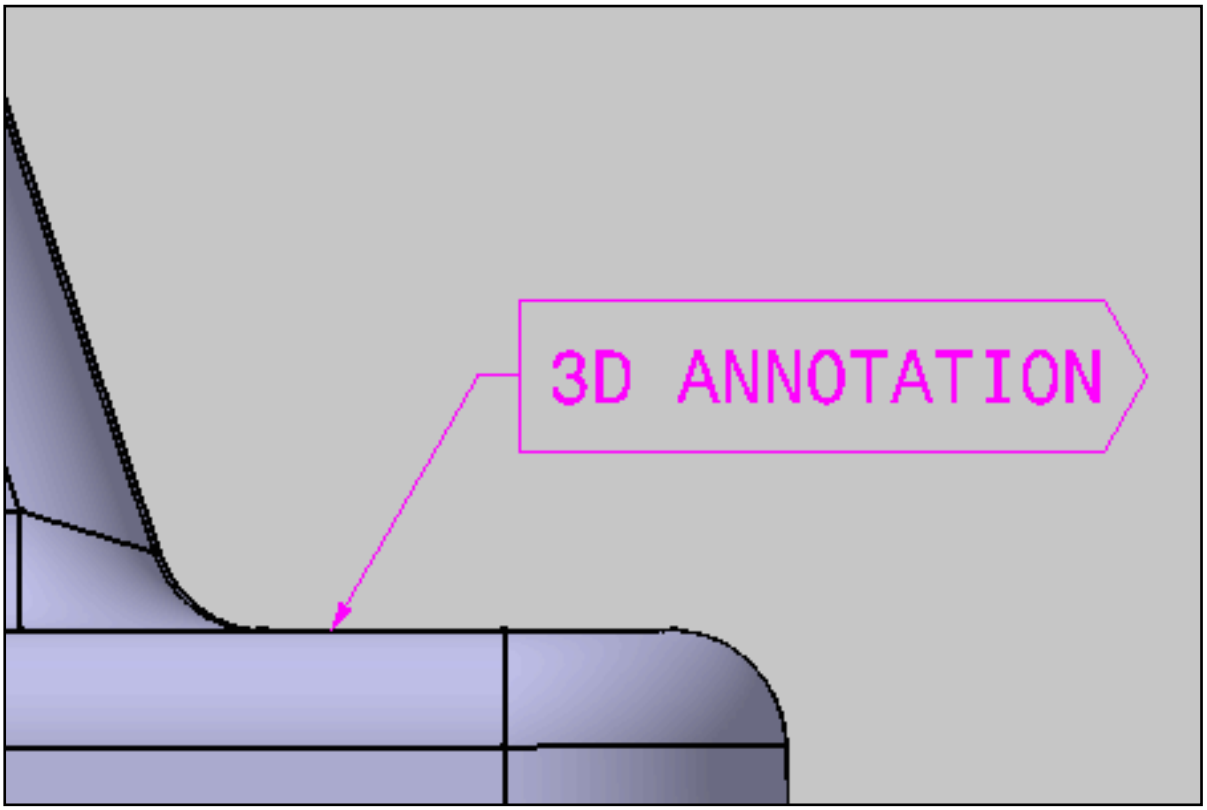


The **Note Object Attribute Instance** dialog box appears. The hidden text specified with the note object attribute is modifiable. To lock it, see Creating from a Text; in this case, the dialog box is disabled.



9. Click **OK**.

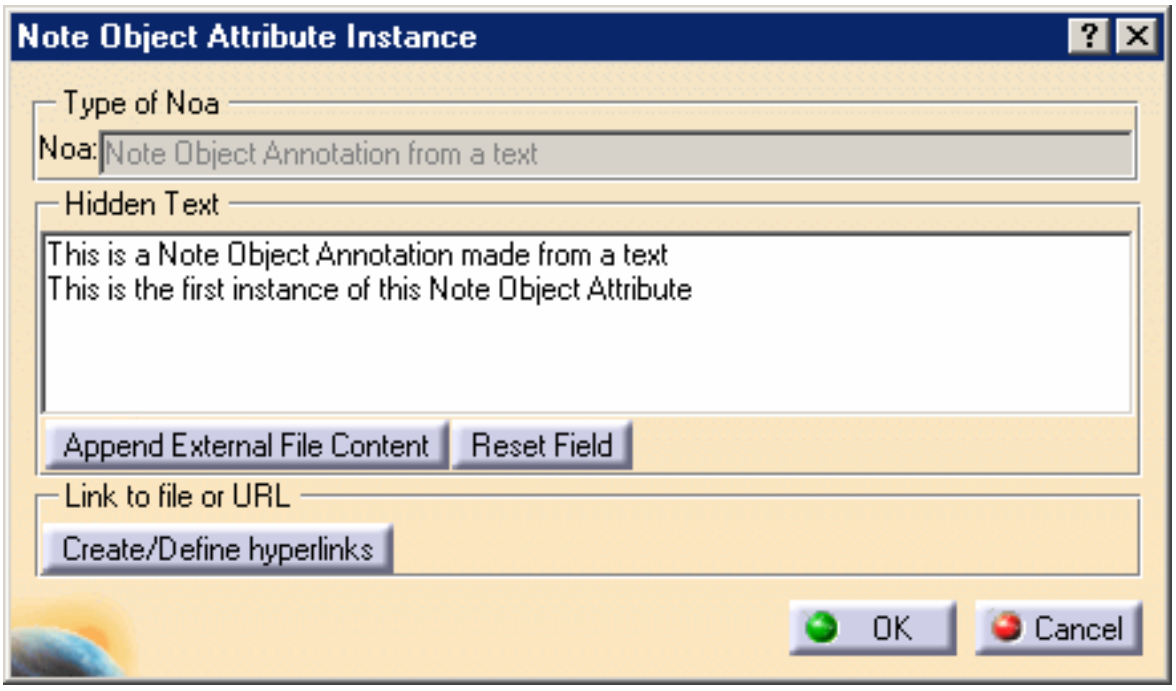
The Note Object Attribute is instantiated.



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

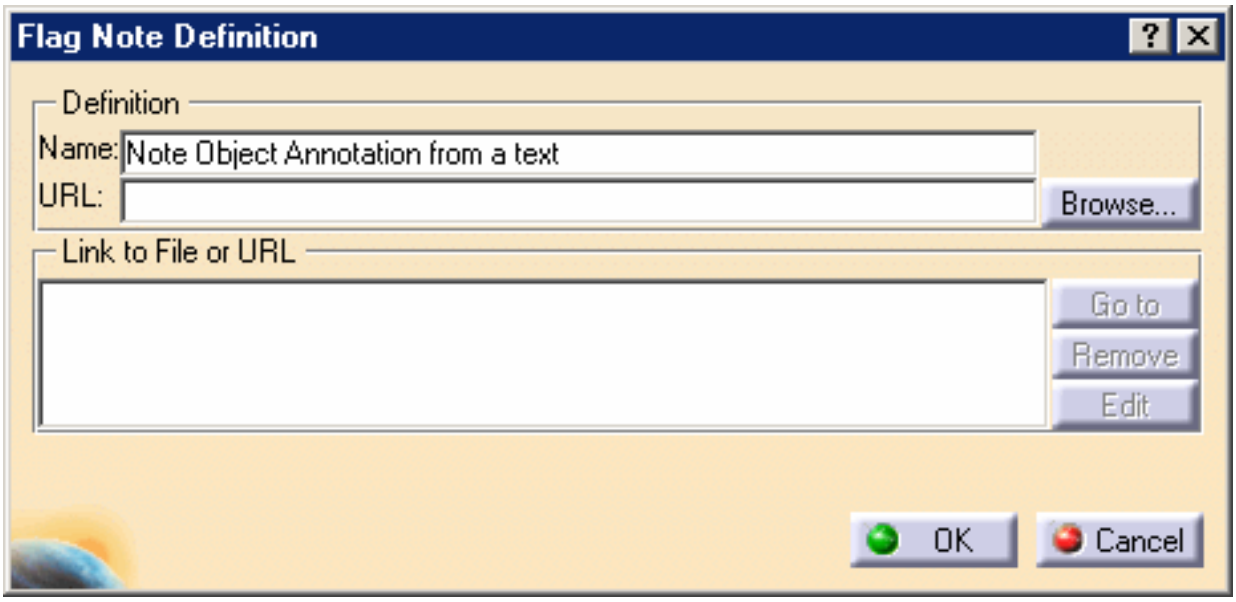
   The **Note Object Attribute Edition** dialog box appears.

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

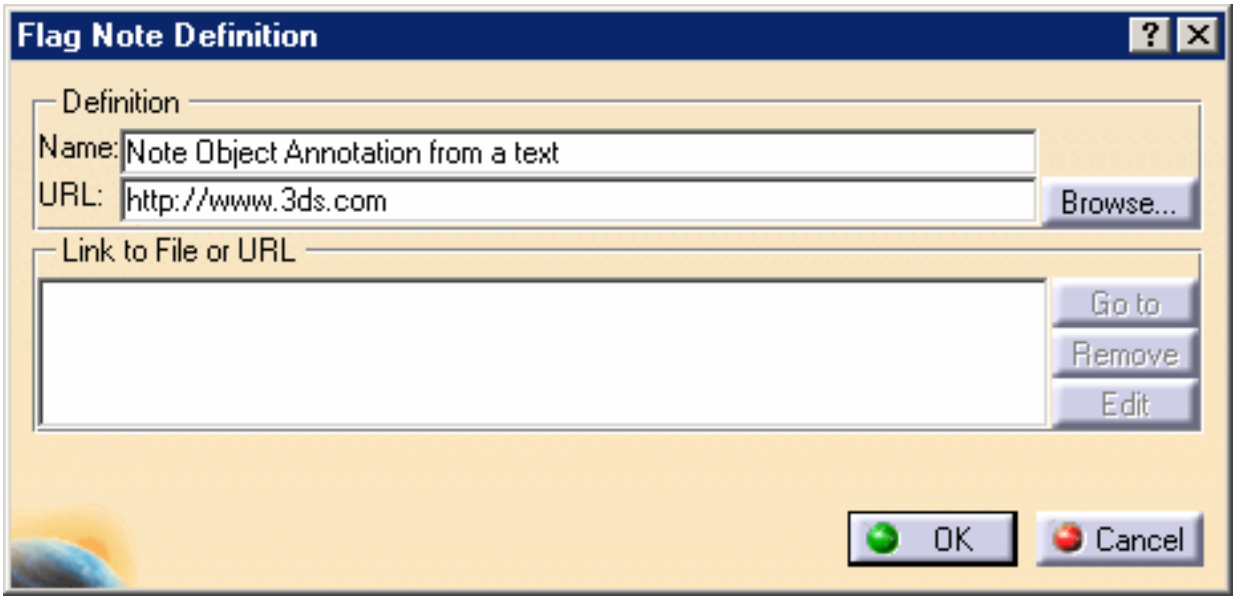


9. Click **Create / Define hyperlinks**.

   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.



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Creating a Partial Surface

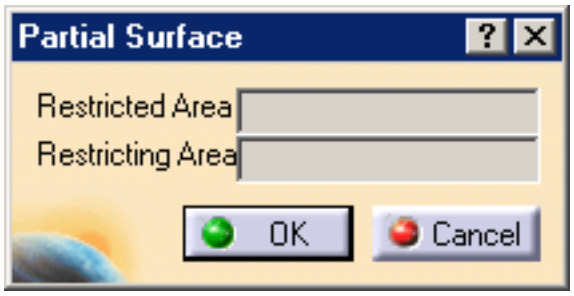
This task shows you how to create a partial surface annotation.

A partial surface annotation allows user to define a delimited surface to be toleranced.
The surface may be defined using Generative Shape Design workbench.

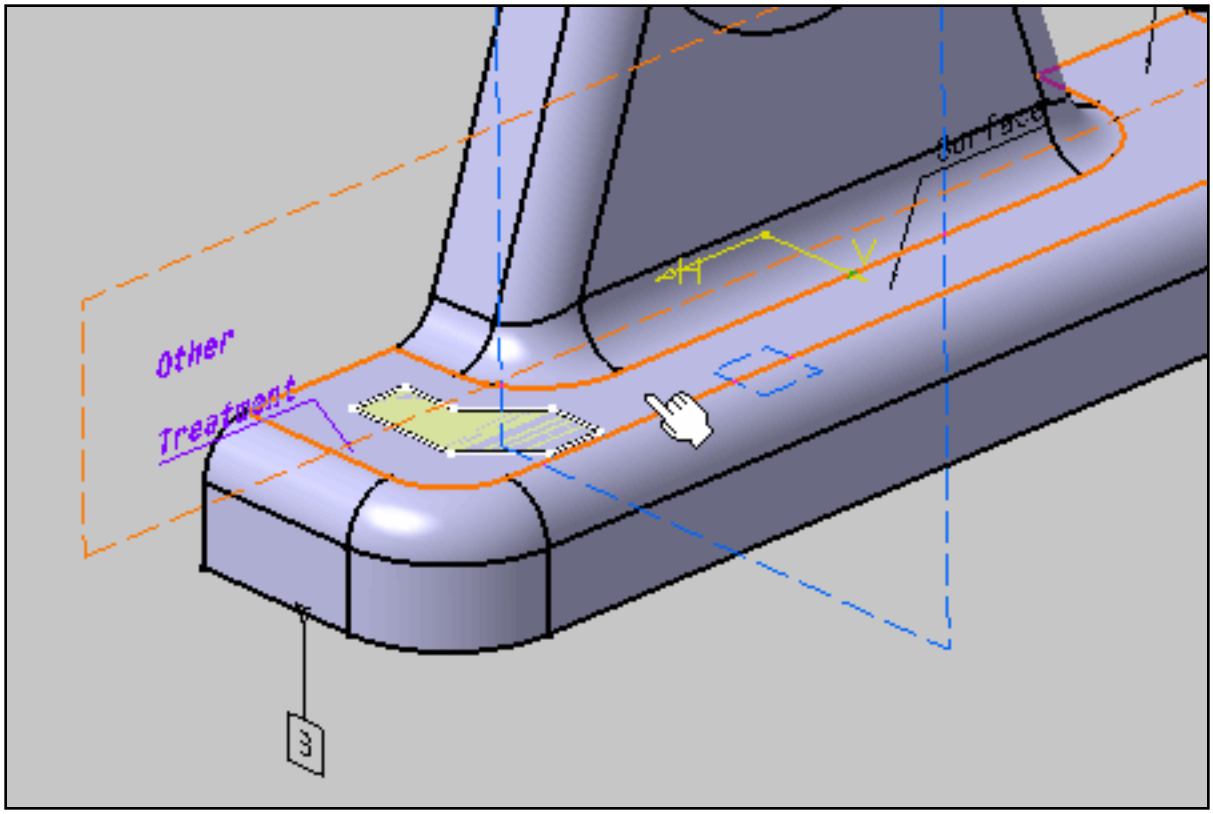
Open the Tolerancing_Annotations_04 CATPart document.

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

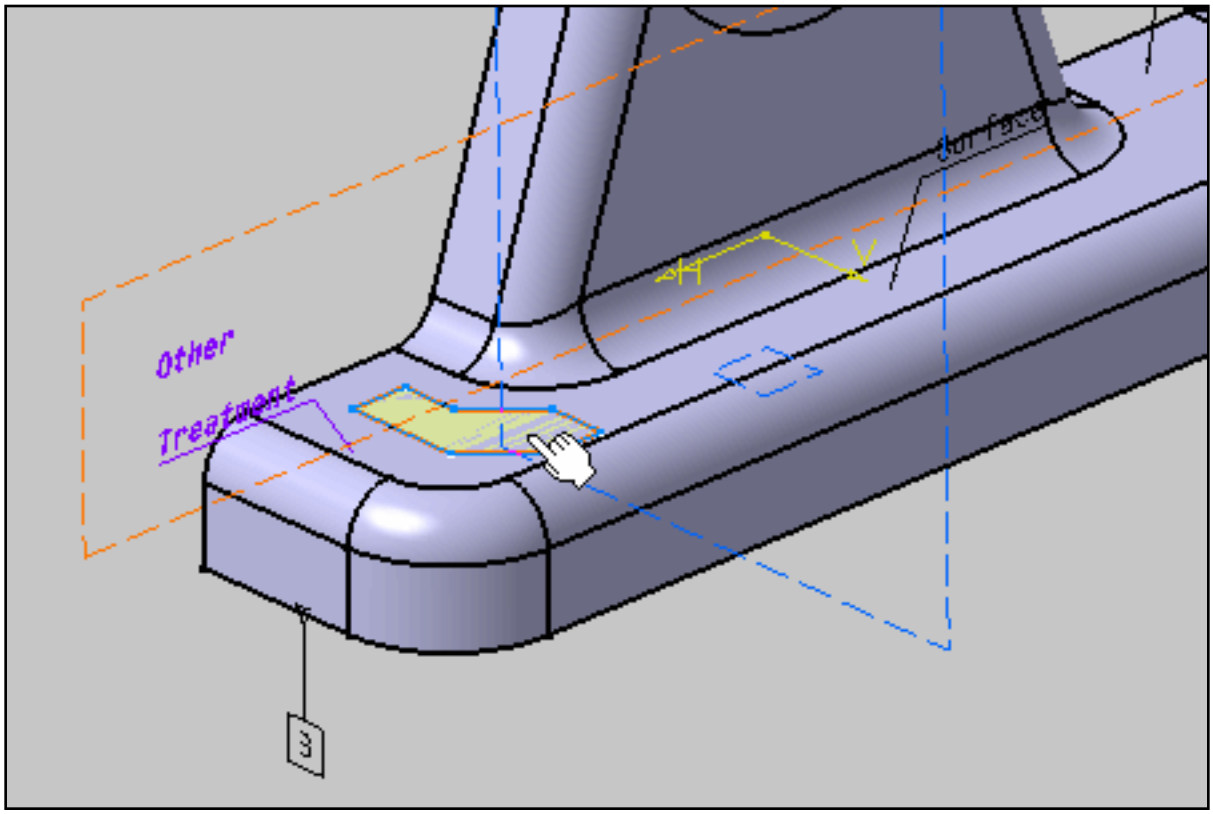
   The **Partial Surface** dialog box appears.

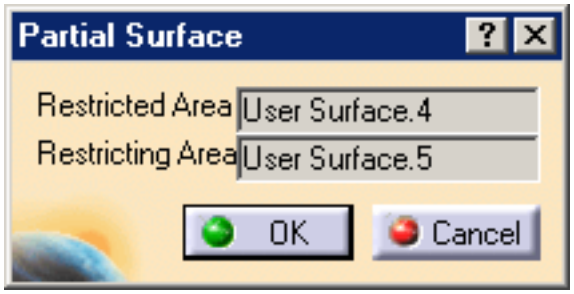


2. Select the restricted surface as shown on the part.



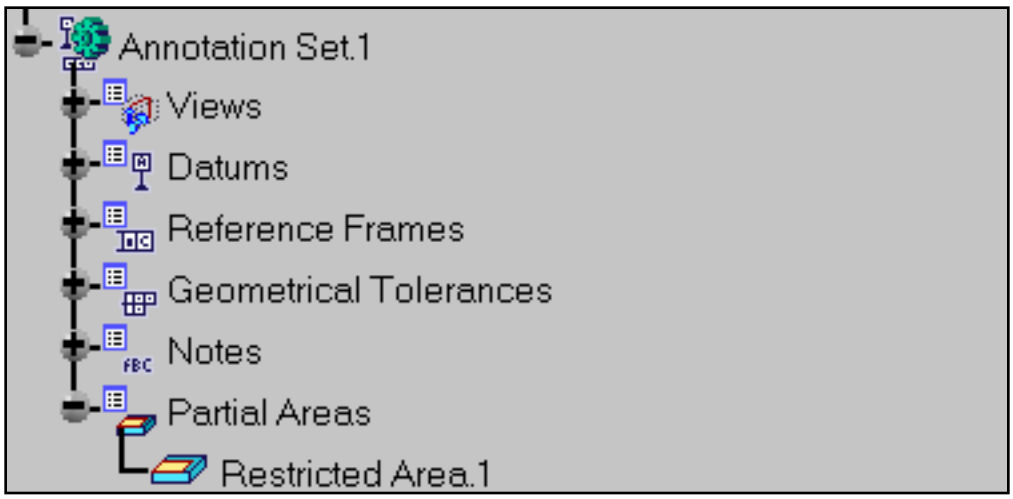
3. Select the restricting surface as shown on the part.





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

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



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Creating a Deviation

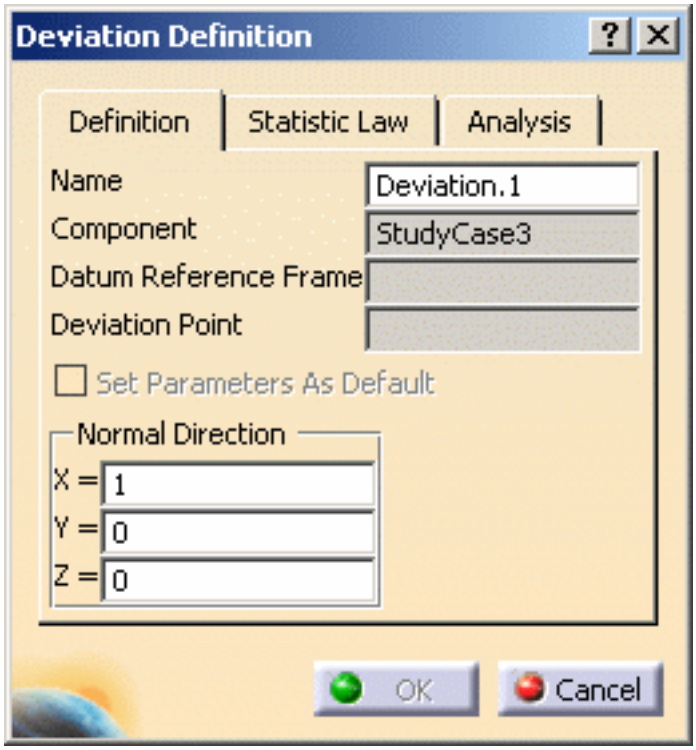
This task will show you how to create a deviation annotation on a part or product. This annotation is used in the Tolerance Analysis of Deformable Assembly workbench.

- 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.

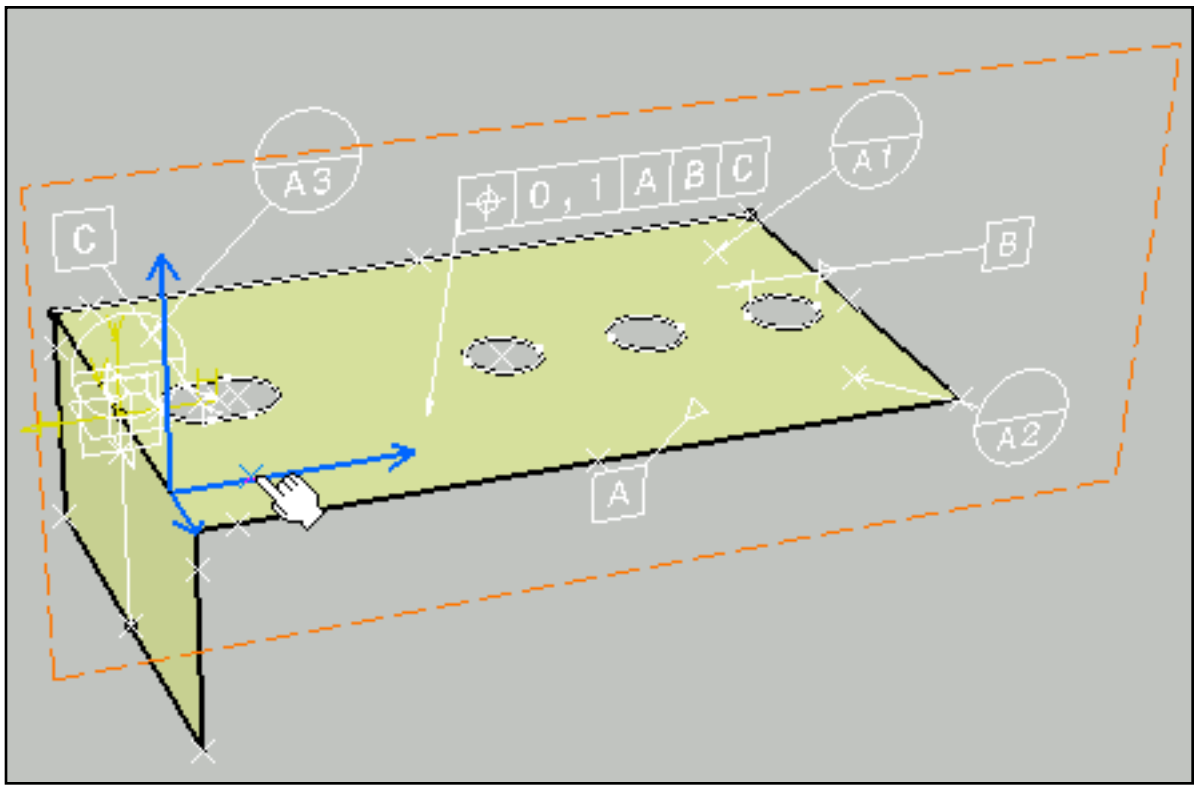
1. Click the **Deviation** icon  in the Deviations toolbar.

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

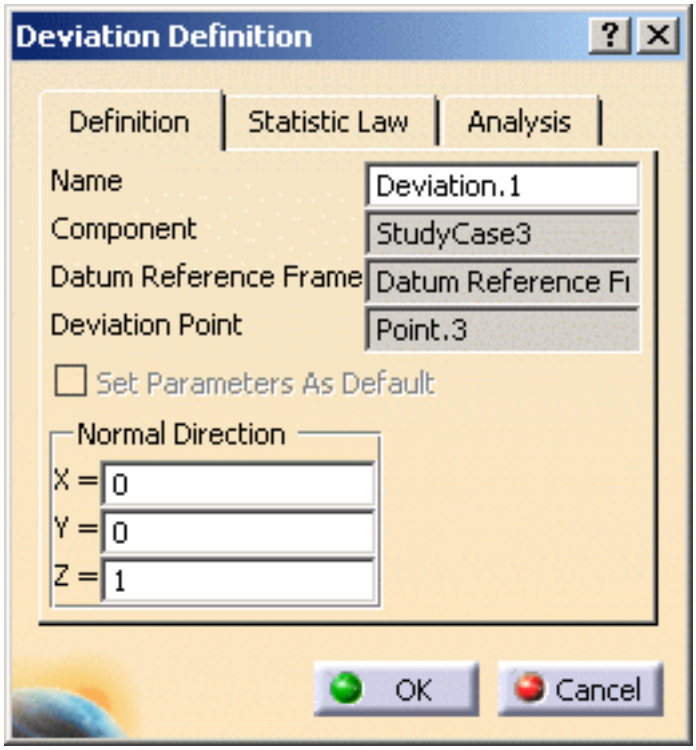


In a CATProduct document, you will need to select the assembly component.

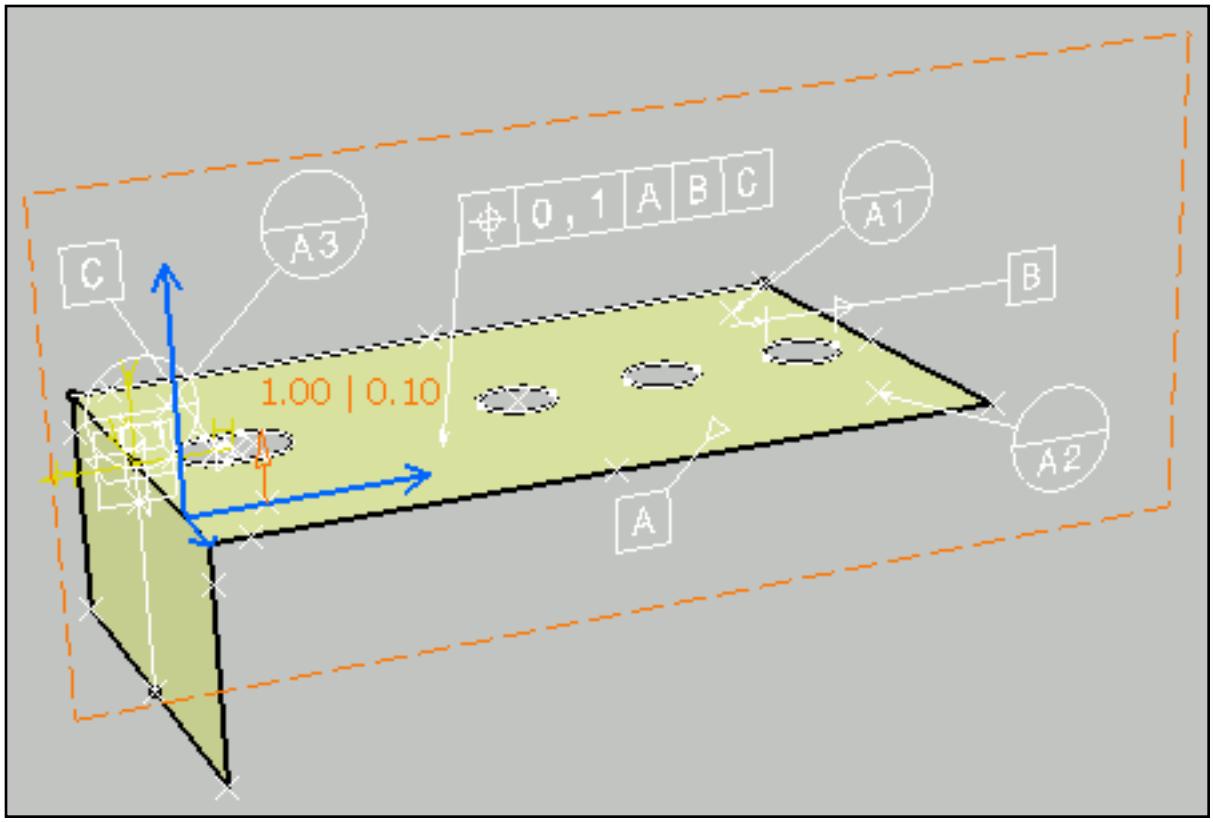
2. Select the deviation's datum reference frame, in this case Datum Reference Frame.3.
3. Select a point or a vertex. For the purpose of this scenario, select Point.3.



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.



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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.

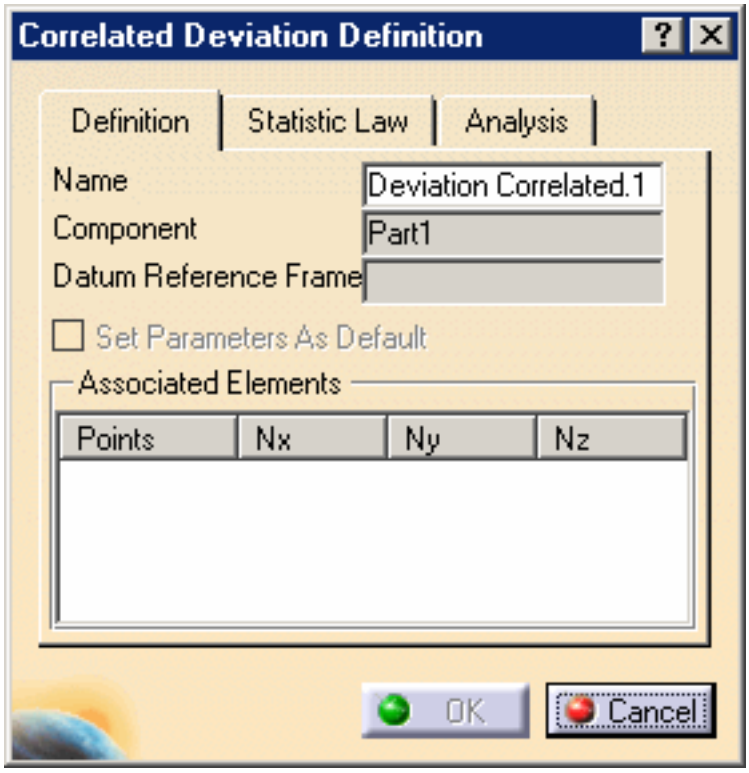
Clicking the **Generate Points** command in the **Deviation Definition** dialog box to generate default annotation points. These points are typical points where the component is the more flexible.

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.

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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.

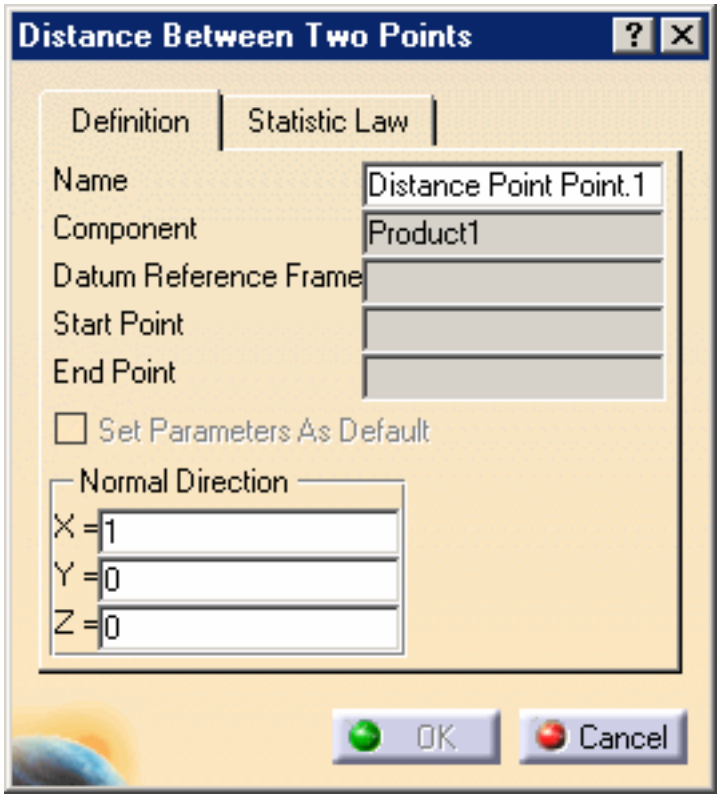
Be associated with the assembly component where the deviation is created.

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.

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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.

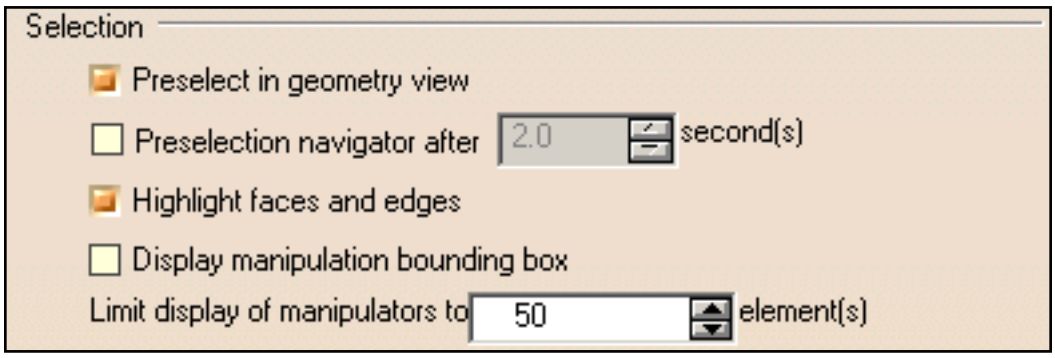
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Selecting Annotation/Annotation Plane

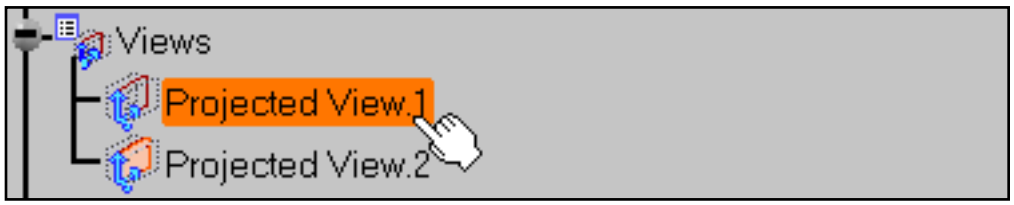
This task shows you how to select the annotations of an annotation plane and the annotation plane of an annotation.

Open the Tolerancing_Annotations_05 CATPart document.

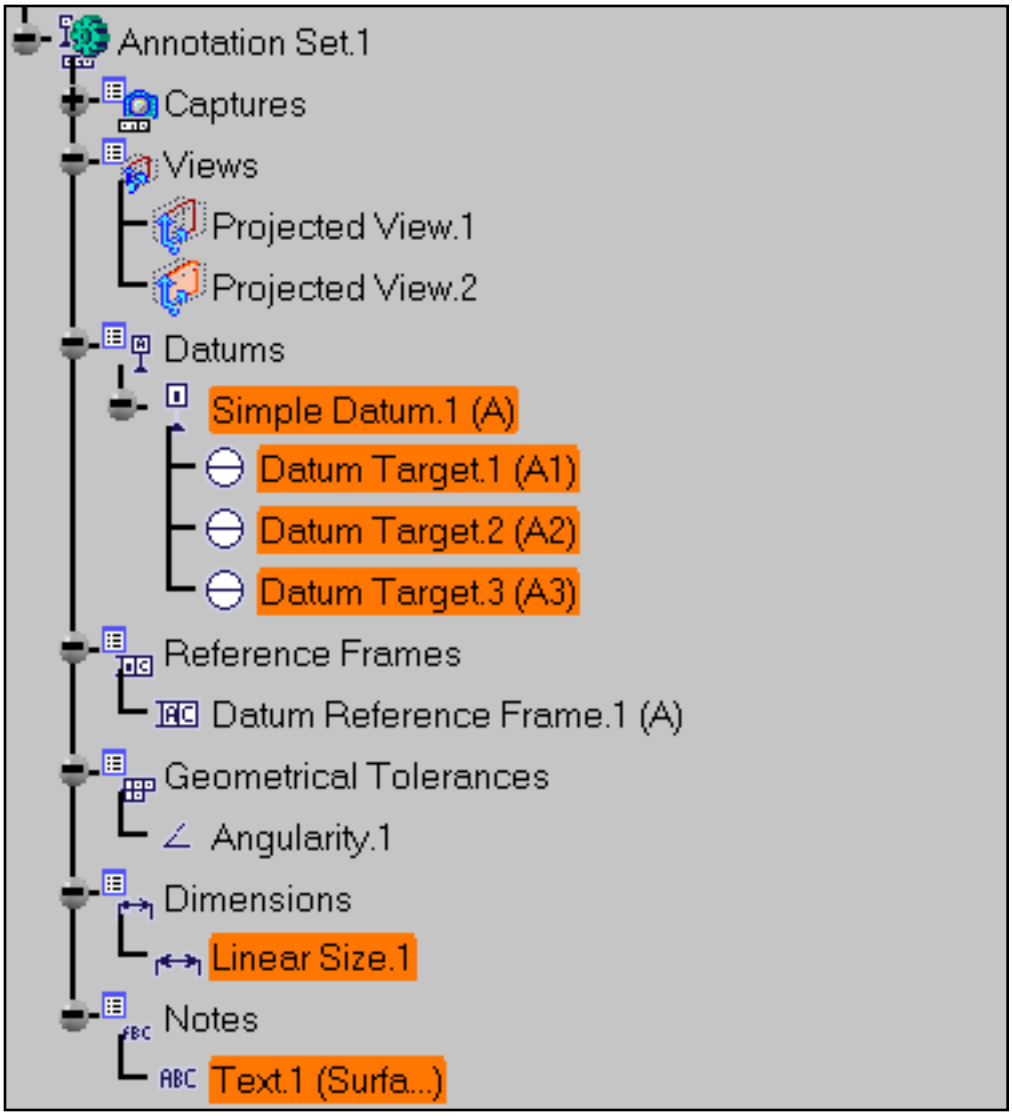
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

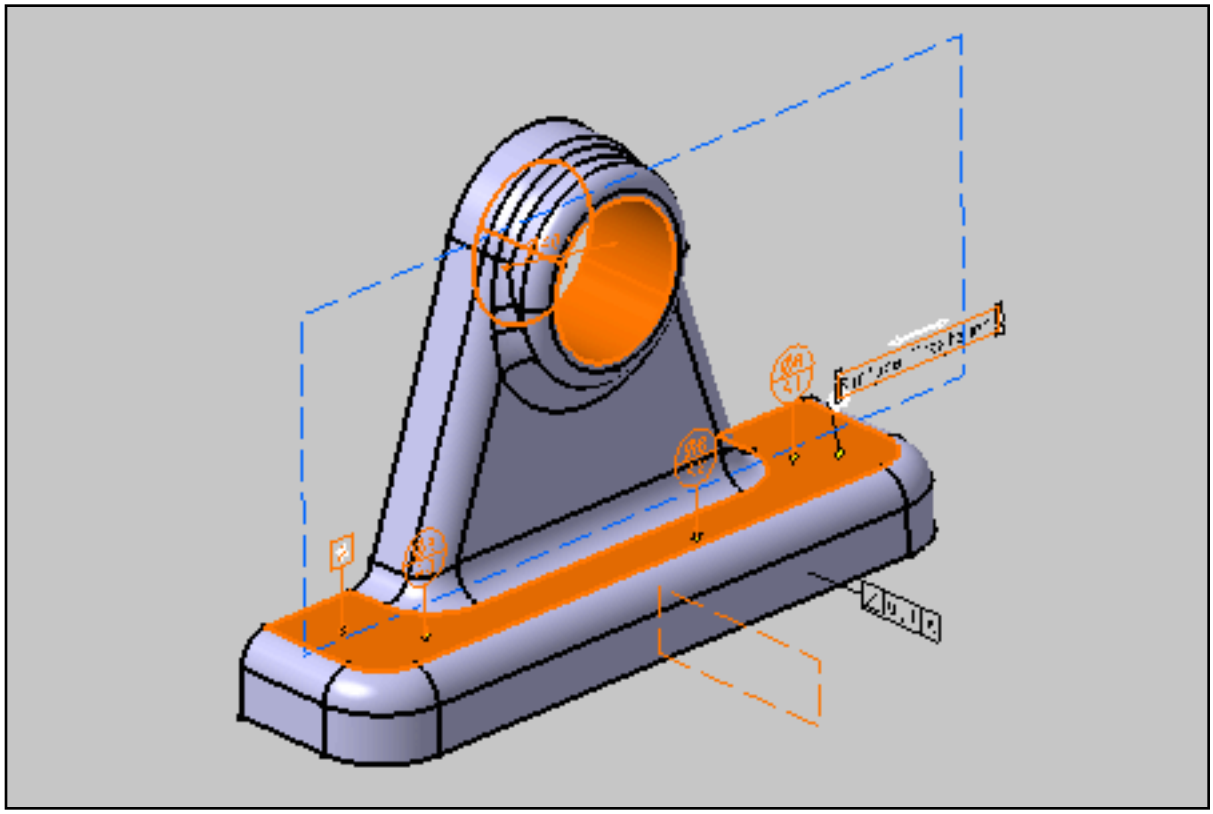


1. Right-click the **Projection View.1** annotation plane and select **Select Annotations** from the contextual menu.

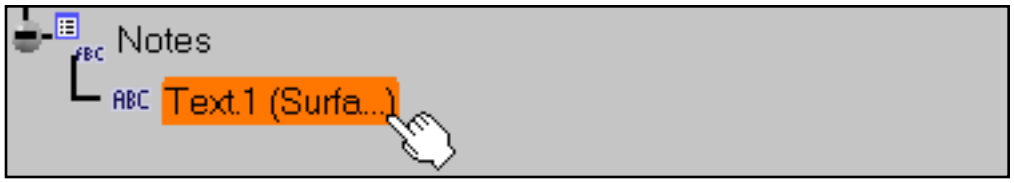


All the annotations of the **Projection View.1** annotation plane are selected in the geometry and the specification tree.
The selected annotation plane is not activated and you can select several annotation planes to perform the command.

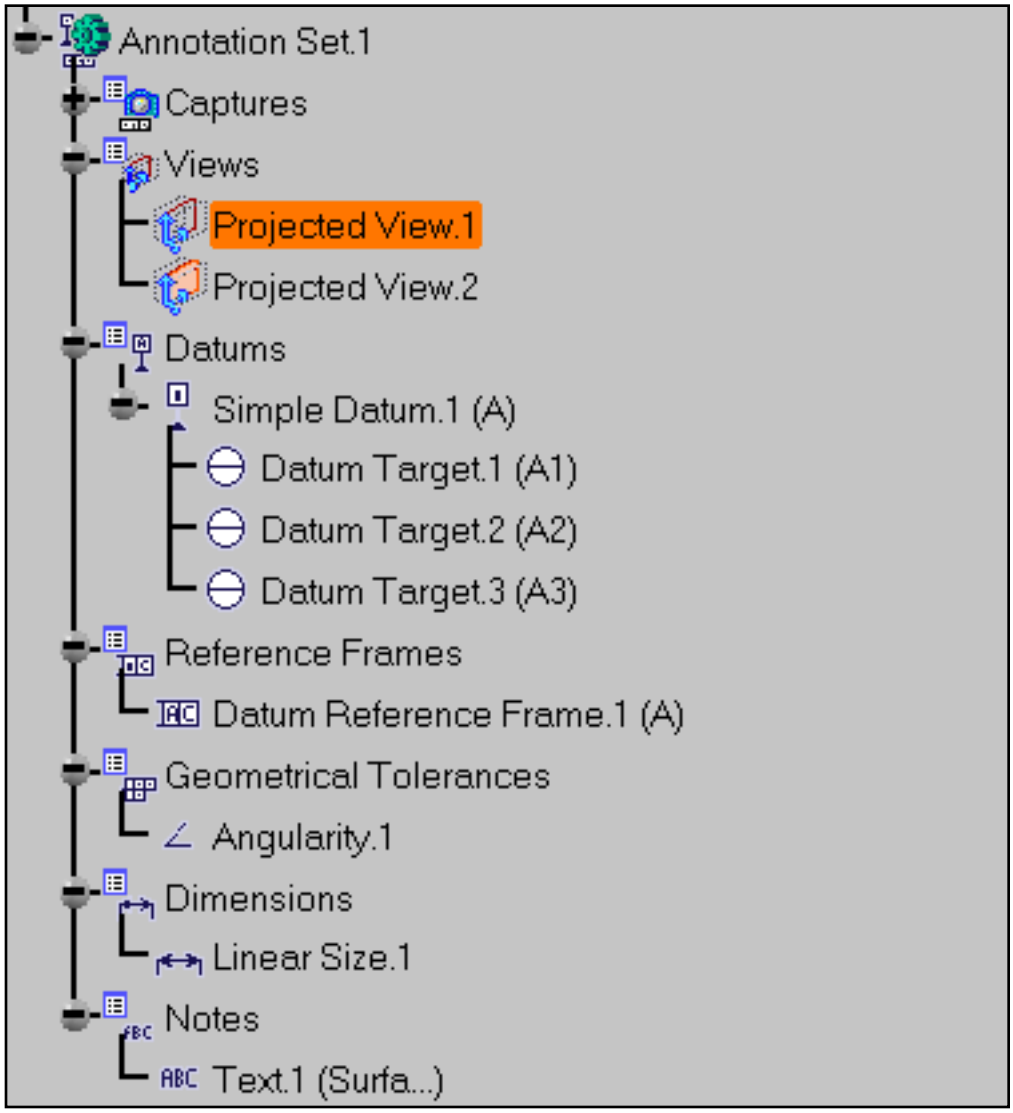

Annotation Set.1
Captures
Views
Projected View.1
Projected View.2
Datums
Simple Datum.1 (A)
Datum Target.1 (A1)
Datum Target.2 (A2)
Datum Target.3 (A3)
Reference Frames
Datum Reference Frame.1 (A)
Geometrical Tolerances
Angularity.1
Dimensions
Linear Size.1
Notes
Text.1 (Surfa...)

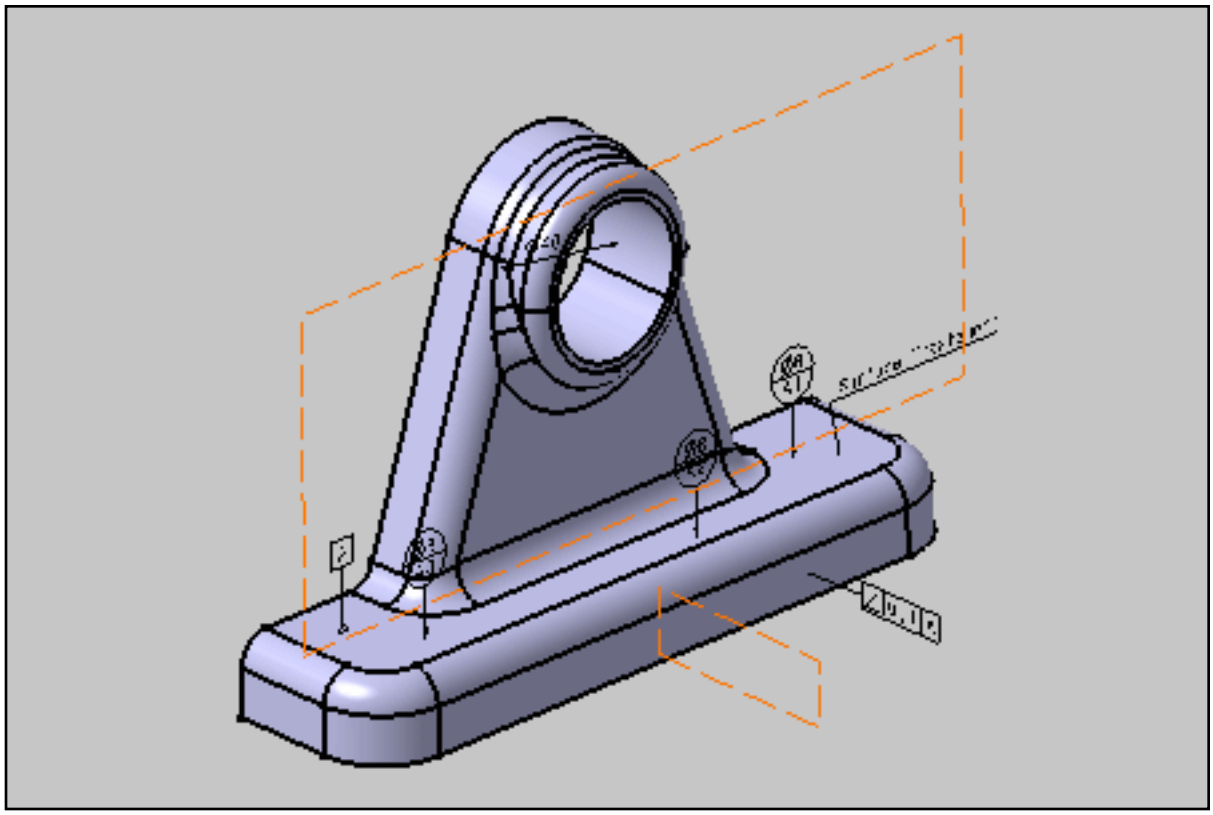


2. Right-click the **Text.1** annotation and select **Select Views/Annotation Plane** from the contextual menu.



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.


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
Text.1 (Surfa...)





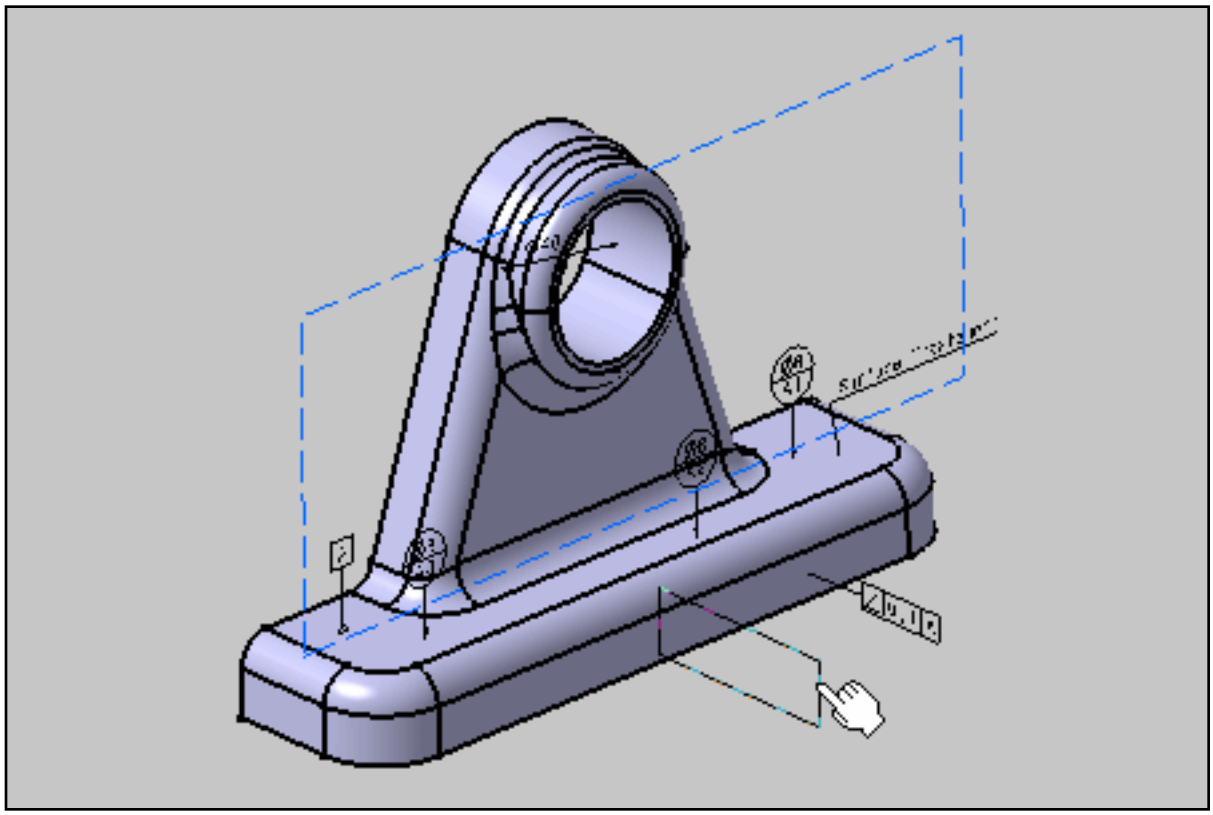
Up Next

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.

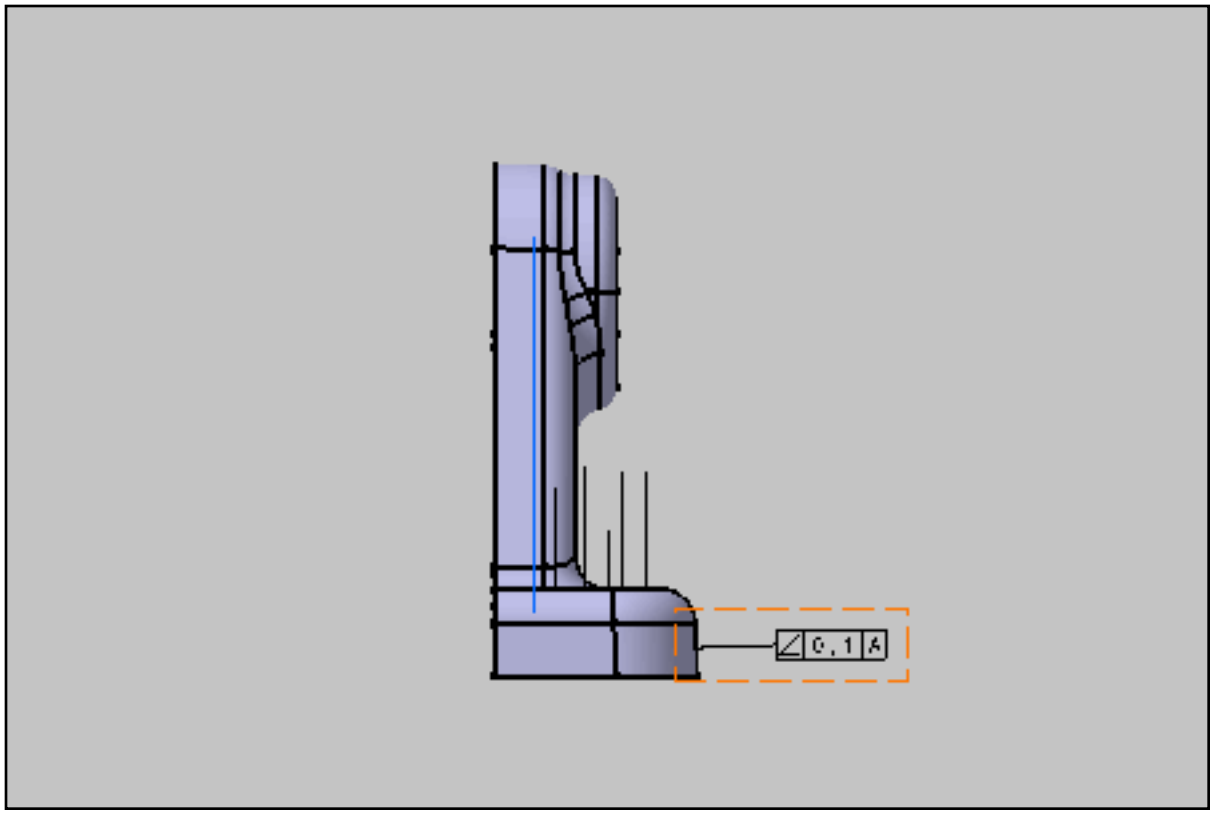
Open the Tolerancing_Annotations_05 CATPart document.

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



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

The annotation plane is displayed in the normal view.



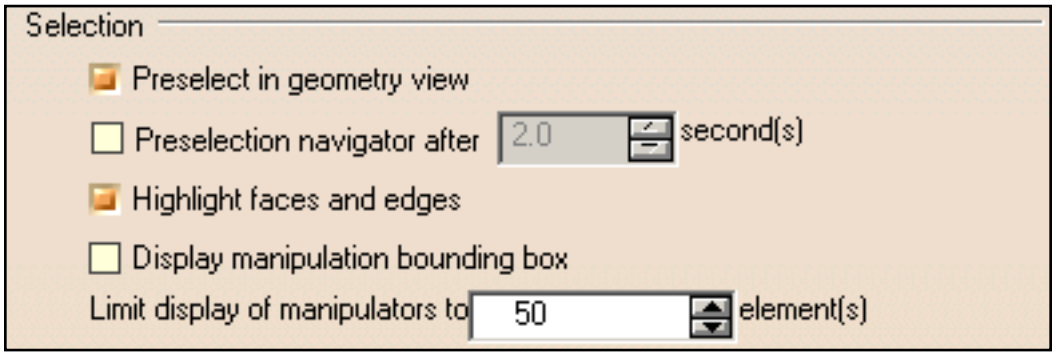
Back Up Next

Moving Annotations

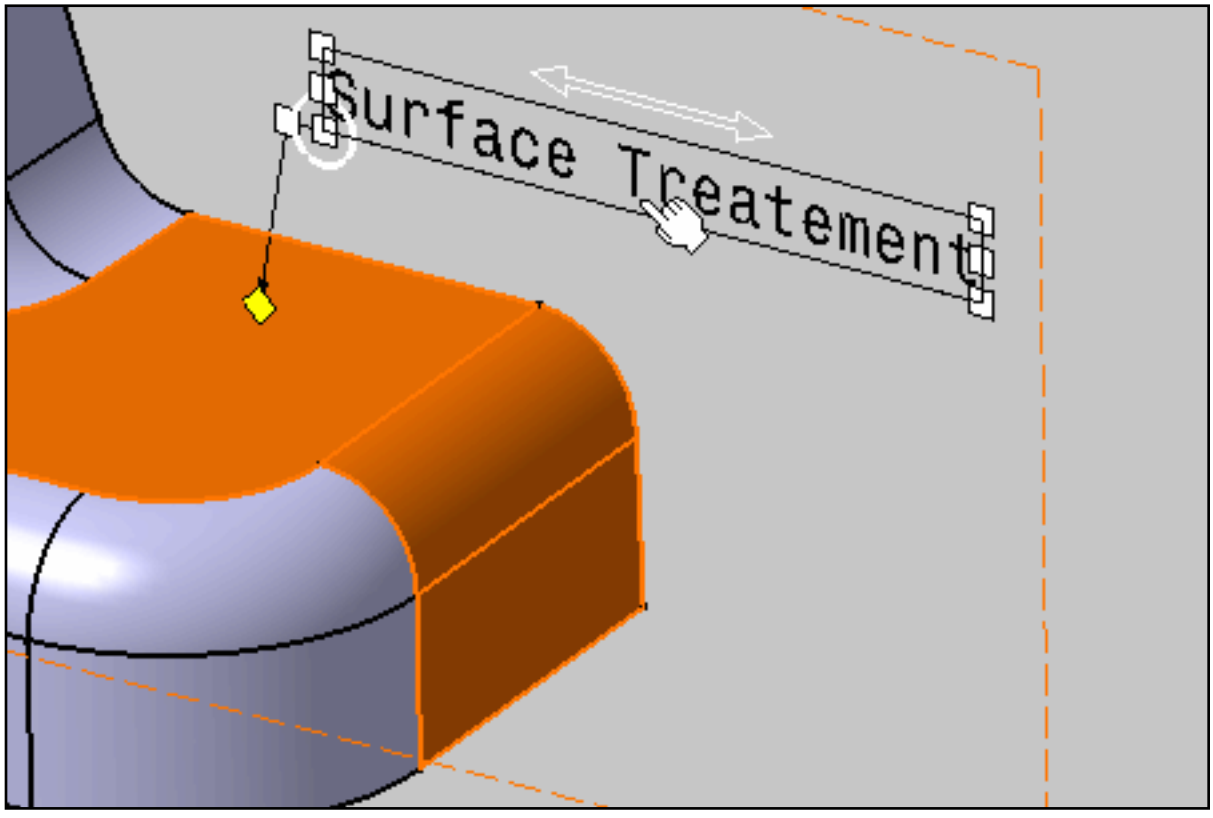
This task shows you two ways of moving a text annotation: by drag and drop, then by using coordinates. The operating mode described here applies to datum elements, datum targets and geometrical tolerances.

Open the Tolerancing_Annotations_04 CATPart document.

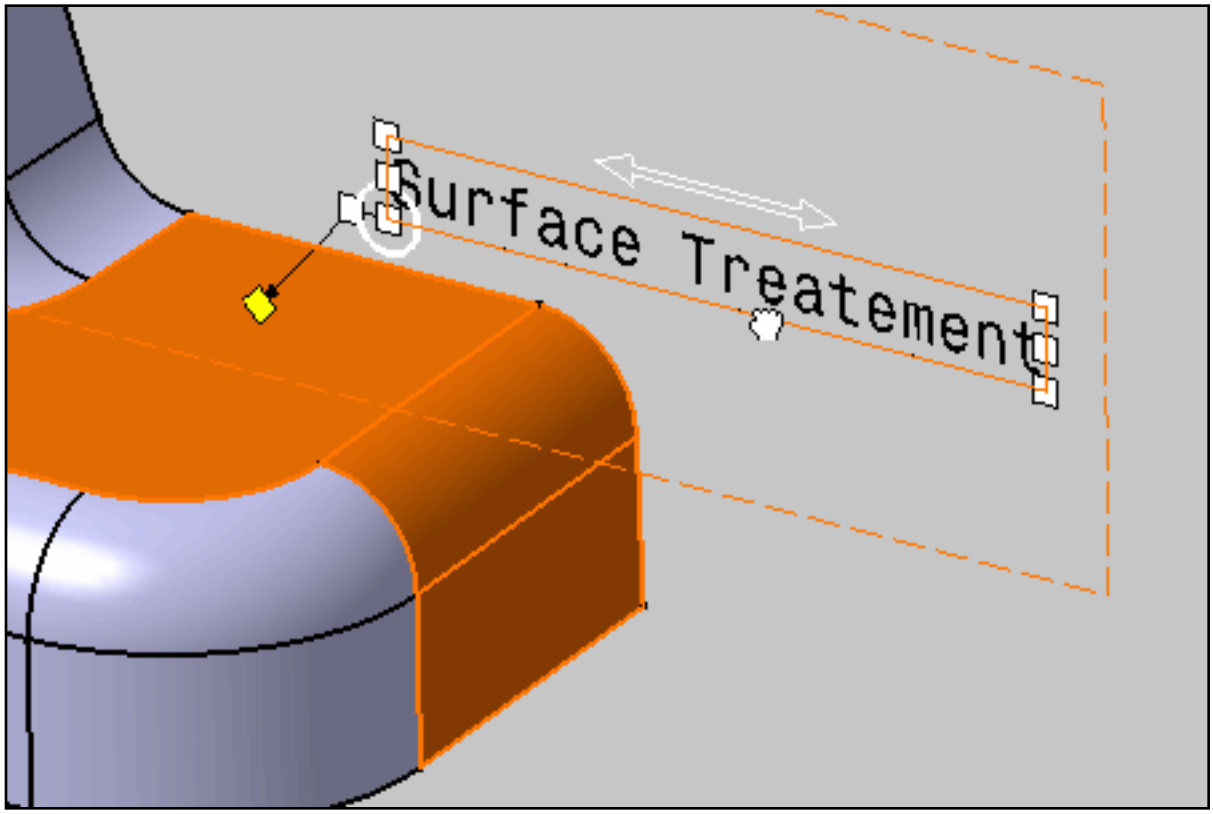
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



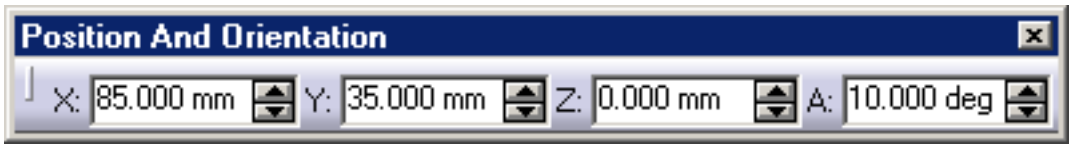
1. Select the annotation text.



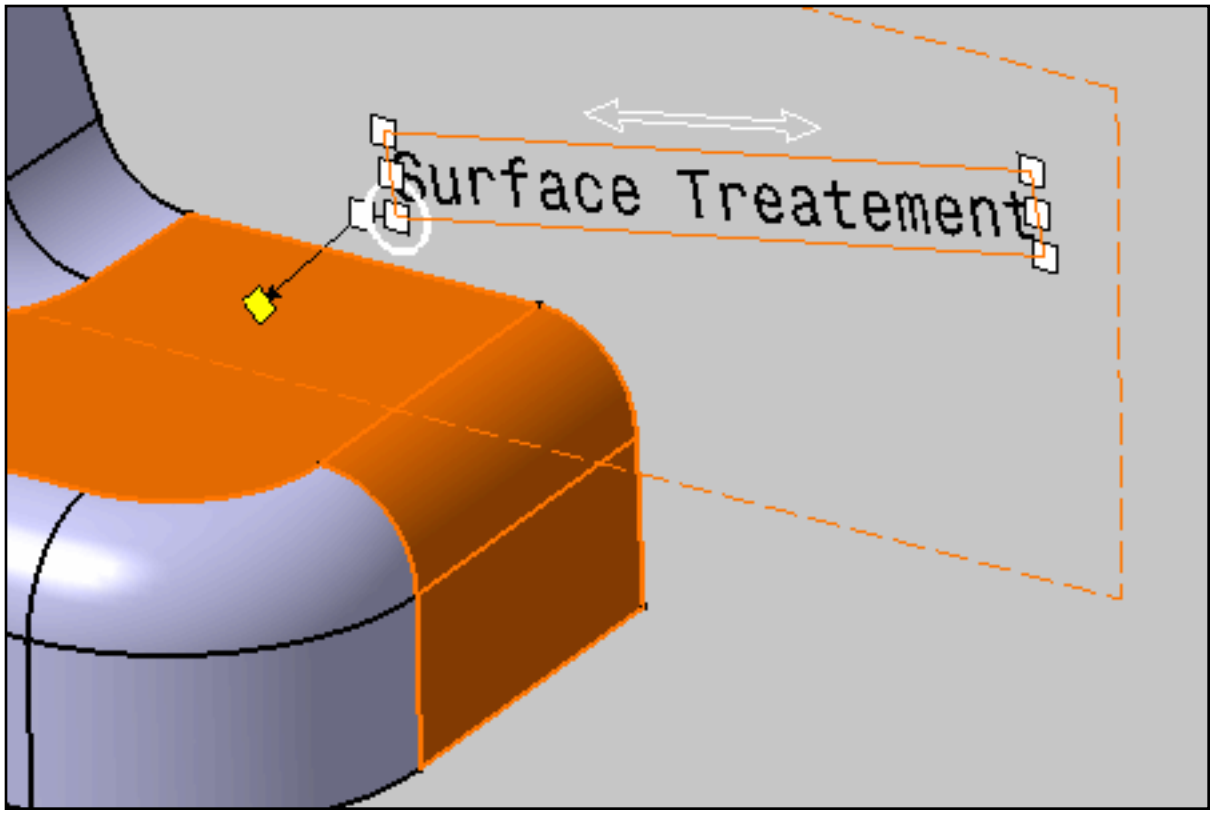
2. Drag it to the desired location.
   Note that you can stretch or reduce the frame too.



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.



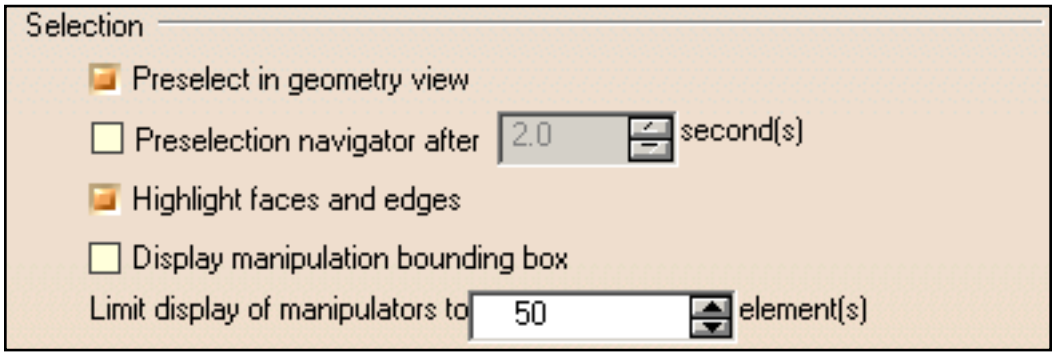
Back Up Next

Transferring Existing Annotations

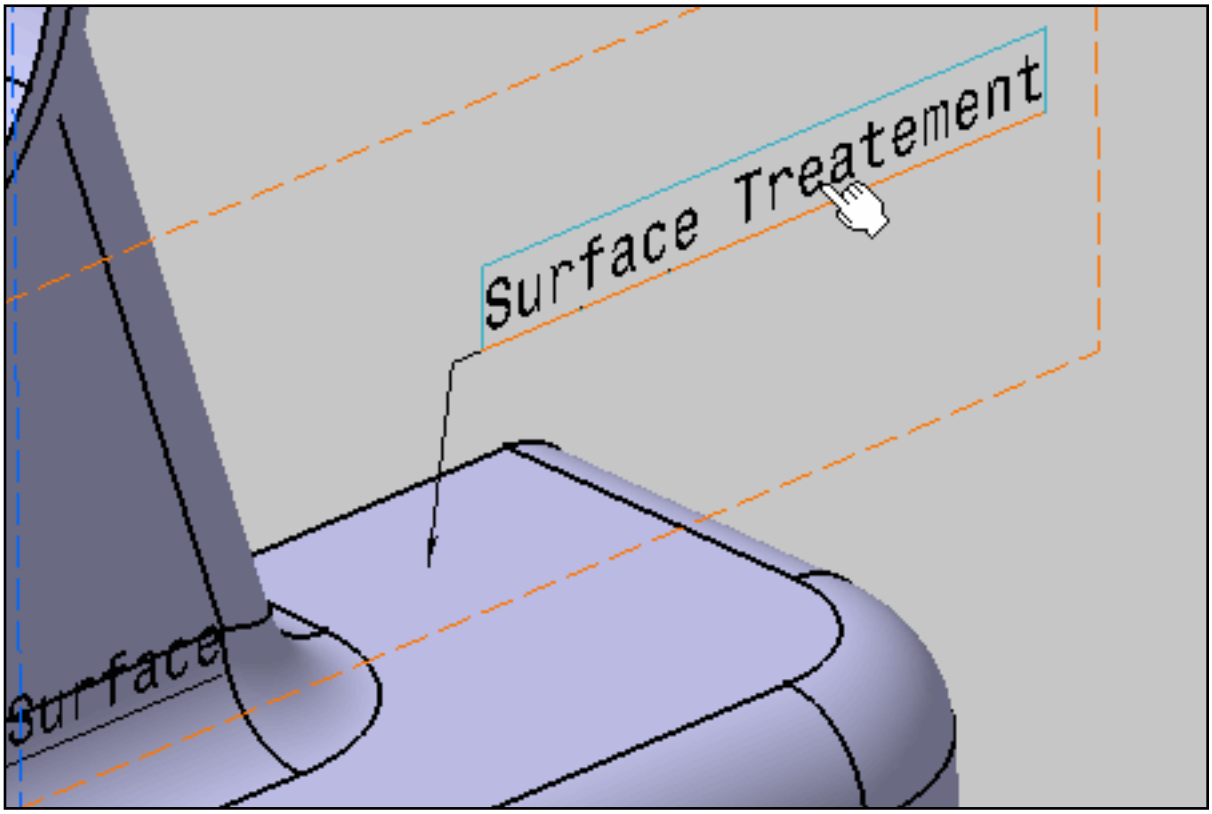
This task first shows you how to transfer an existing textual annotation from one view to another.
The operating mode described here applies to datum elements, datum targets and geometrical tolerances too.

Open the Tolerancing_Annotations_04 CATPart document.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

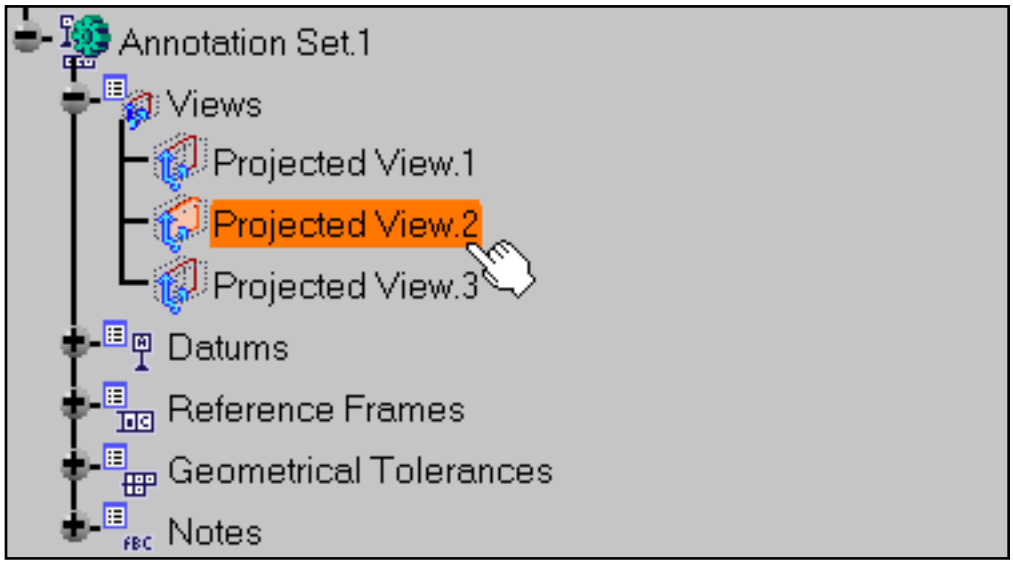


1. Select the text (or its leader line).

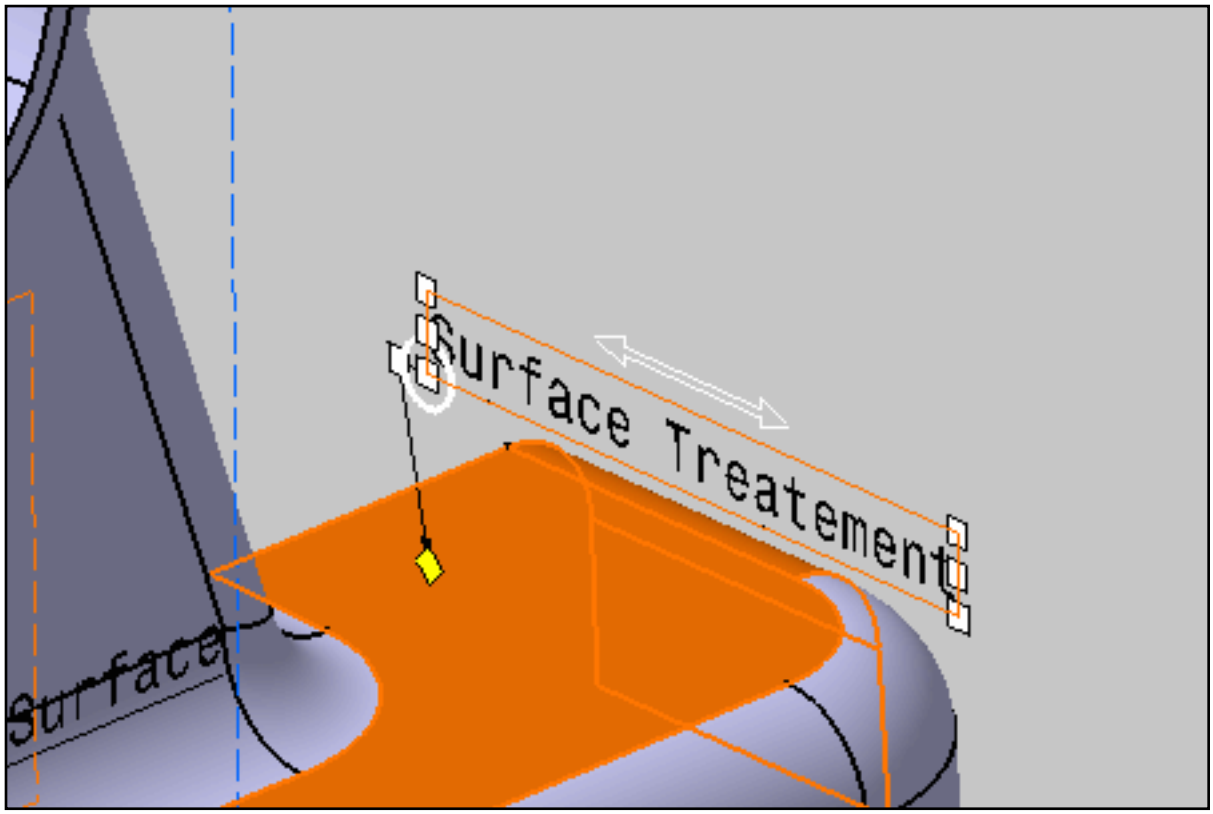


2. Right-click to select the **Transfer to View / Annotation Plane** contextual menu.

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.





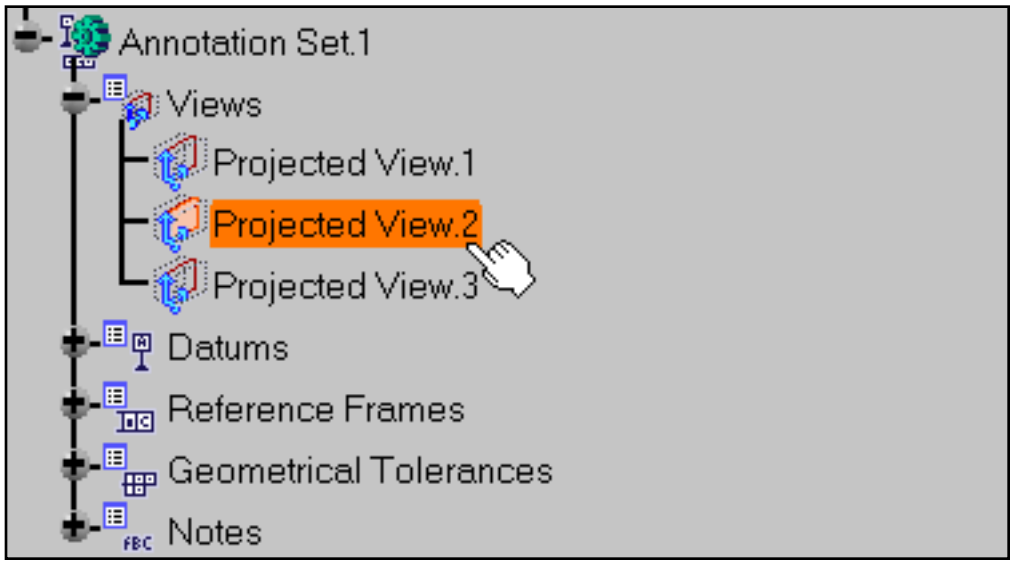
Back Up Next

Transferring Annotations During Creation

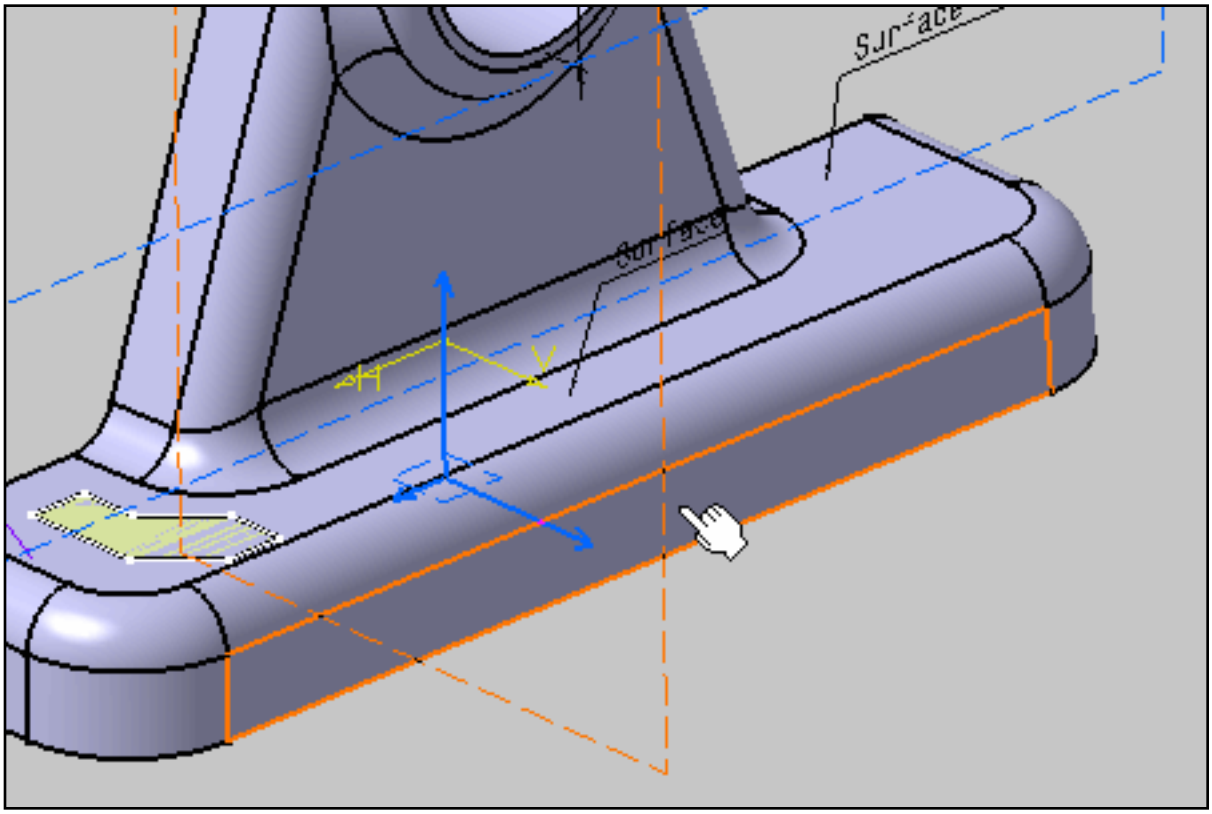
This task first shows you how to transfer the annotation you are creating to another view.
The operating mode described here applies to datum elements, datum targets and geometrical tolerances too.

Open the Tolerancing_Annotations_04 CATPart document.

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



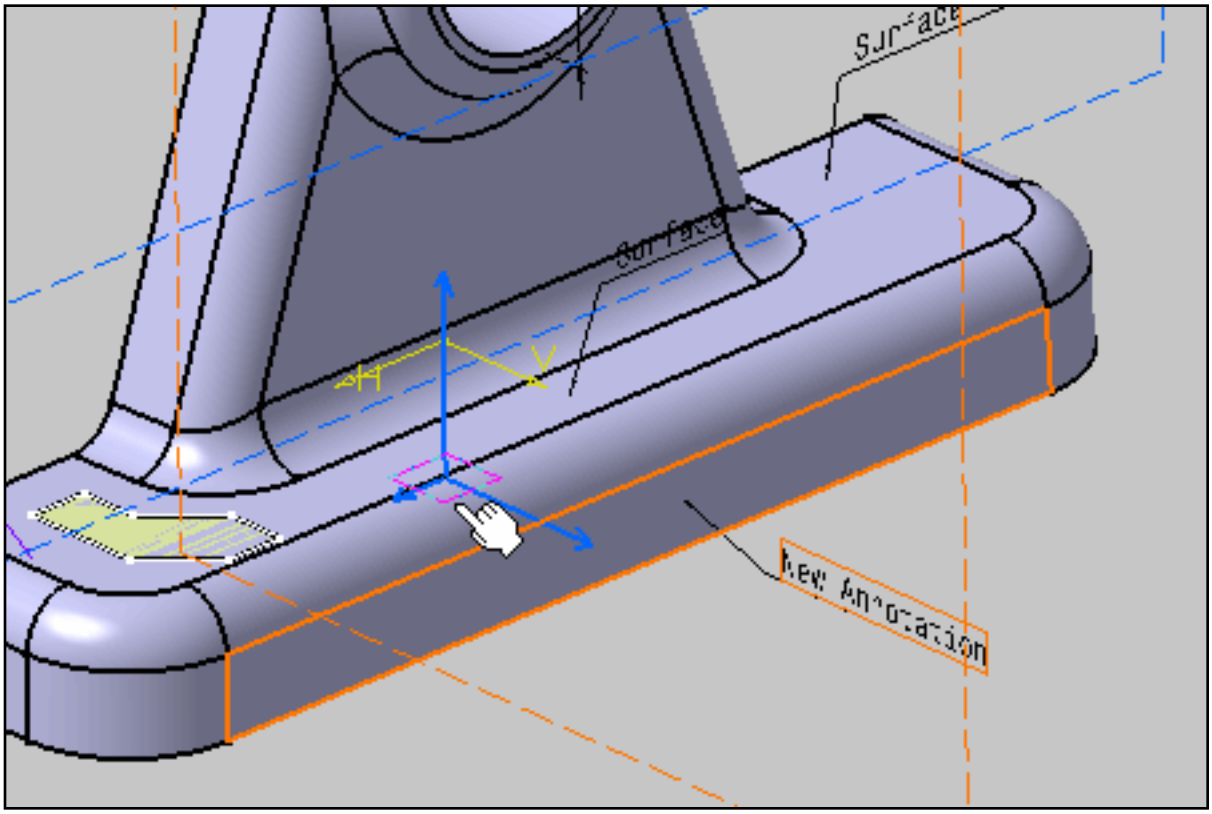
2. Select the face as shown to define a location for the arrow end of the leader.



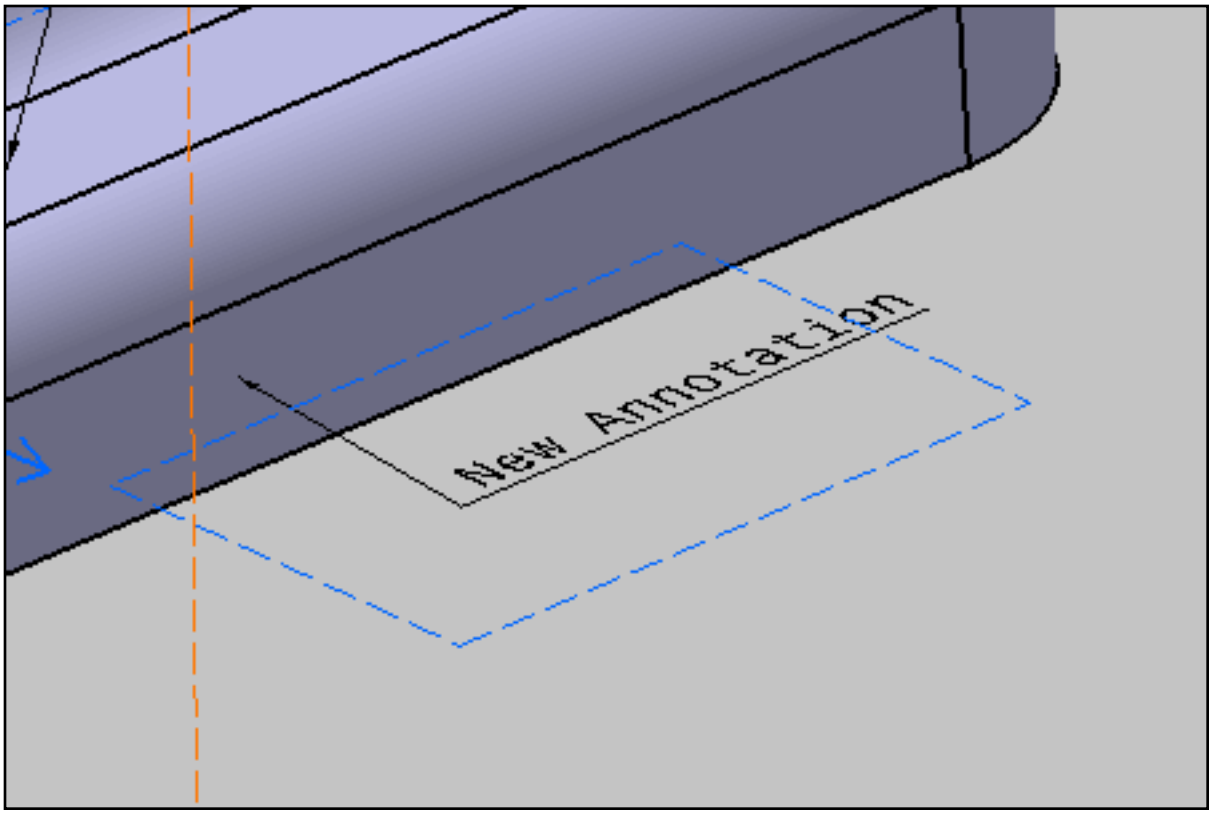
3. Click the **Text with Leader** icon: 

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.



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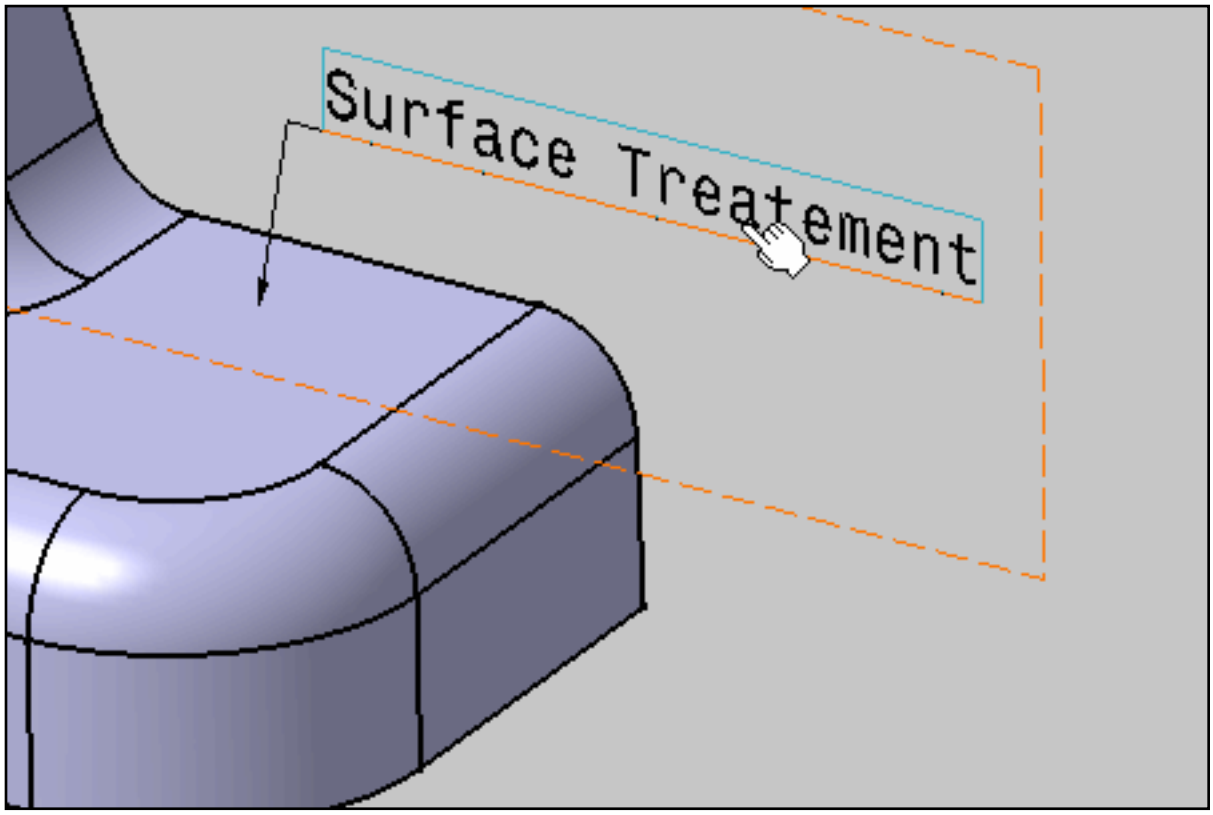
Grouping Annotations During Creation

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.

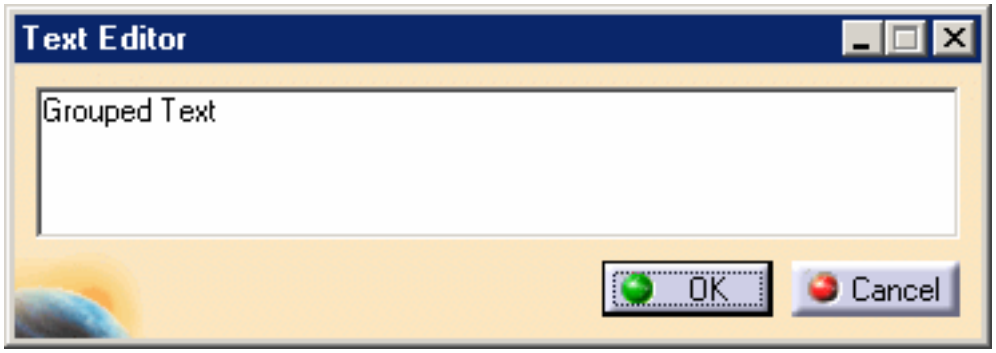
Open the Tolerancing_Annotations_04 CATPart document.

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

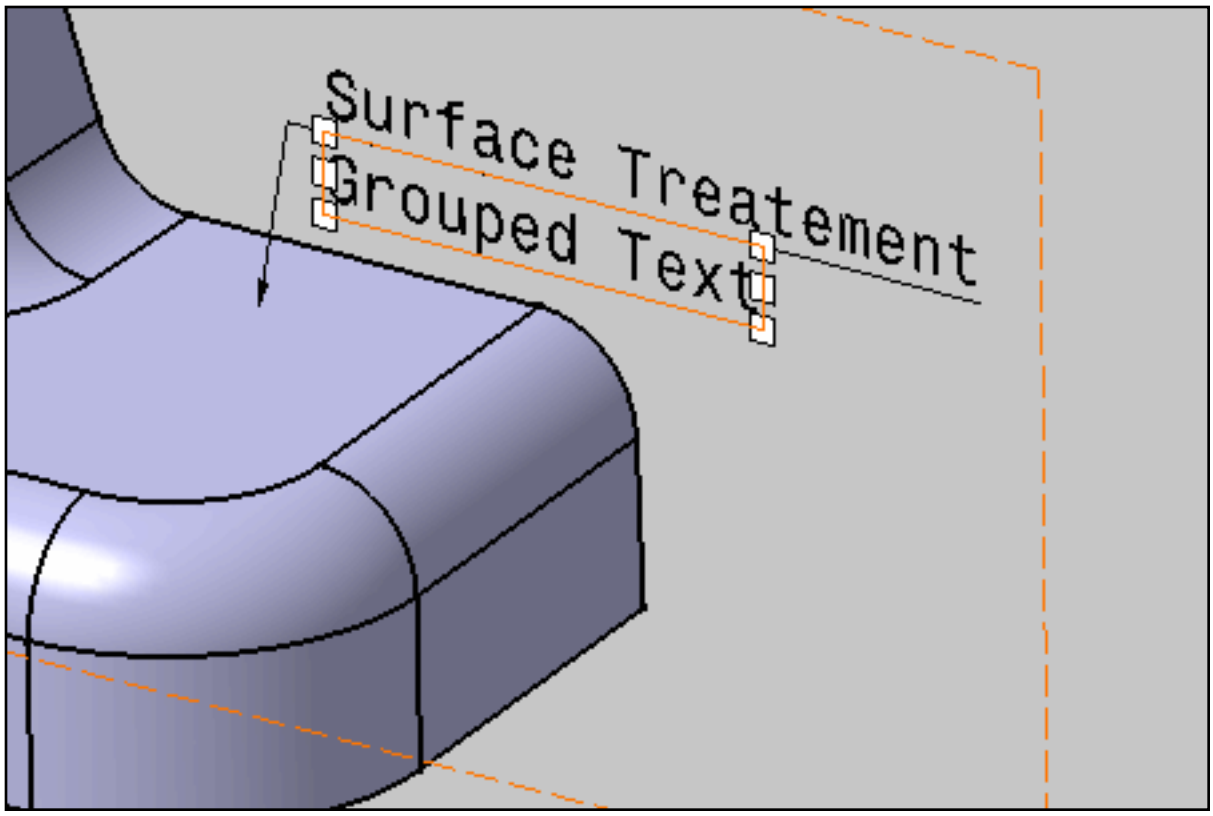
2. Select "Surface Treatment" as the annotation to which you want to attach the annotation you are going to specify.



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.

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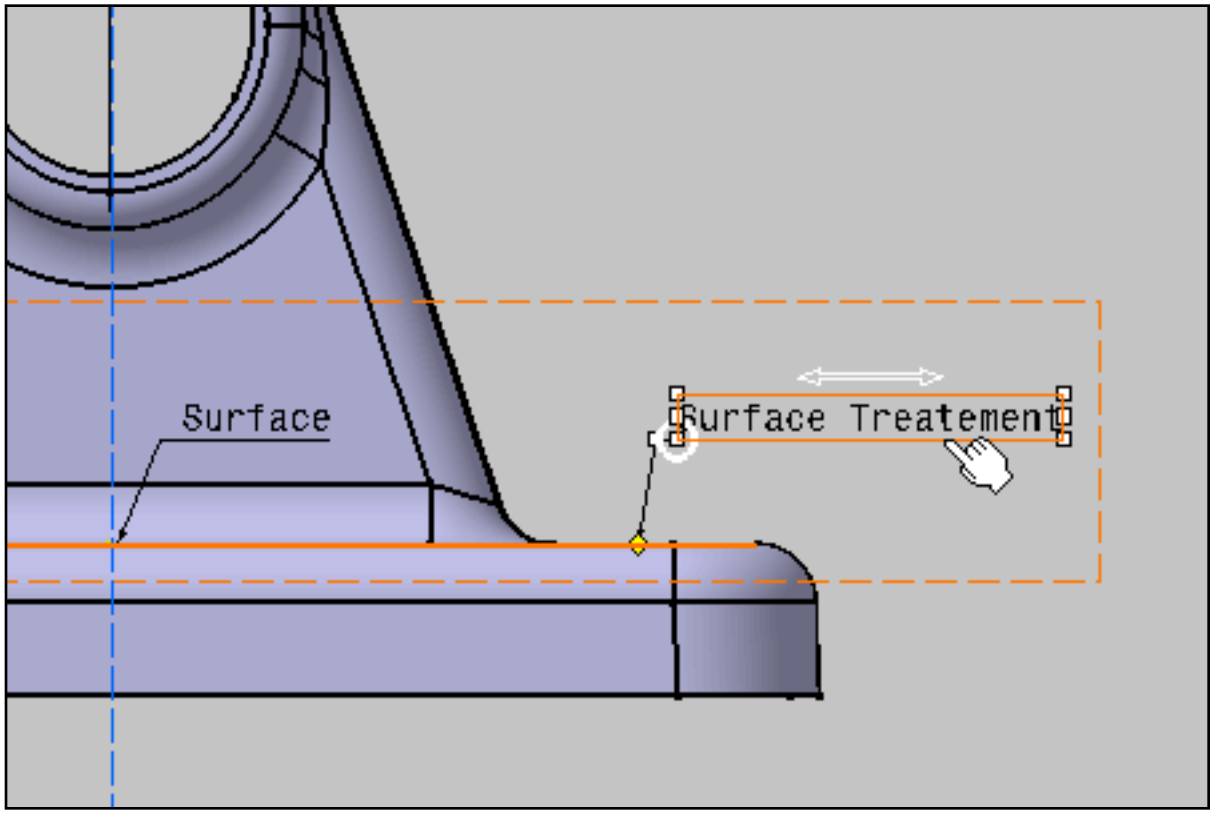
Grouping Annotations Automatically

This task shows you how to group automatically annotations.

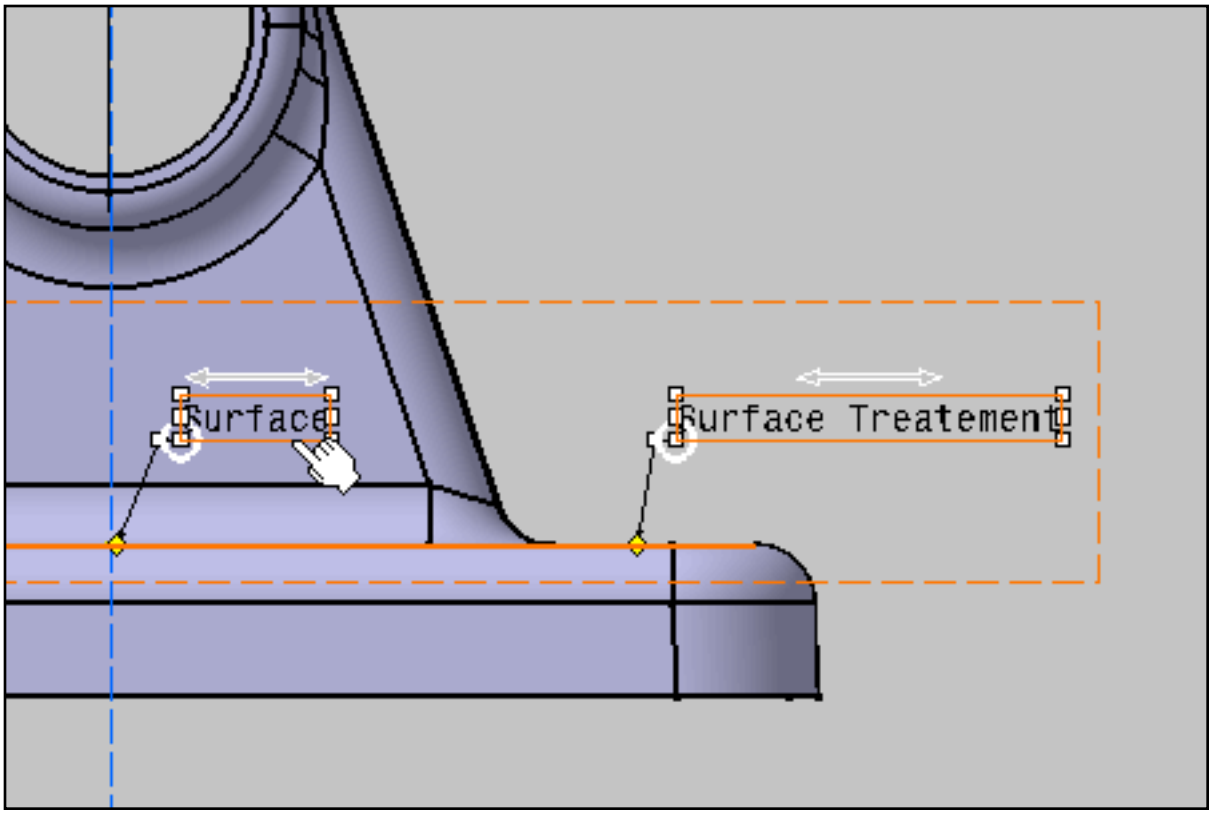
You can select as many annotations as you want.
Selected annotations must associated with the same geometrical elements.
According to the standard annotations are grouped with an specific order.

Open the Tolerancing_Annotations_04 CATPart document.

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

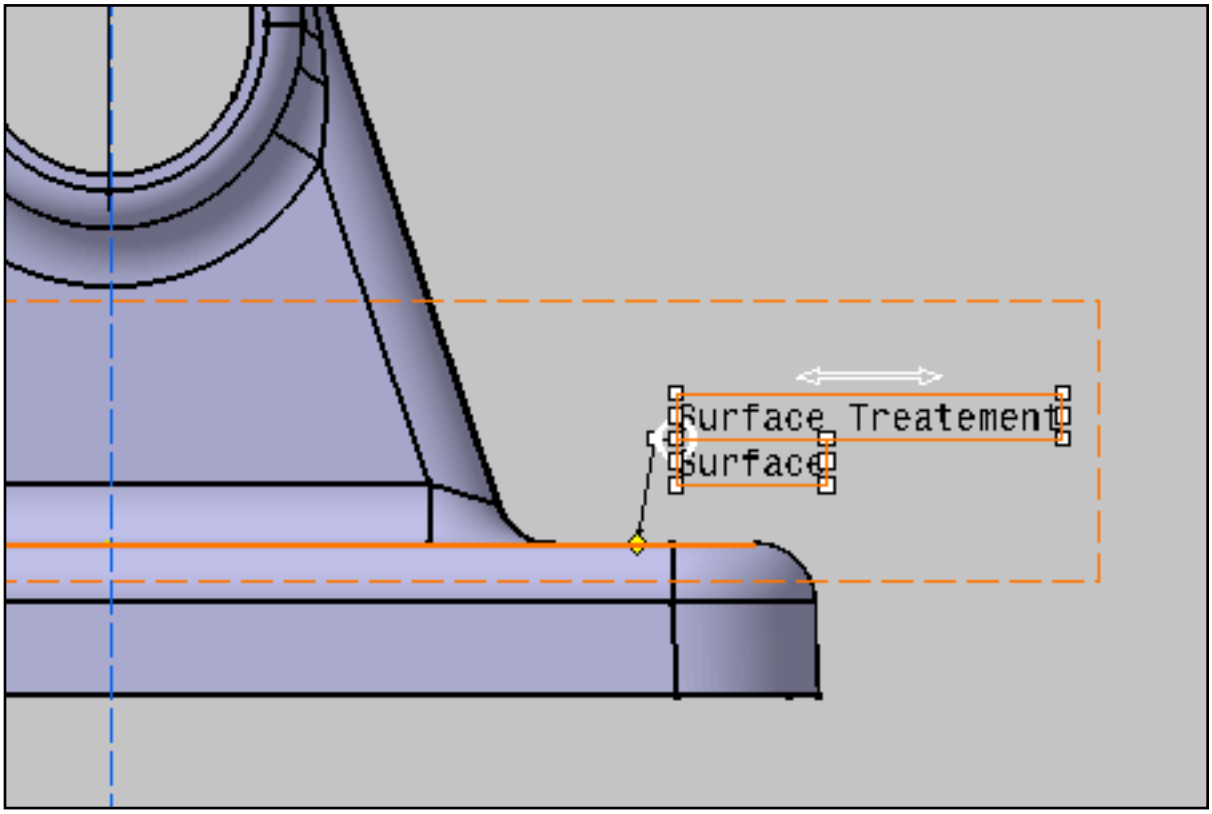


2. Multi-select **Surface** as the second annotation to be grouped.



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

Annotations are grouped.





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Grouping and Ordering Annotations

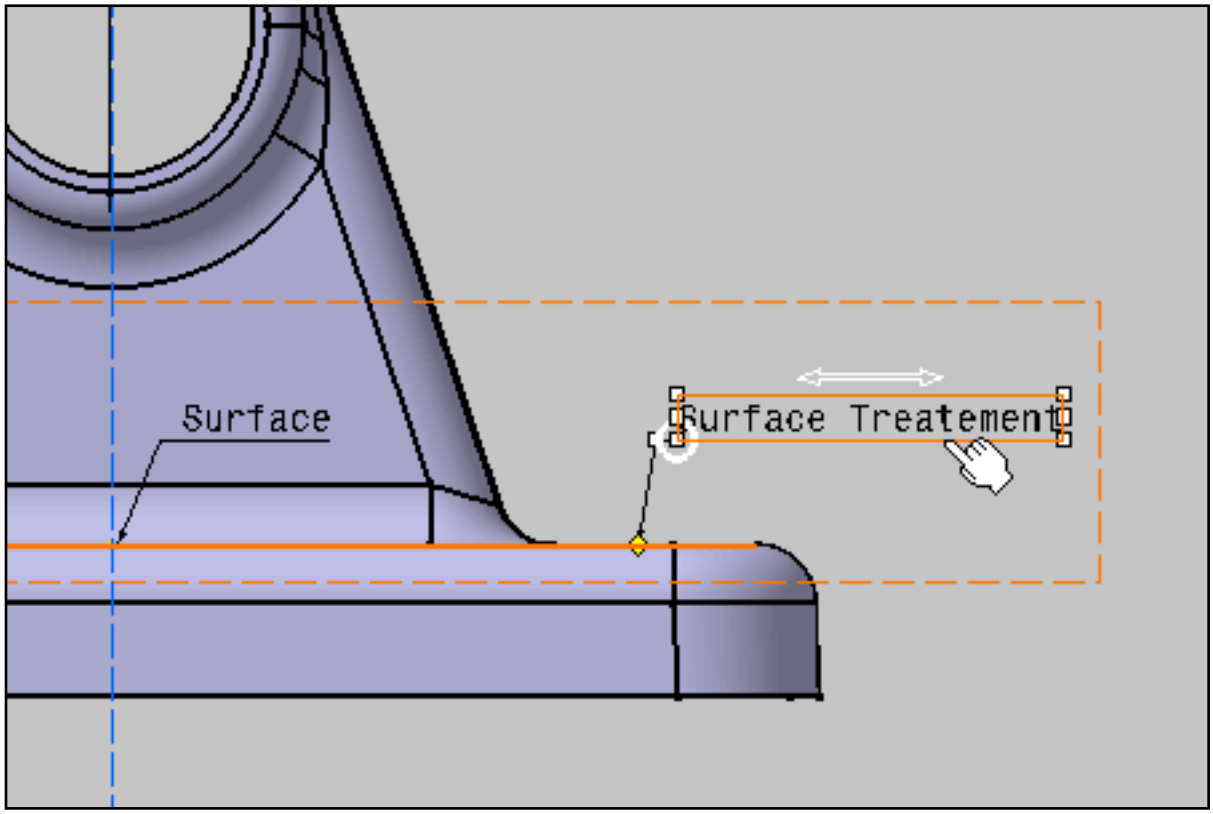
This task shows you how to group manually and order annotations.

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.

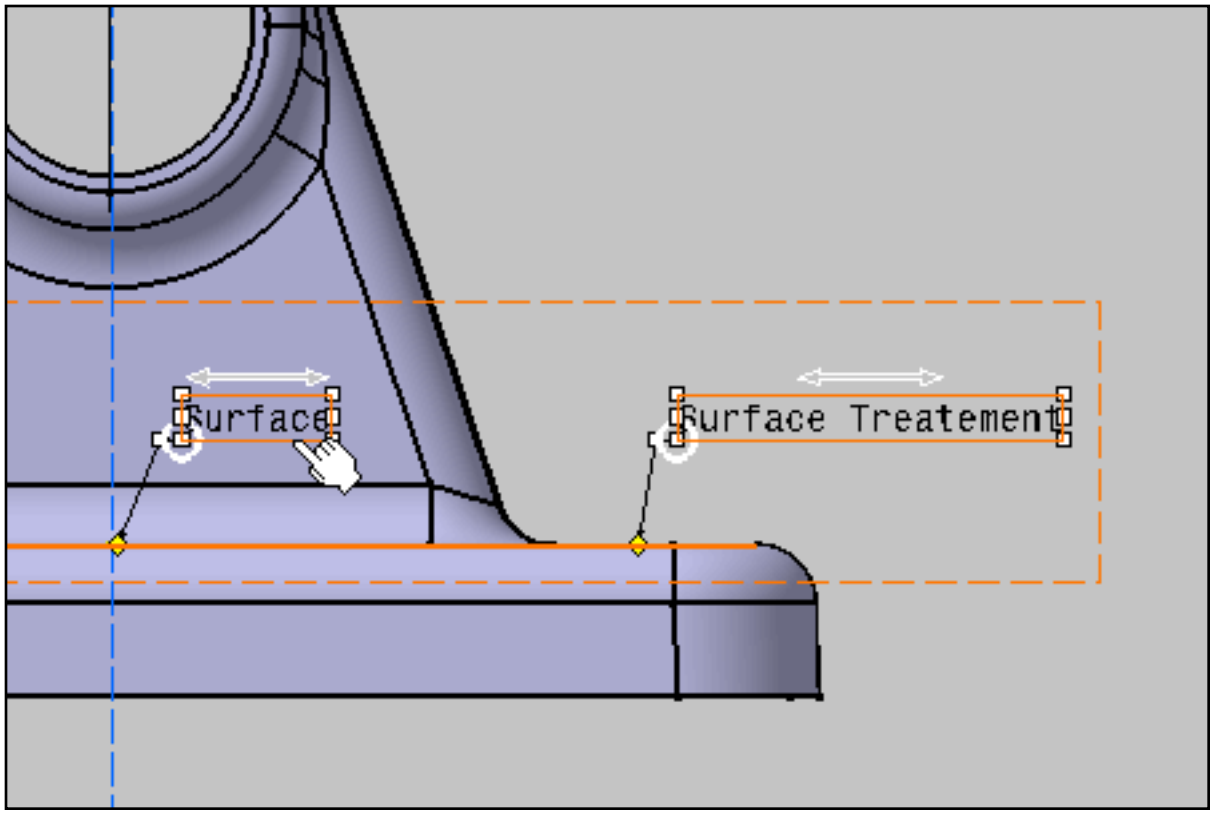
Open the Tolerancing_Annotations_04 CATPart document.

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

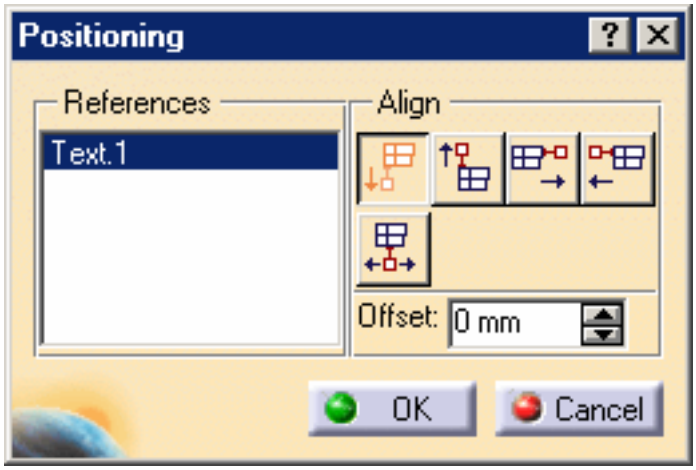
2. Select **Surface Treatment** (**Text.1**) as the reference annotation.



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.

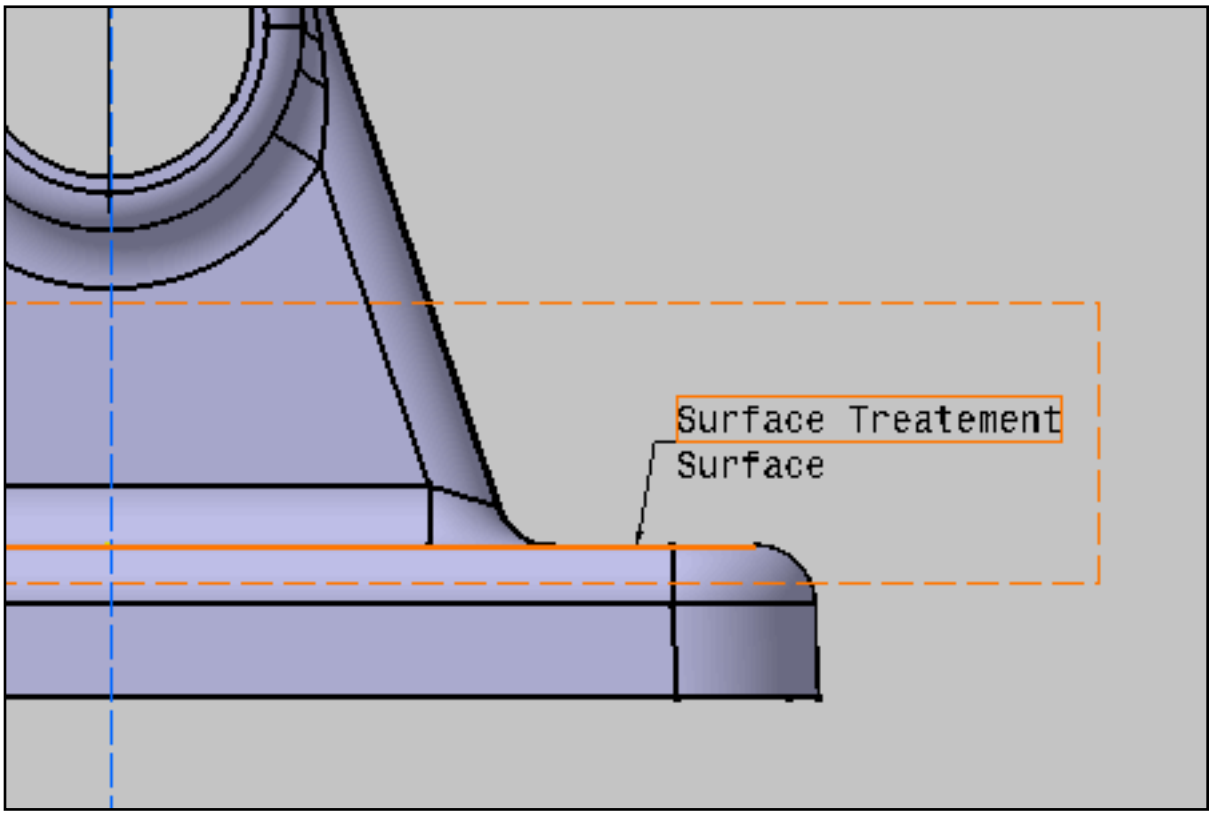
Align Right the selected annotation according to the reference.

Align Left the selected annotation according to the reference.

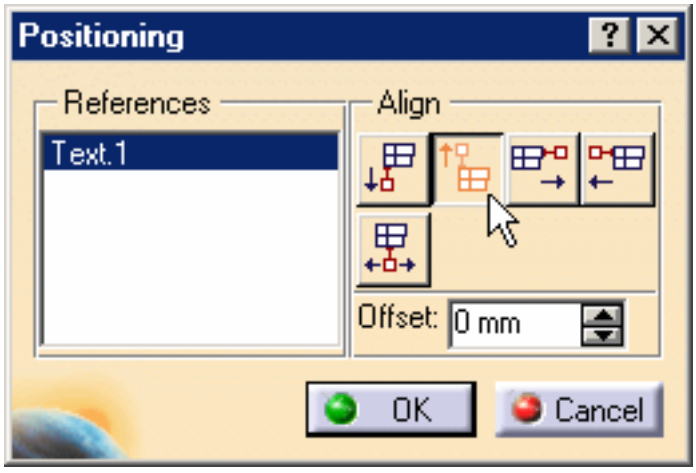
Center Horizontally the selected annotation according to the reference.

Set the **Offset** between the selected annotation and the reference.

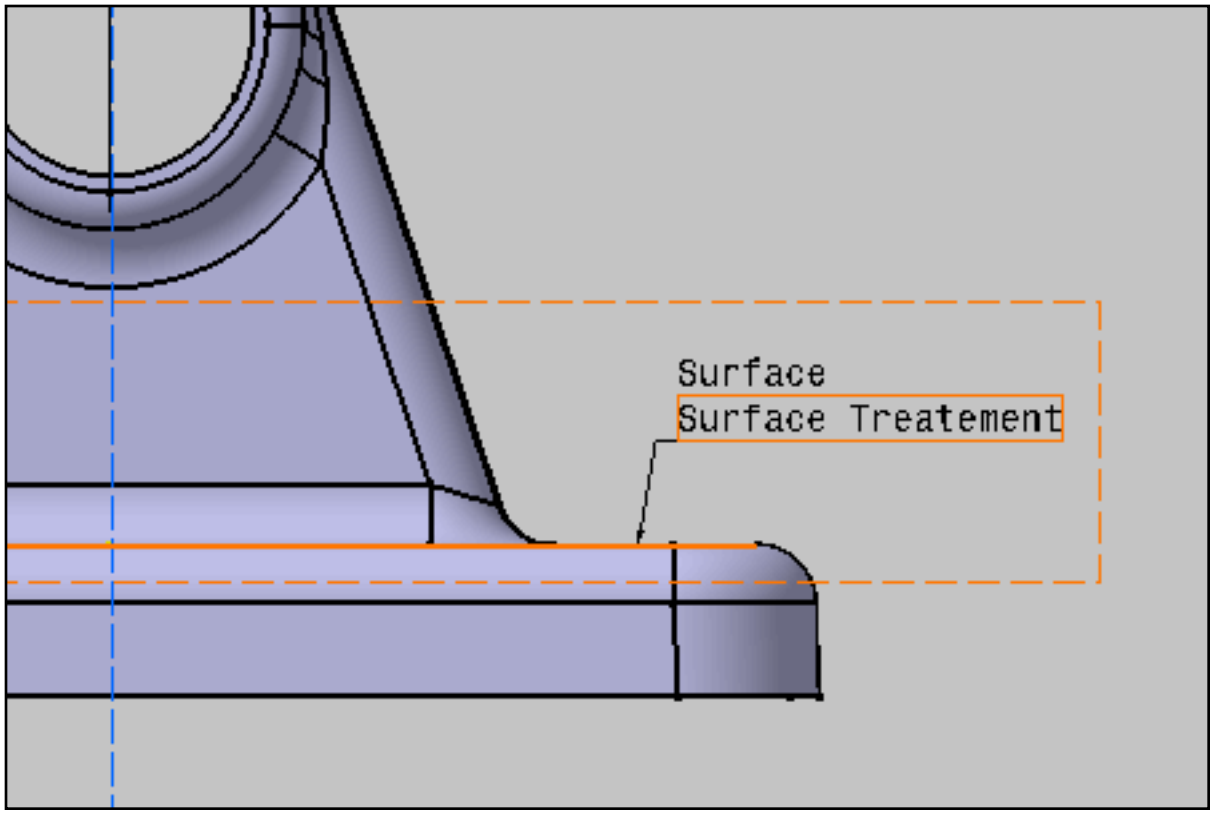
Annotations are grouped according to the dialog box settings: **Align Bottom** and 0mm **O**.
Text.1 reference annotation is orange framed.



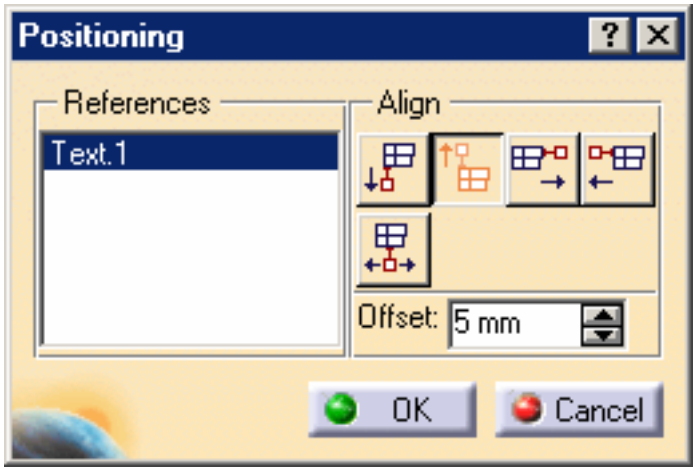
4. Click the **Align Top** icon in the **Positioning** dialog box.



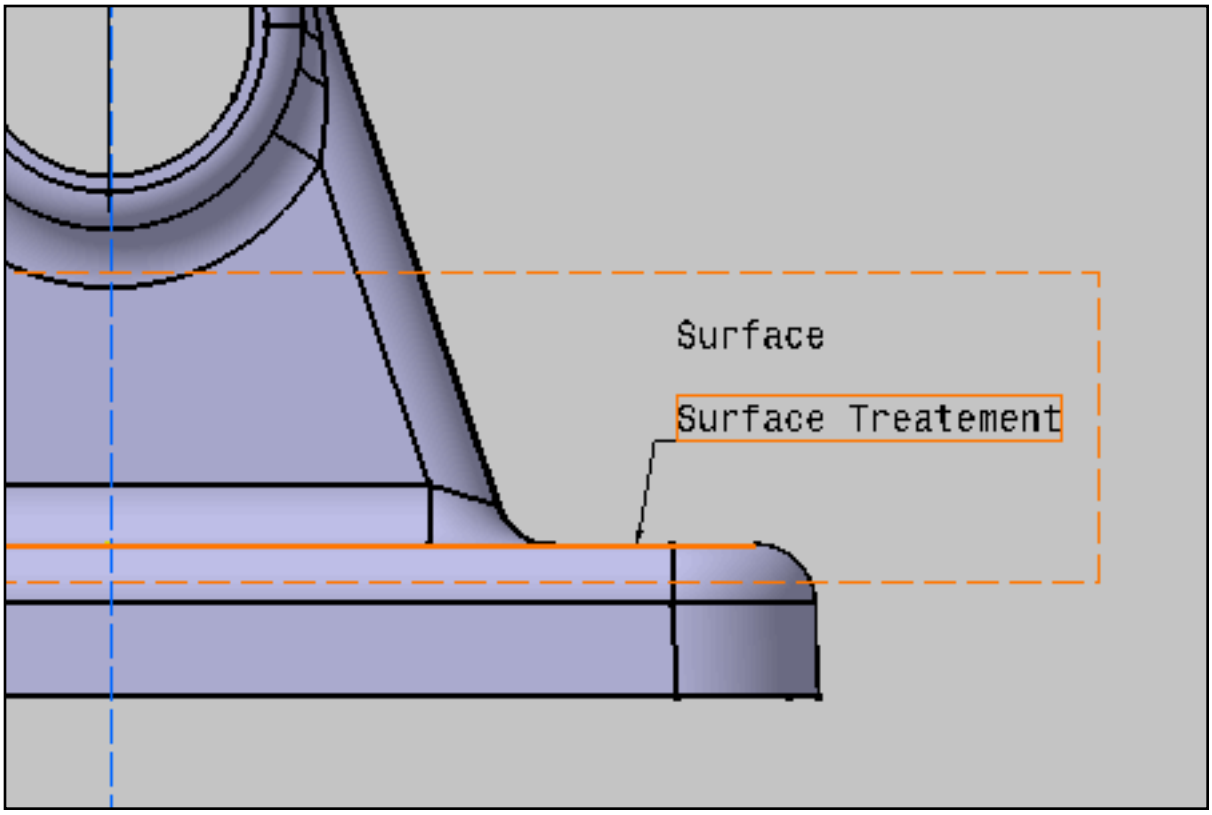
Surface (**Text.2**) is aligned to the top of **Surface Treatment** (**Text.1**).



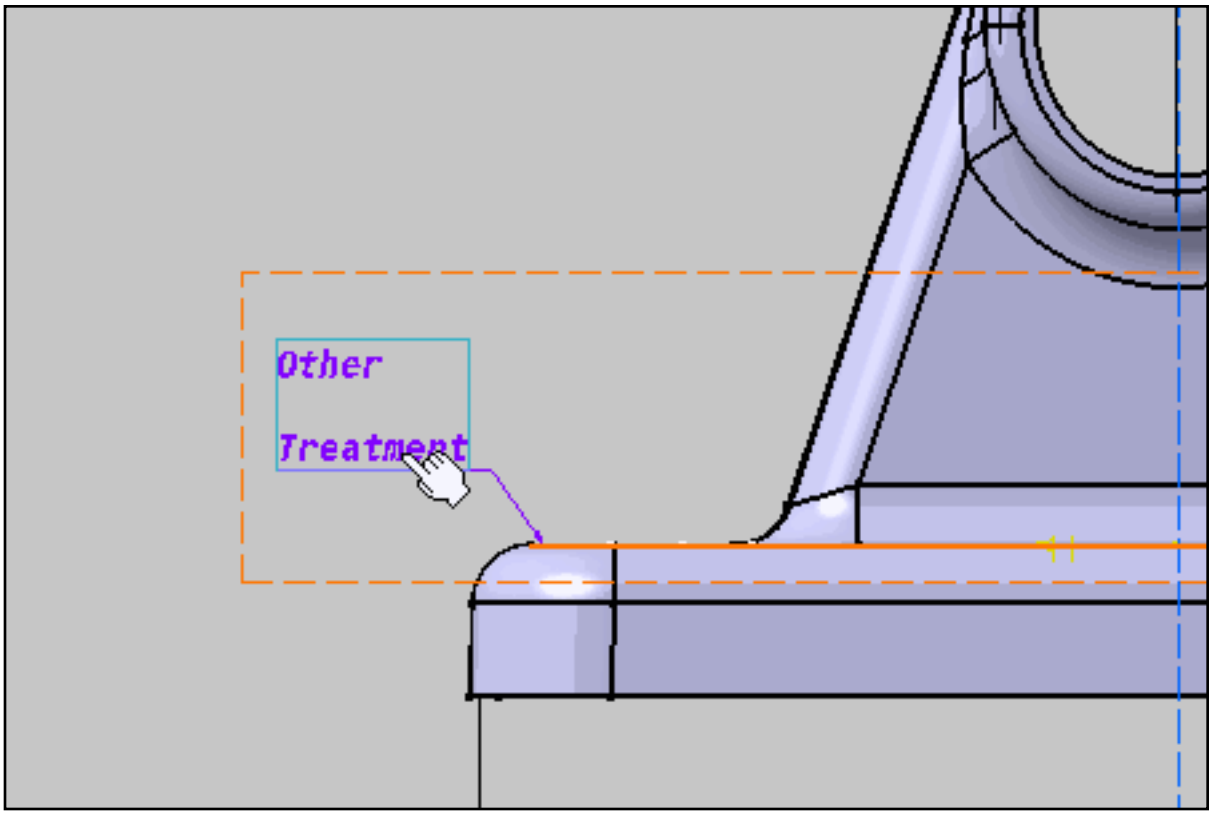
5. Set the **Offset** to 5mm in the **Positioning** dialog box.



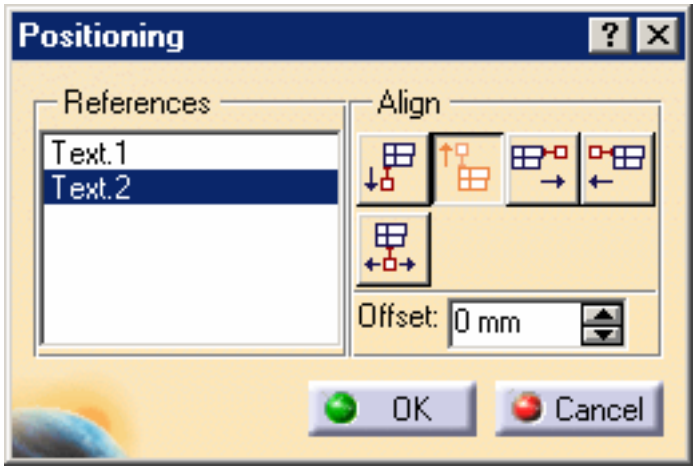
Surface (**Text.2**) is aligned to the top of **Surface Treatment** (**Text.1**).



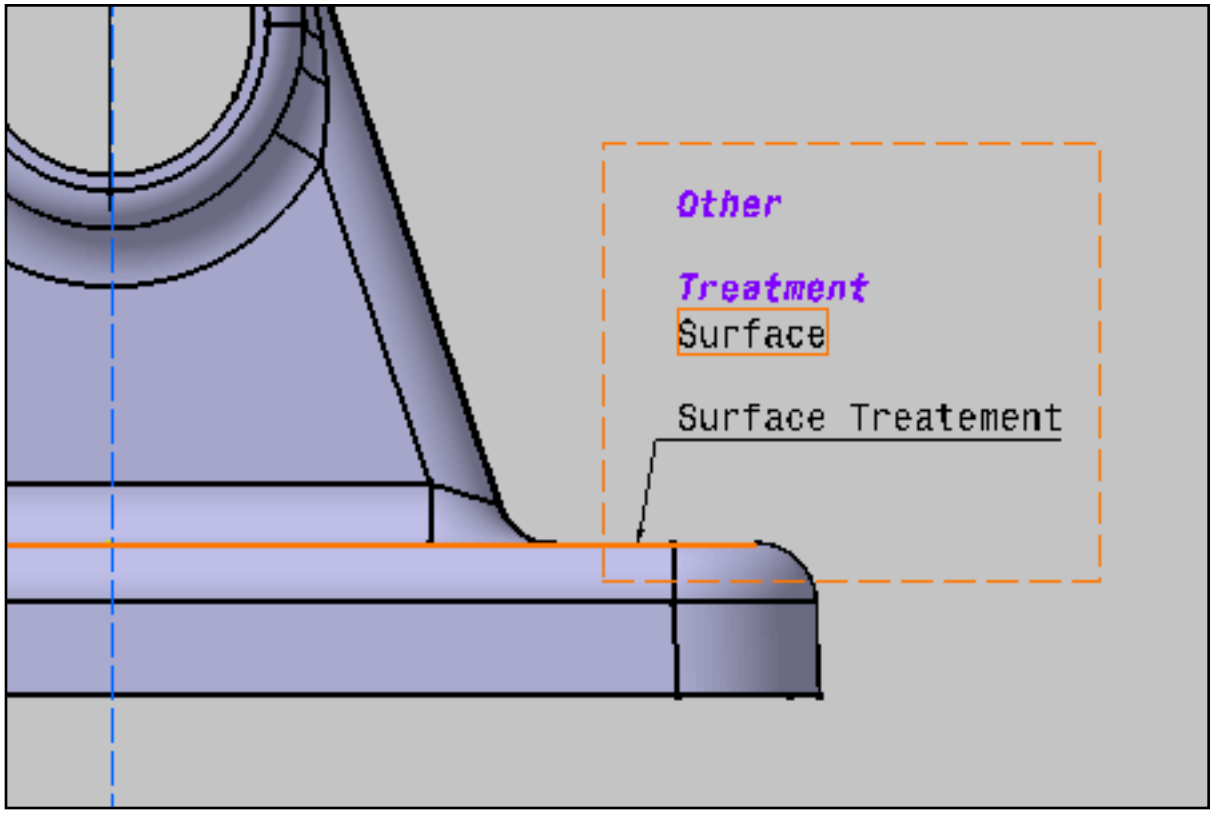
6. Select **Other Surface** (**Text.3**) as the new slave annotation.



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.



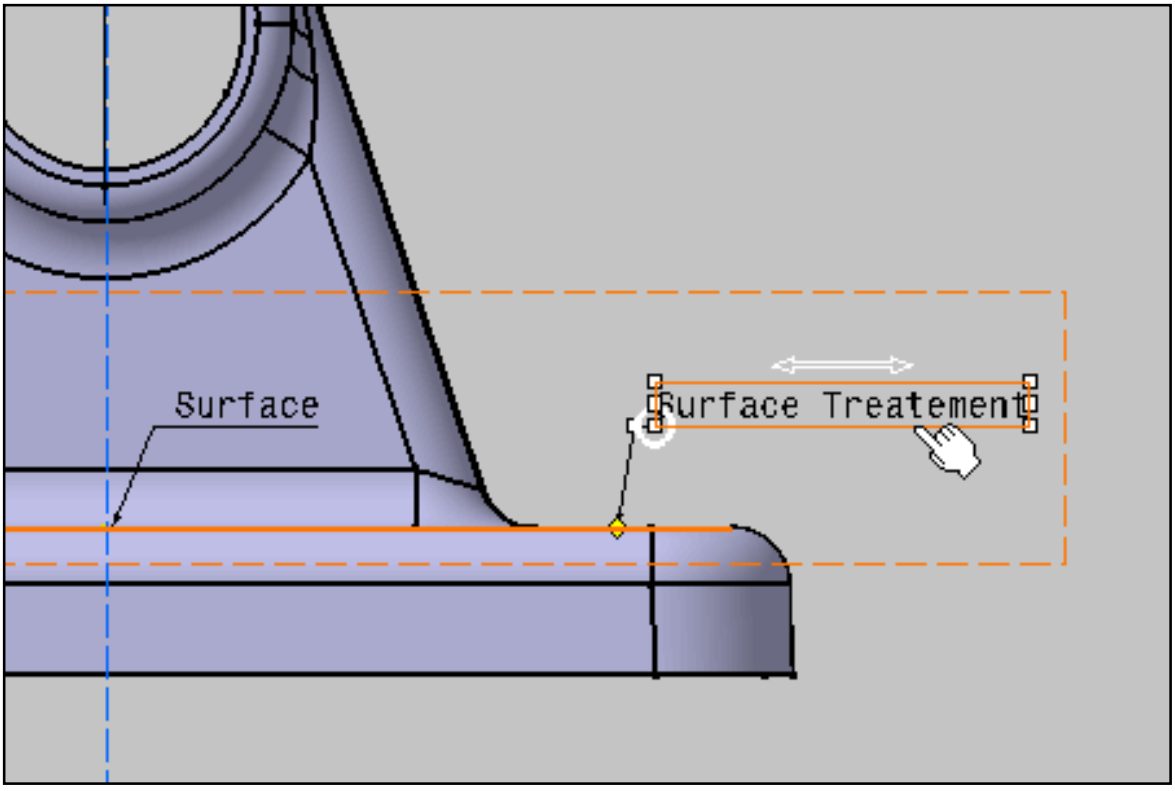
Back Up Next

Making the Position of a Text Associative

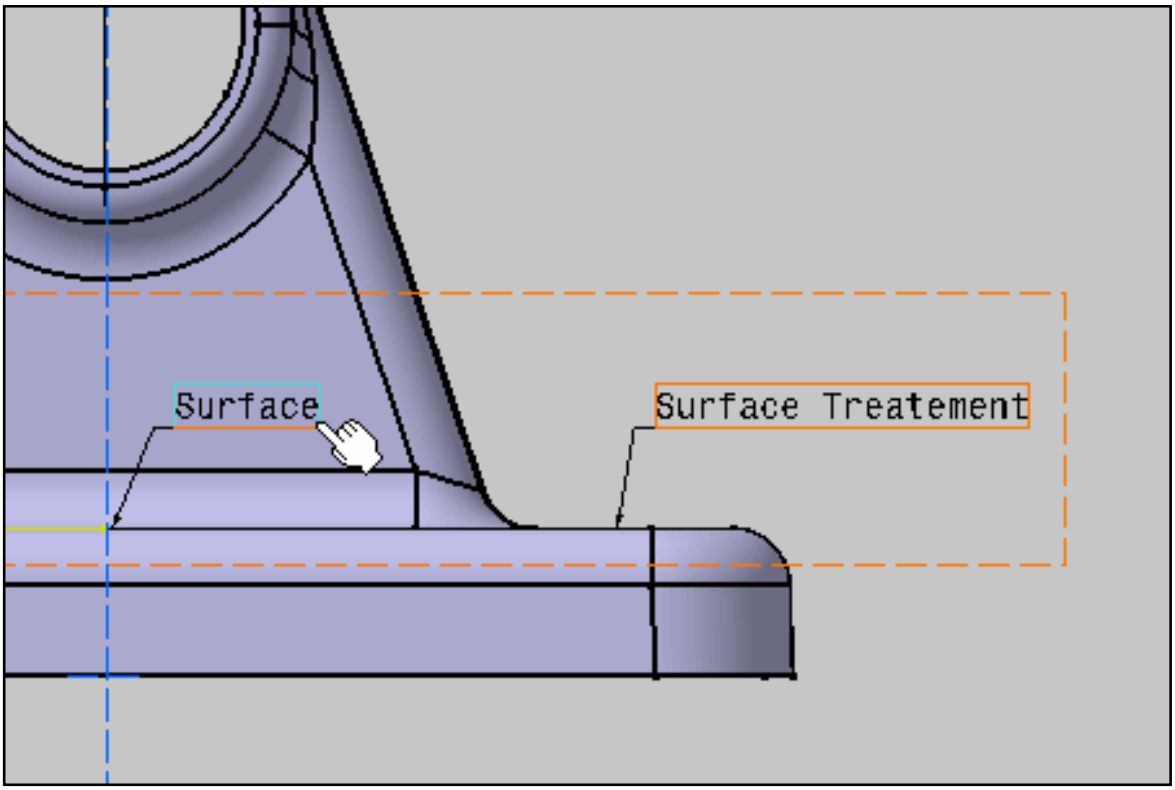
This task shows you how to set a positional link between a text and another element. This allows you to move several annotations in only one interaction.

Open the Tolerancing_Annotations_04 CATPart document.

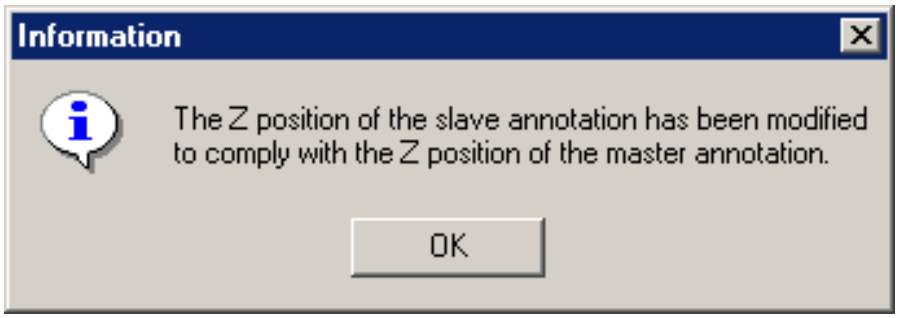
1. Right-click the slave text (text itself, frame or leader) and select the **Annotation Links -> Create Positional Link** contextual menu.



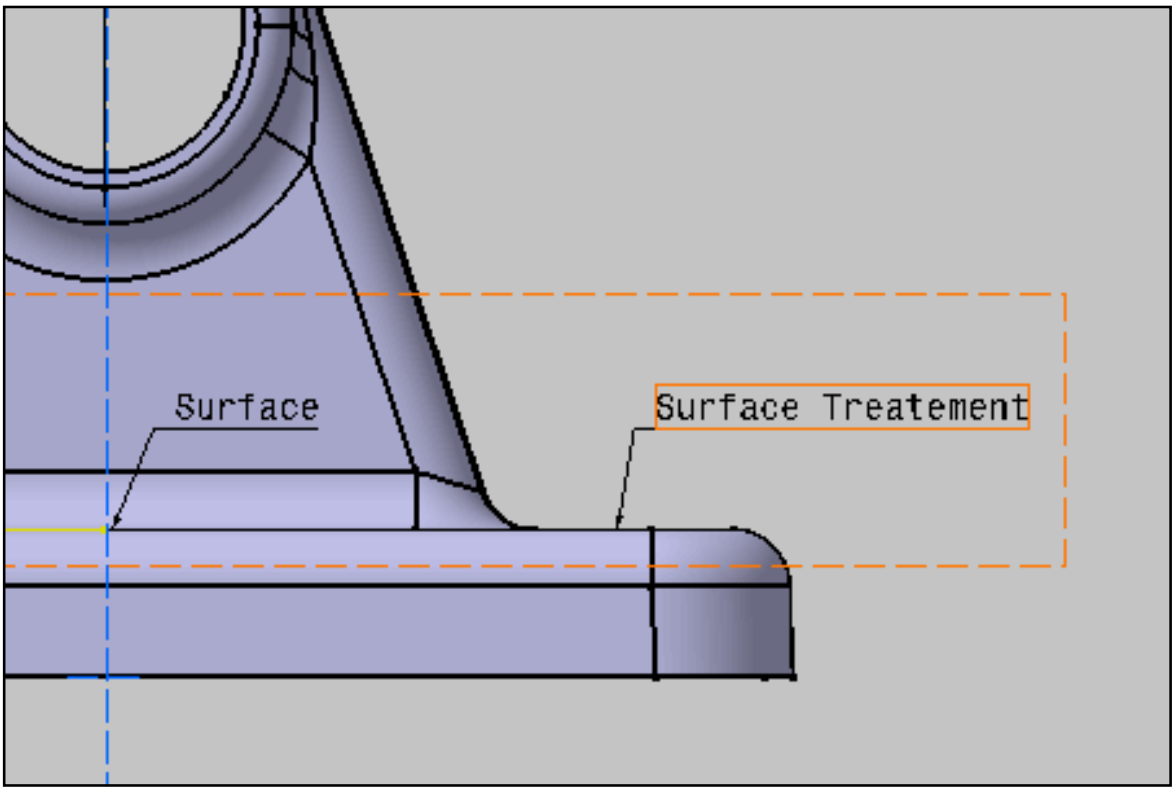
2. Select the master text (text itself, frame or leader).



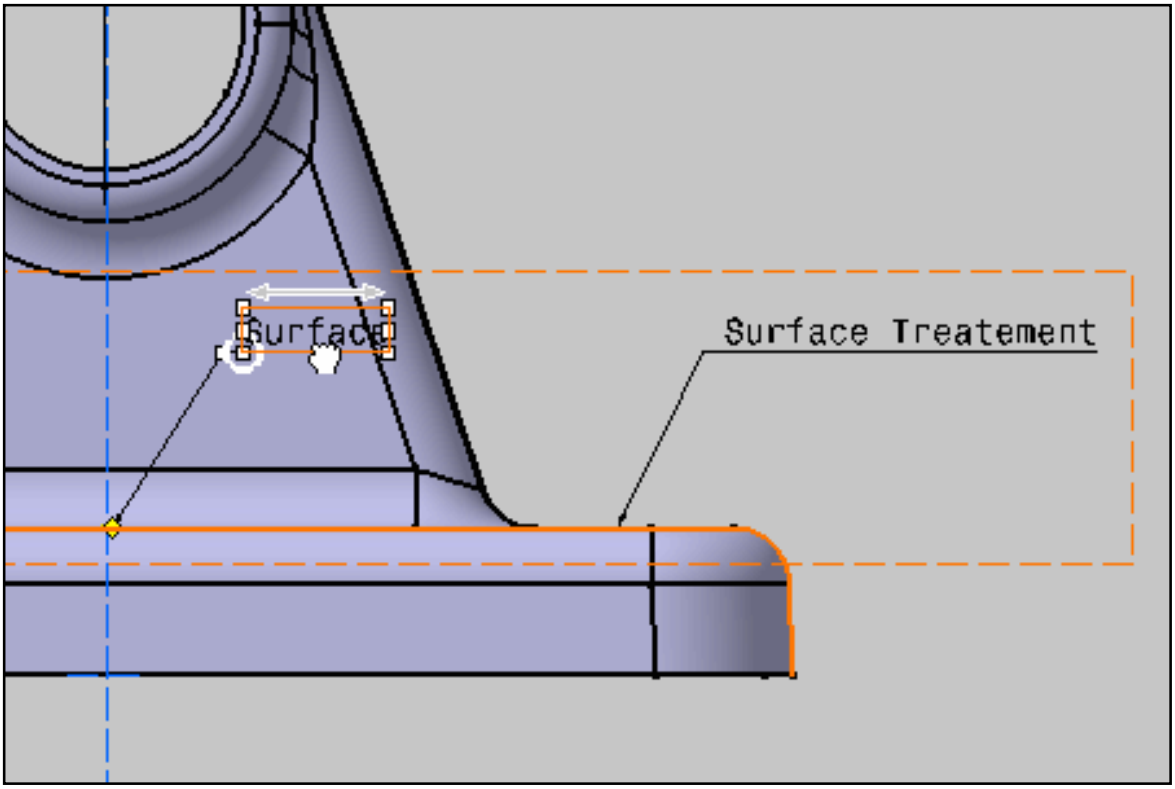
An information popup appears to warn you that the slave text is now at the master elevation.



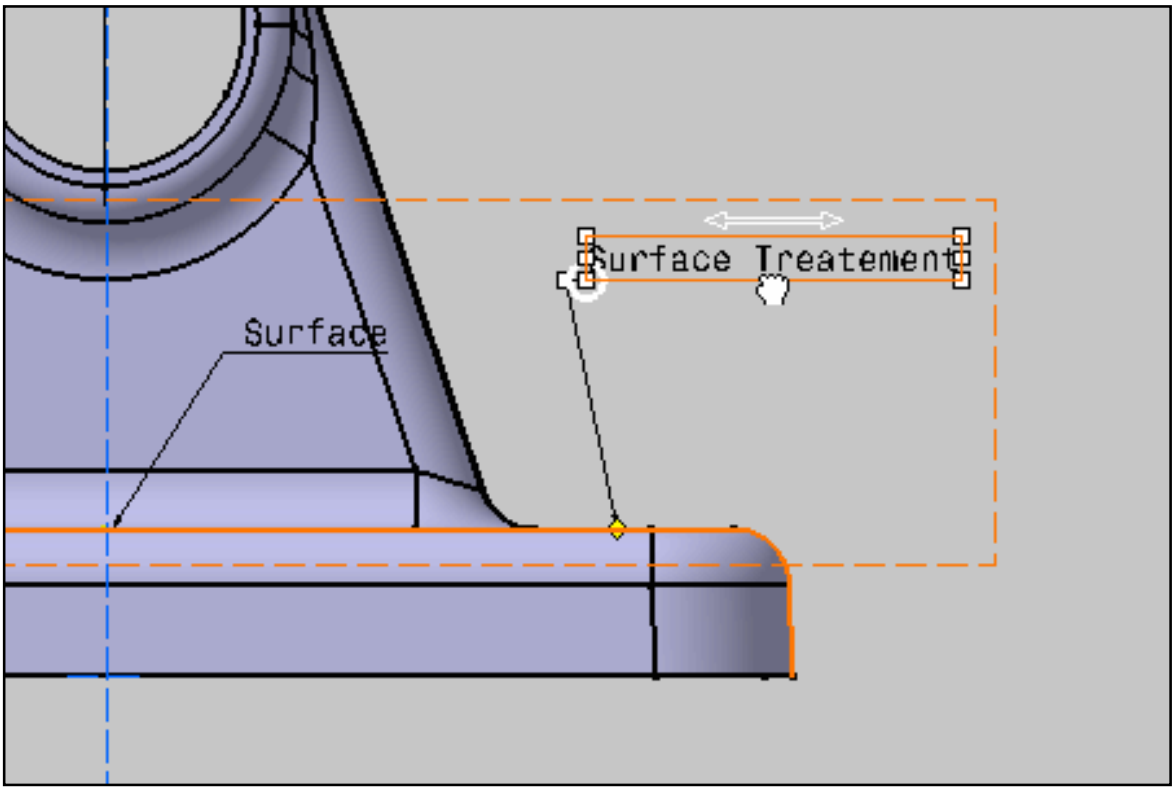
Master and slaves texts must belong to the same active view and associated geometrical element.



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

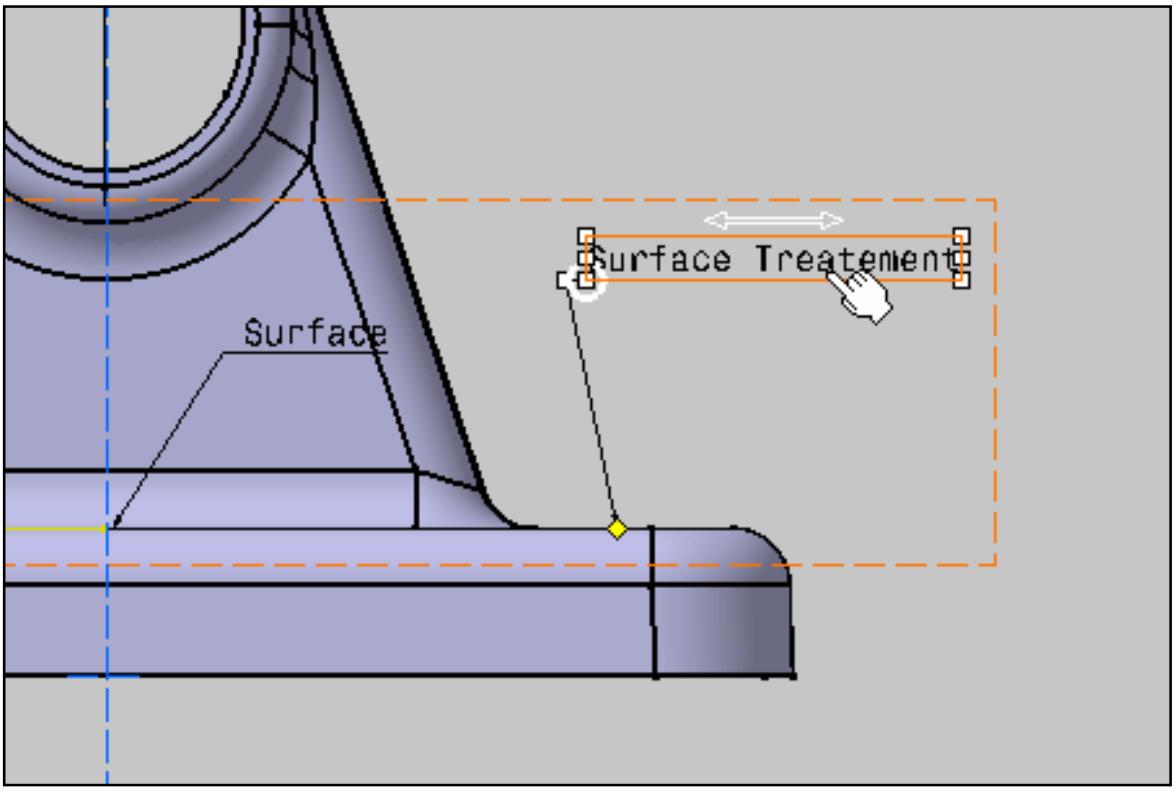


4. Now, if you move the slave text you selected, only this annotation is moved.

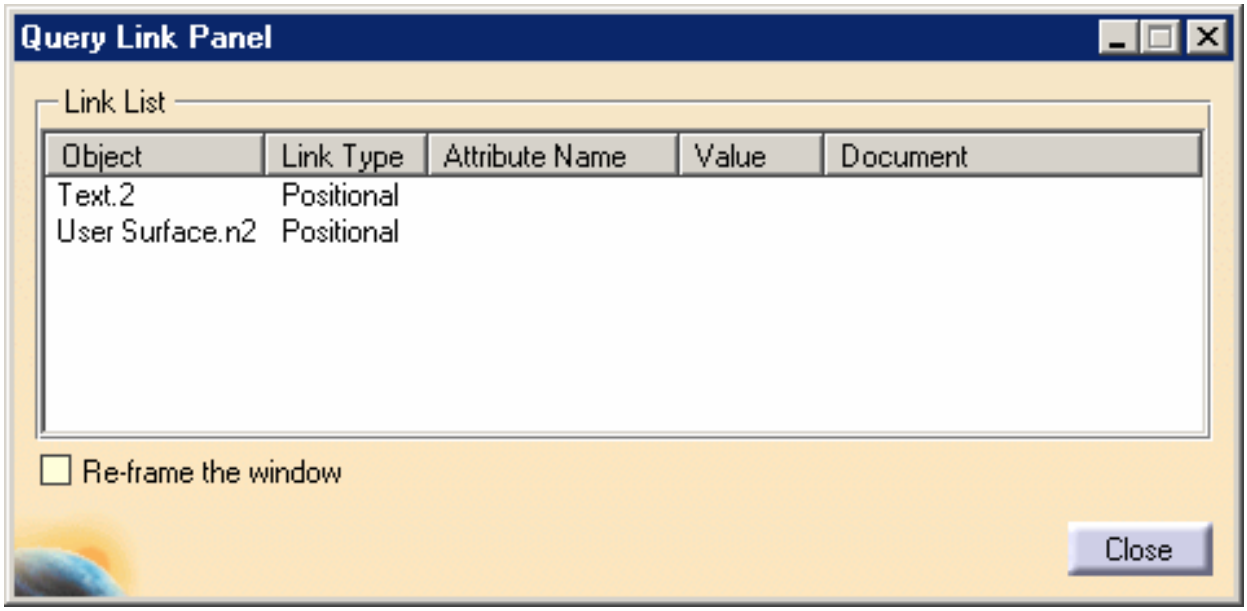


To delete the associativity, right-click the slave text and select the **Annotation Links -> Delete Positional Link** contextual menu.

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.

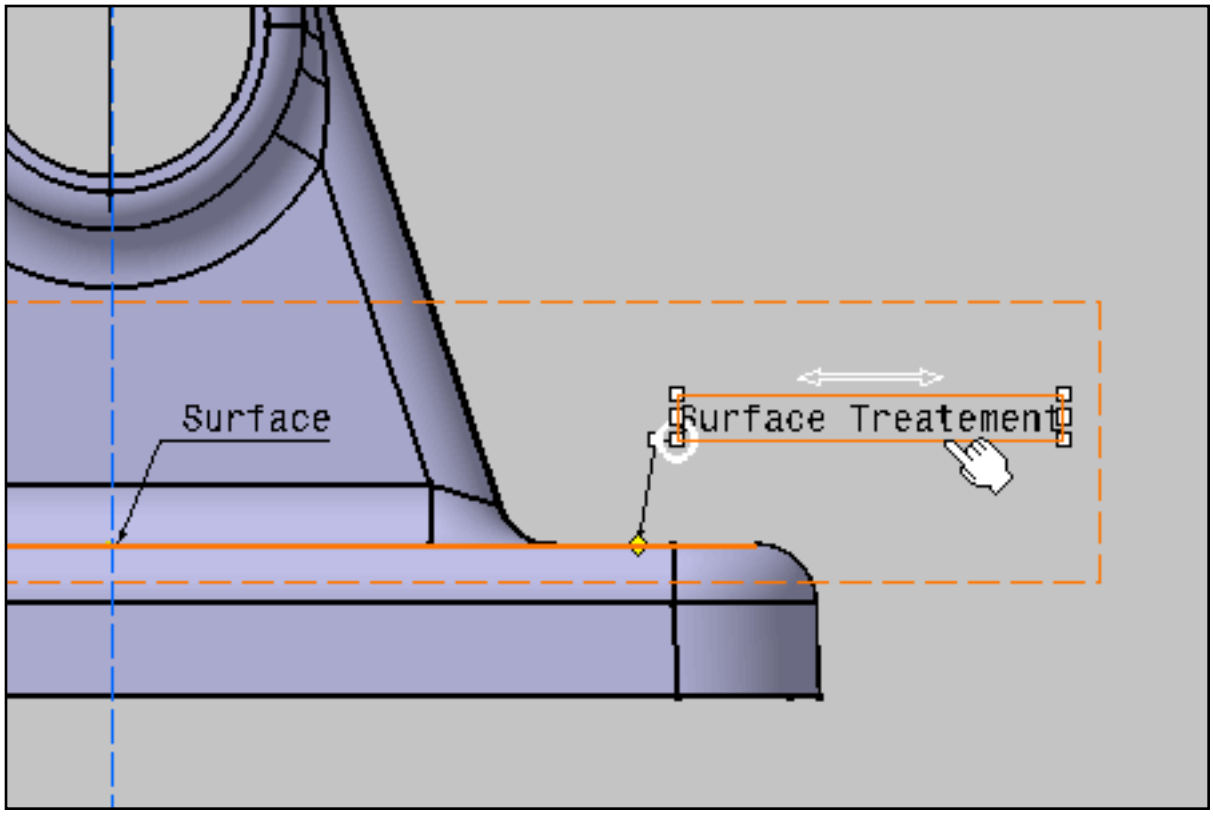
Back Up Next

Making the Orientation of a Text Associative

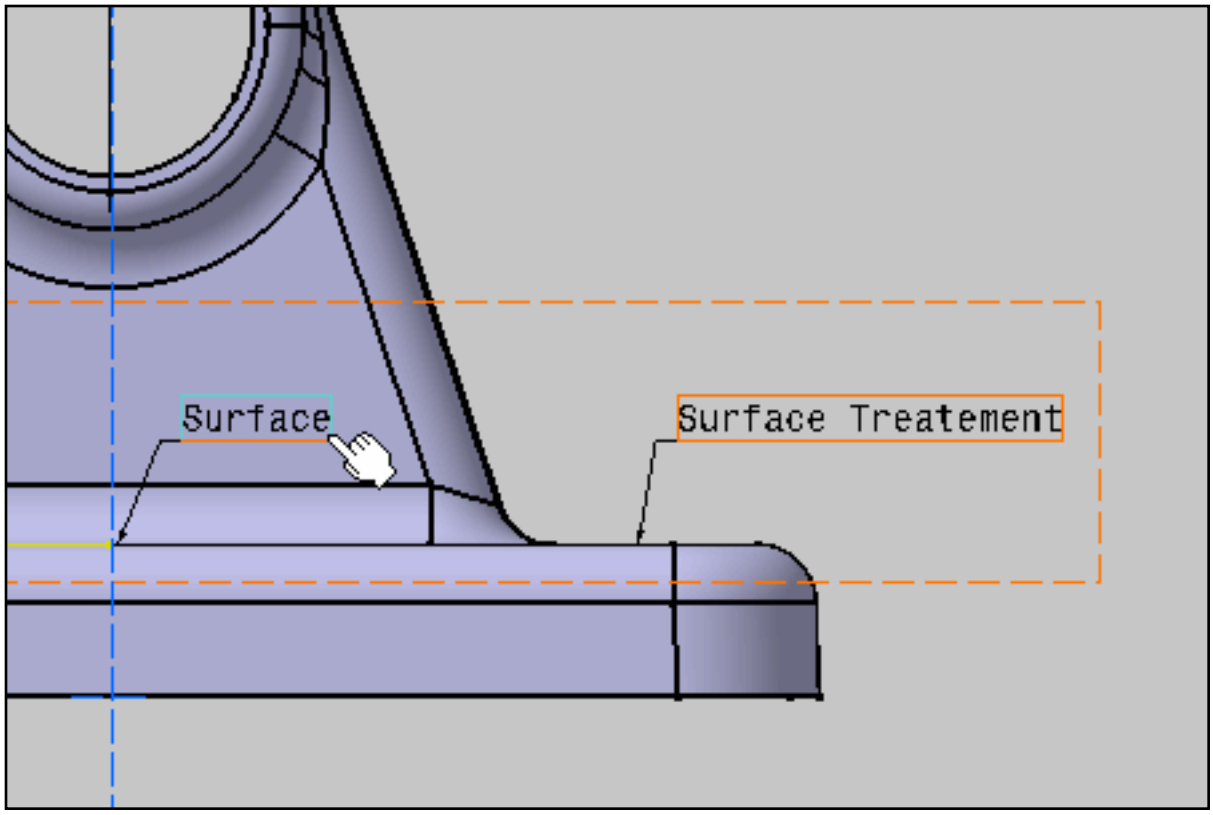
This task shows you how to set a orientation link between a text and another element.
This allows you to rotate several annotations in only one interaction.

Open the Tolerancing_Annotations_04 CATPart document.

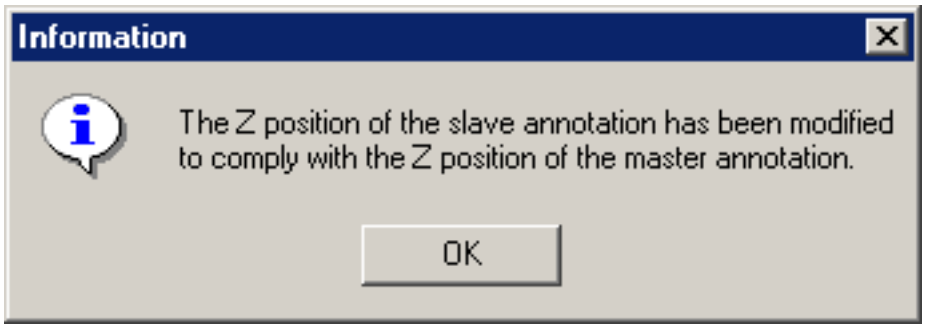
1. Right-click the slave text (text itself, frame or leader) and select the **Annotation Links -> Create Orientation Link** contextual menu.



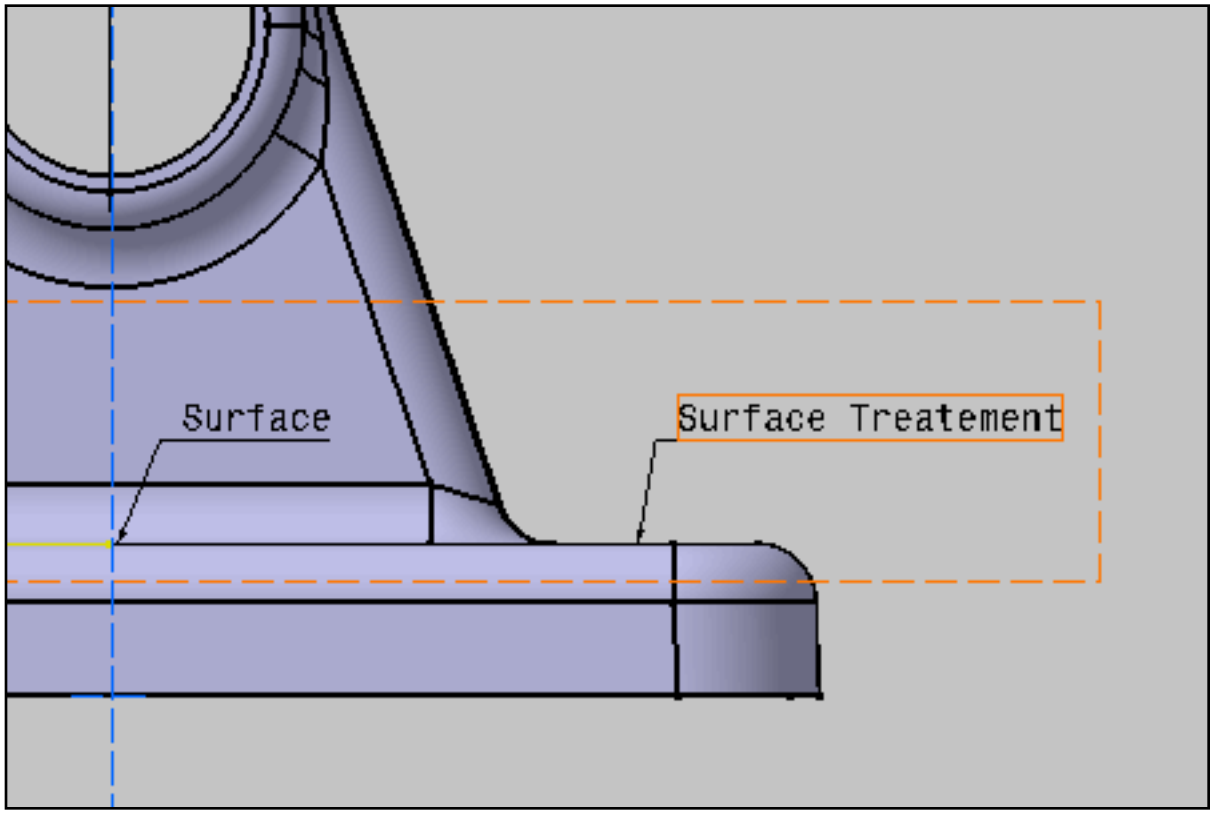
2. Select the master text (text itself, frame or leader).



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

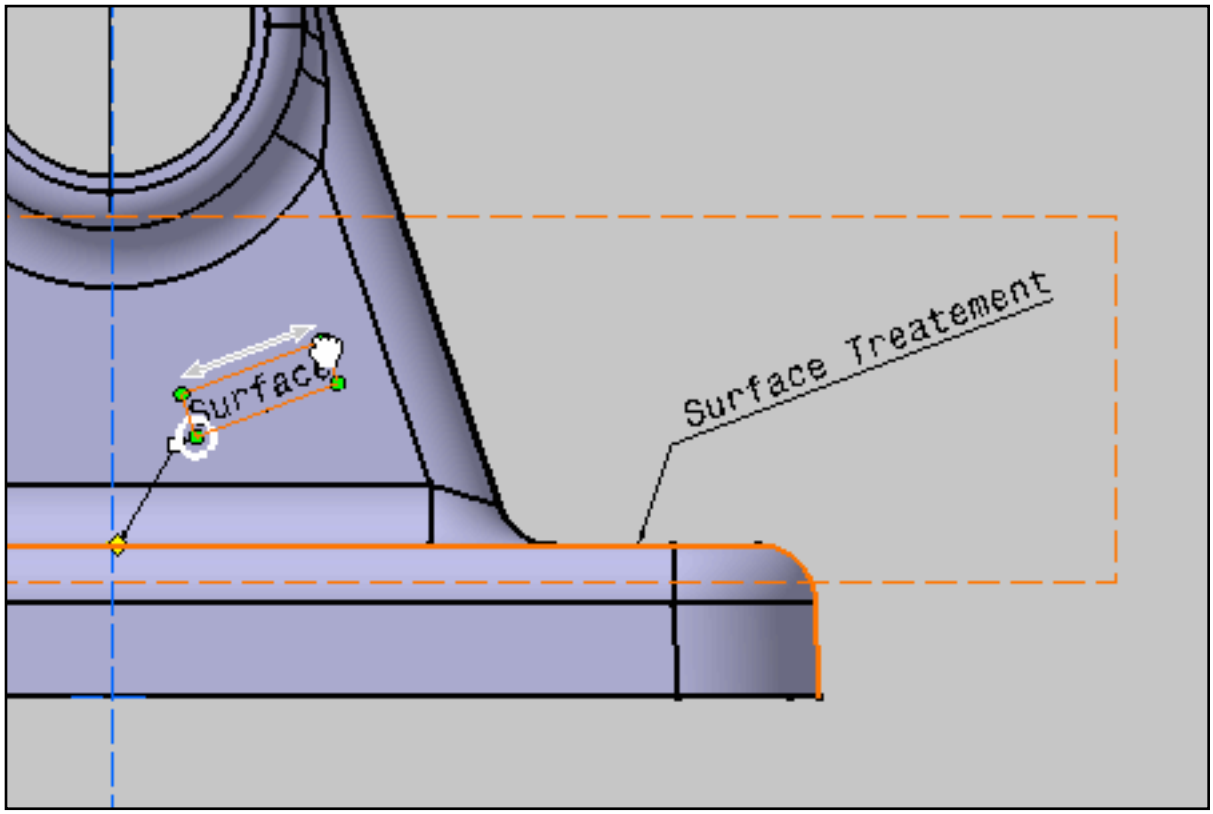


Master and slaves texts must belong to the same active view and associated geometrical element.

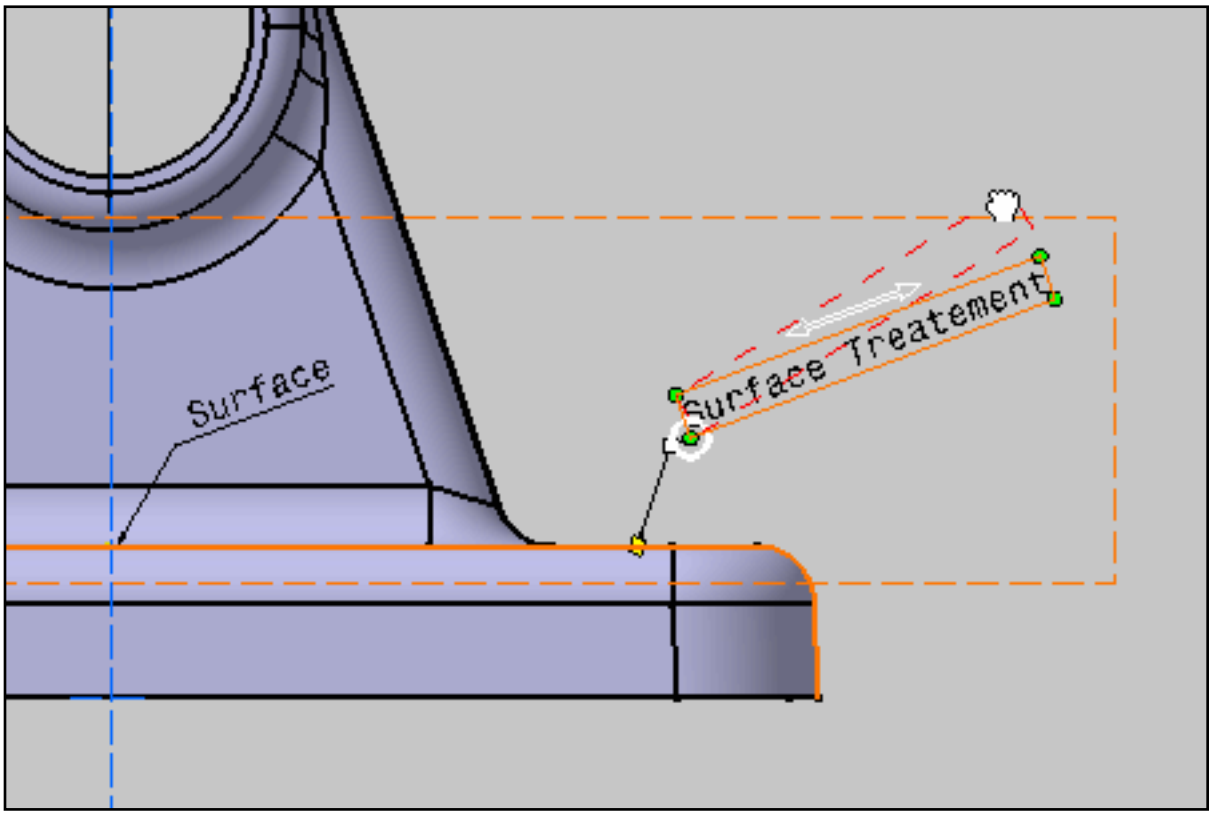


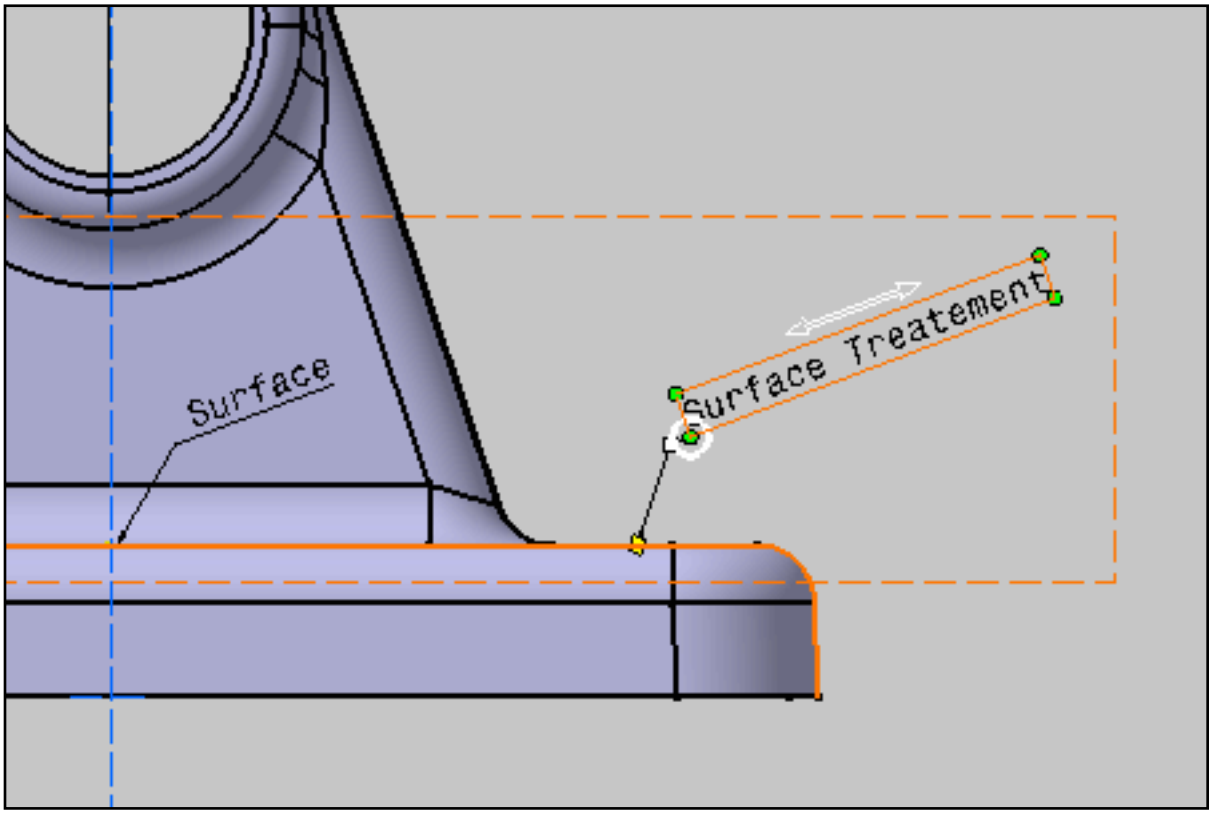
3. Select the **Free rotation** icon: 

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.



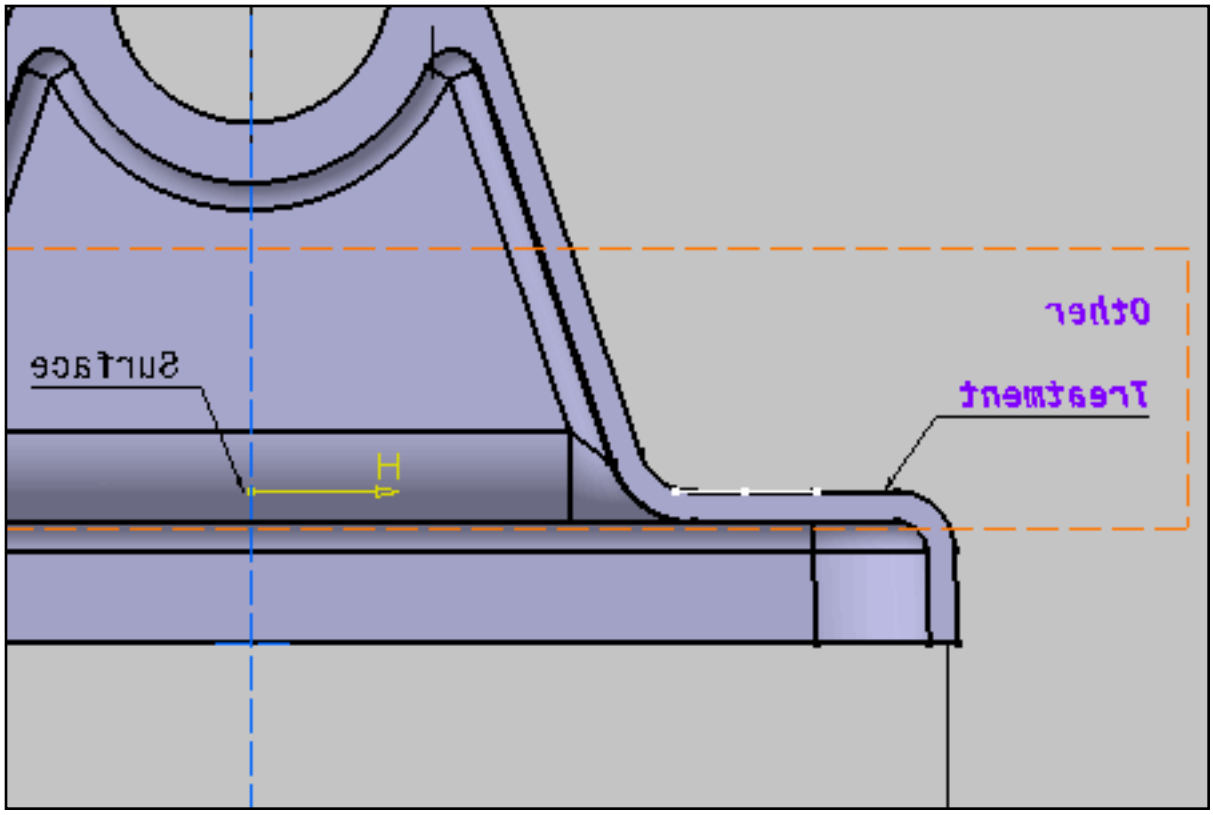
Back Up Next

Mirroring Annotations

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

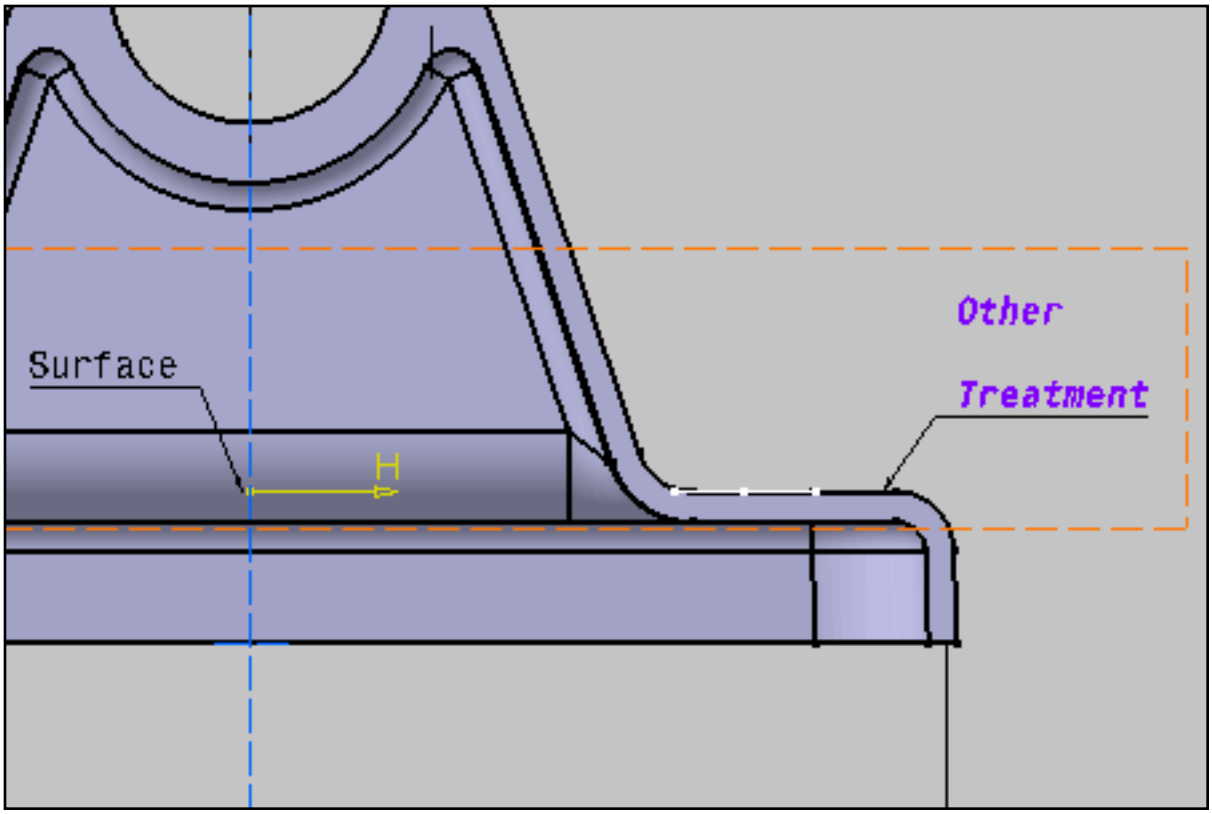
Open the Tolerancing_Annotations_04 CATPart document.

1. Turn the part to show annotations reversed.



2. Click the **Mirror Annotations** icon 

All reversed annotations are mirrored.



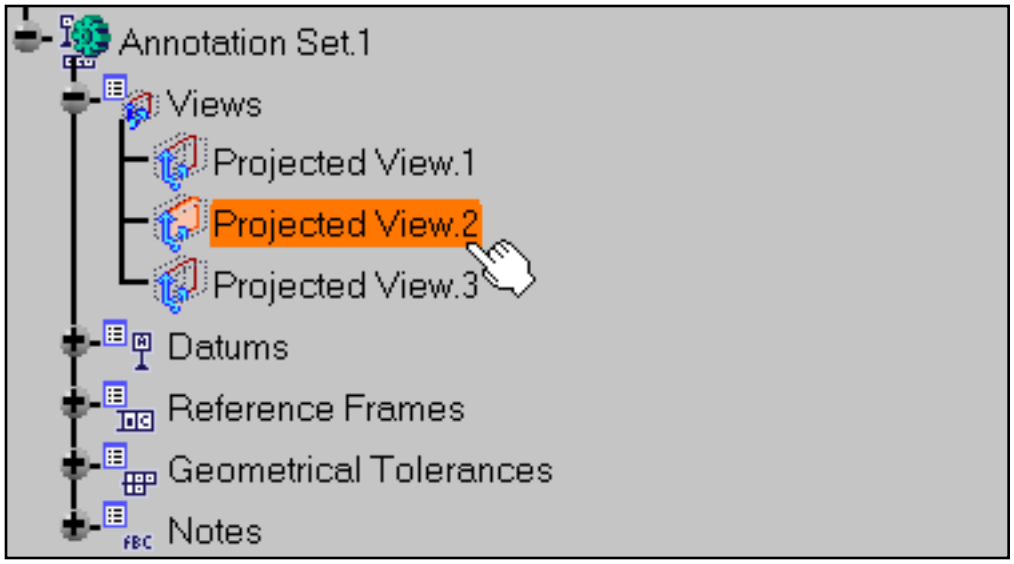
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Clipping Annotations Plane

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

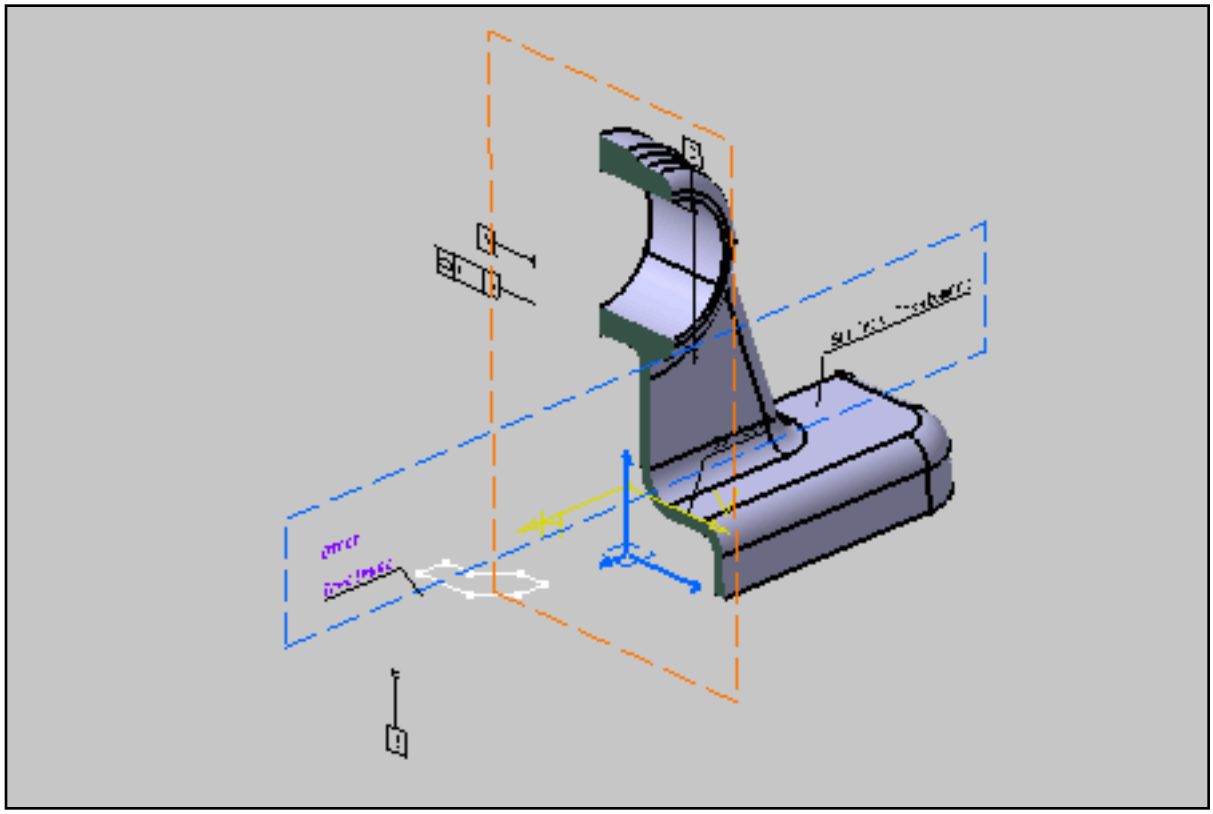
Open the Tolerancing_Annotations_04 CATPart document.

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.



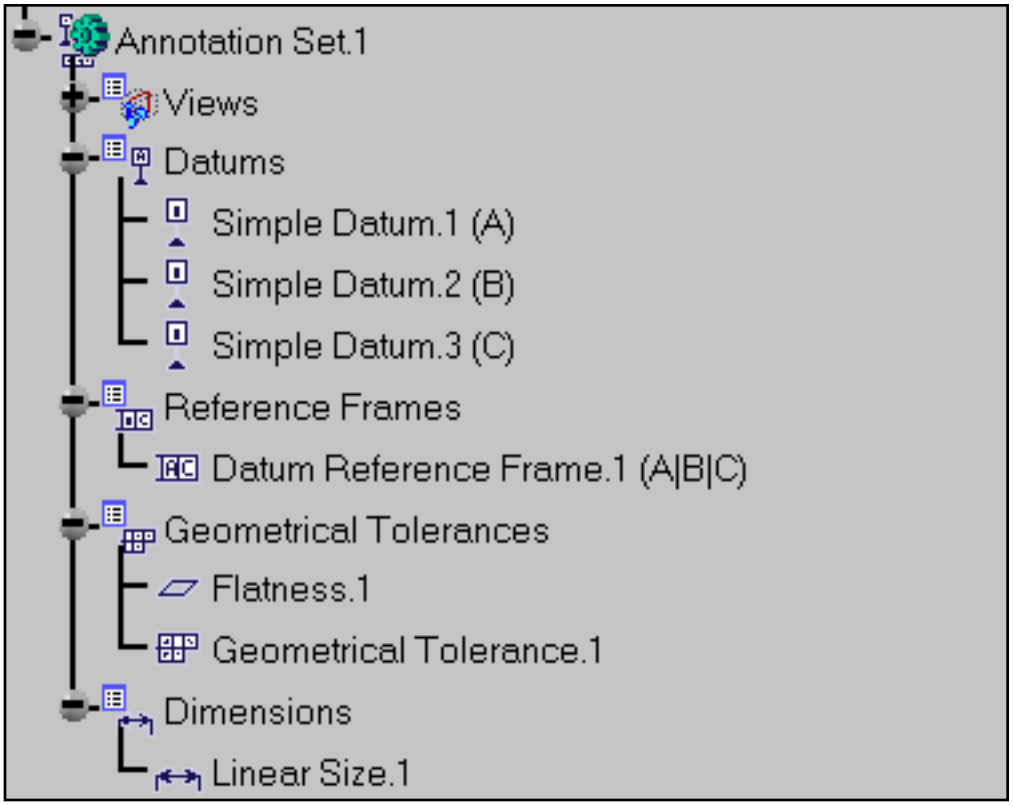
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Marking Non-semantic Annotations

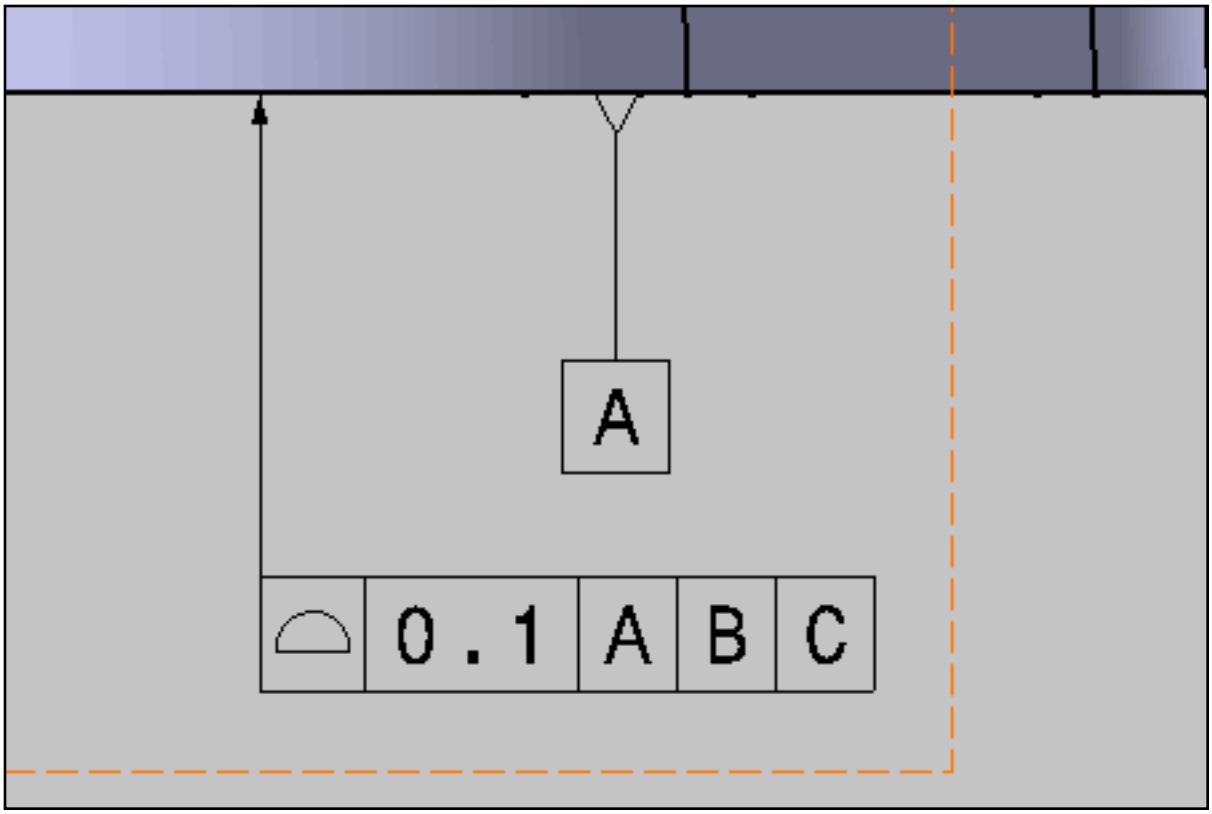
This task shows you how to mark non-semantic annotations with a wavy red line in the specification tree and geometry.
It allows you to graphically distinguish non-semantic annotations (datum, datum targets, geometric tolerances, dimensions) from semantic one, see Concepts.

Open the Tolerancing_Annotations_03 CATPart document.

The specification tree looks like this:

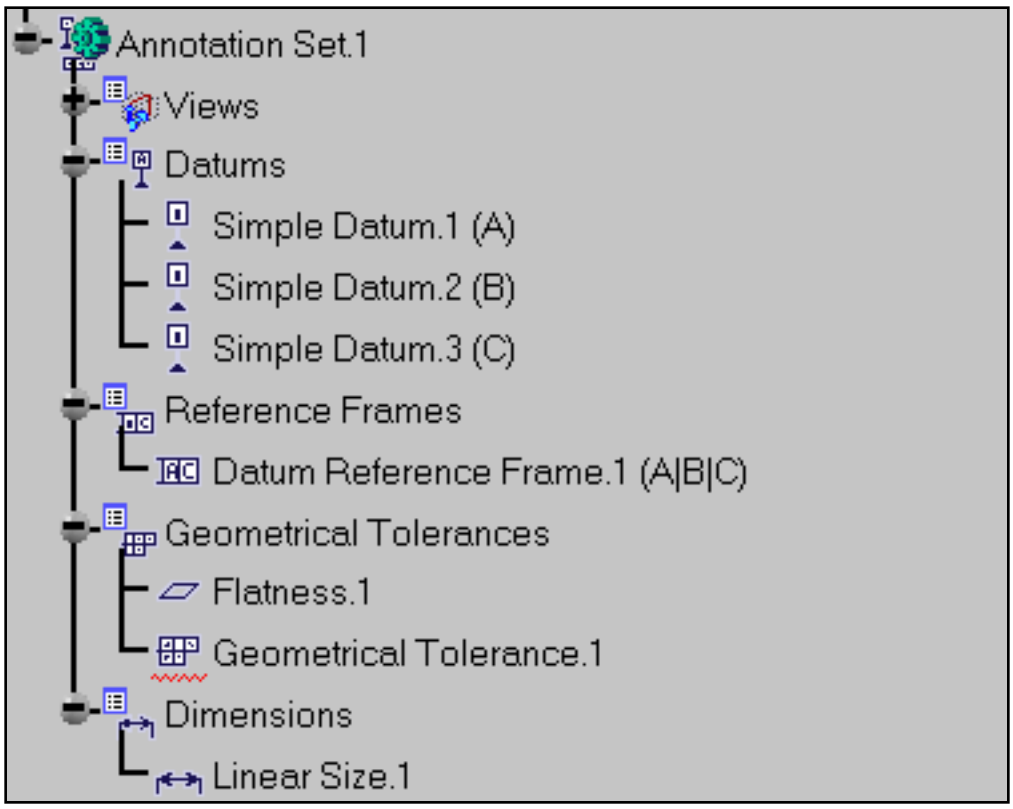


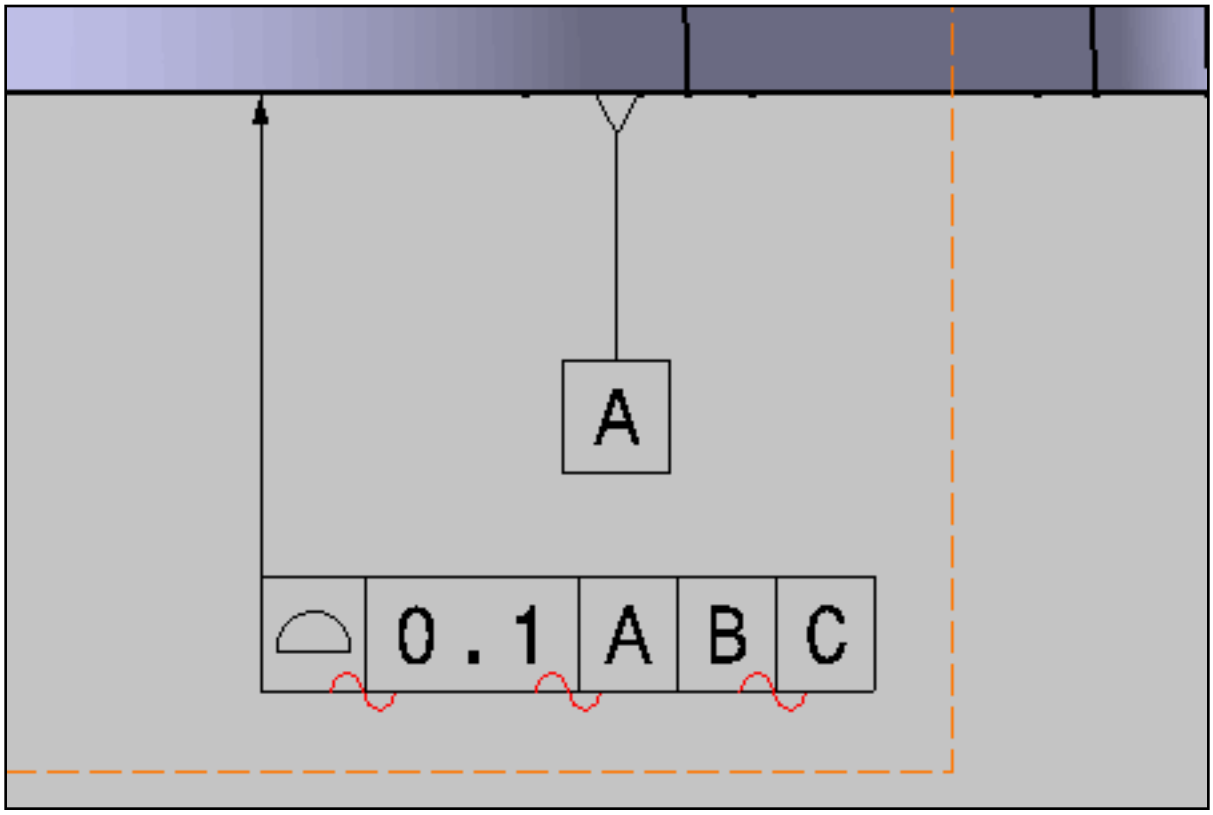
The geometry looks like this:



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:





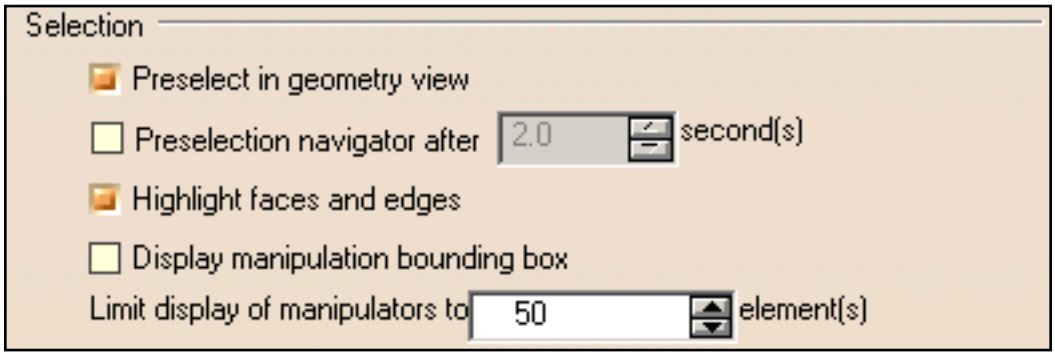
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Setting Annotation Parallel to Screen

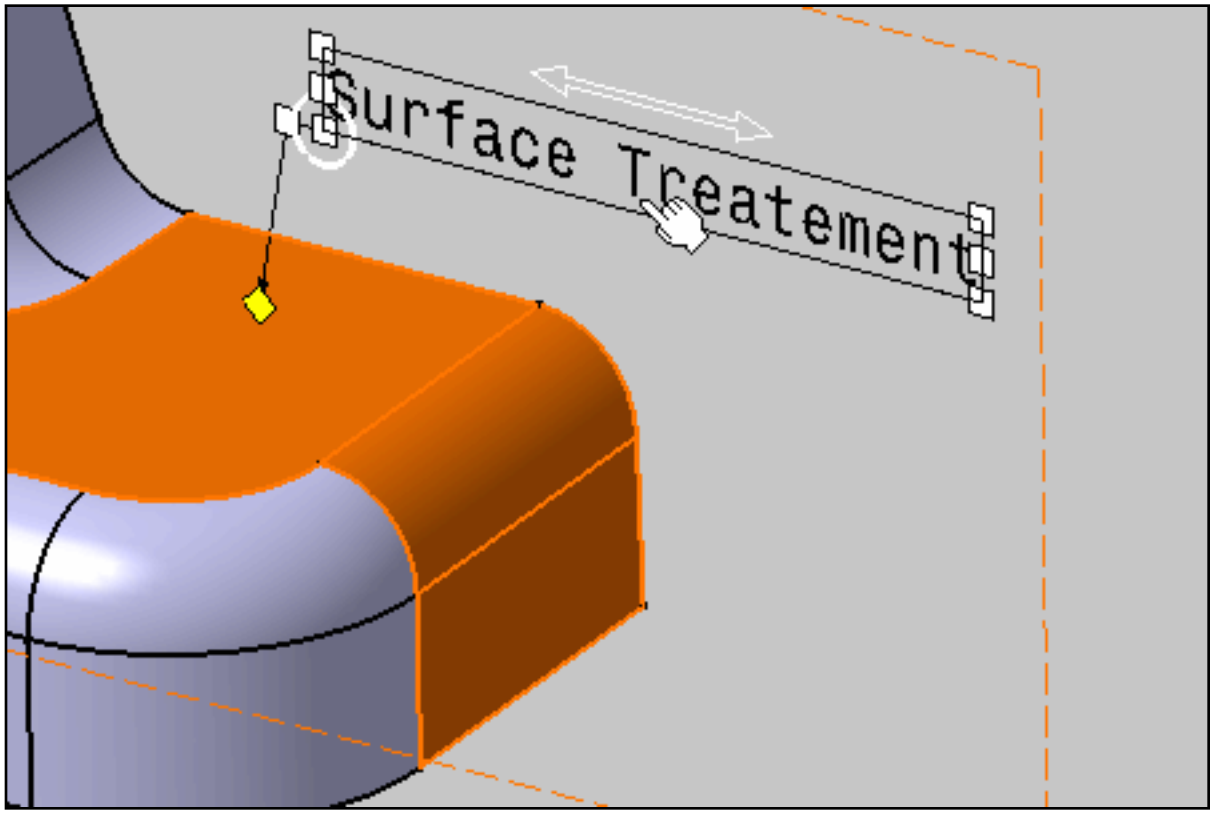
This task shows you how to set an annotation text parallel to screen.
The operating mode described here applies to text, flagnote and note object attribute too.

Open the Tolerancing_Annotations_04 CATPart document.

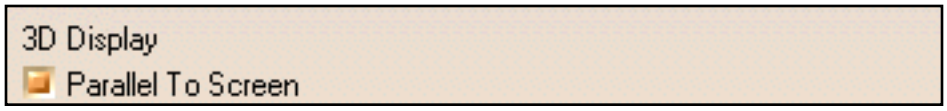
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



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

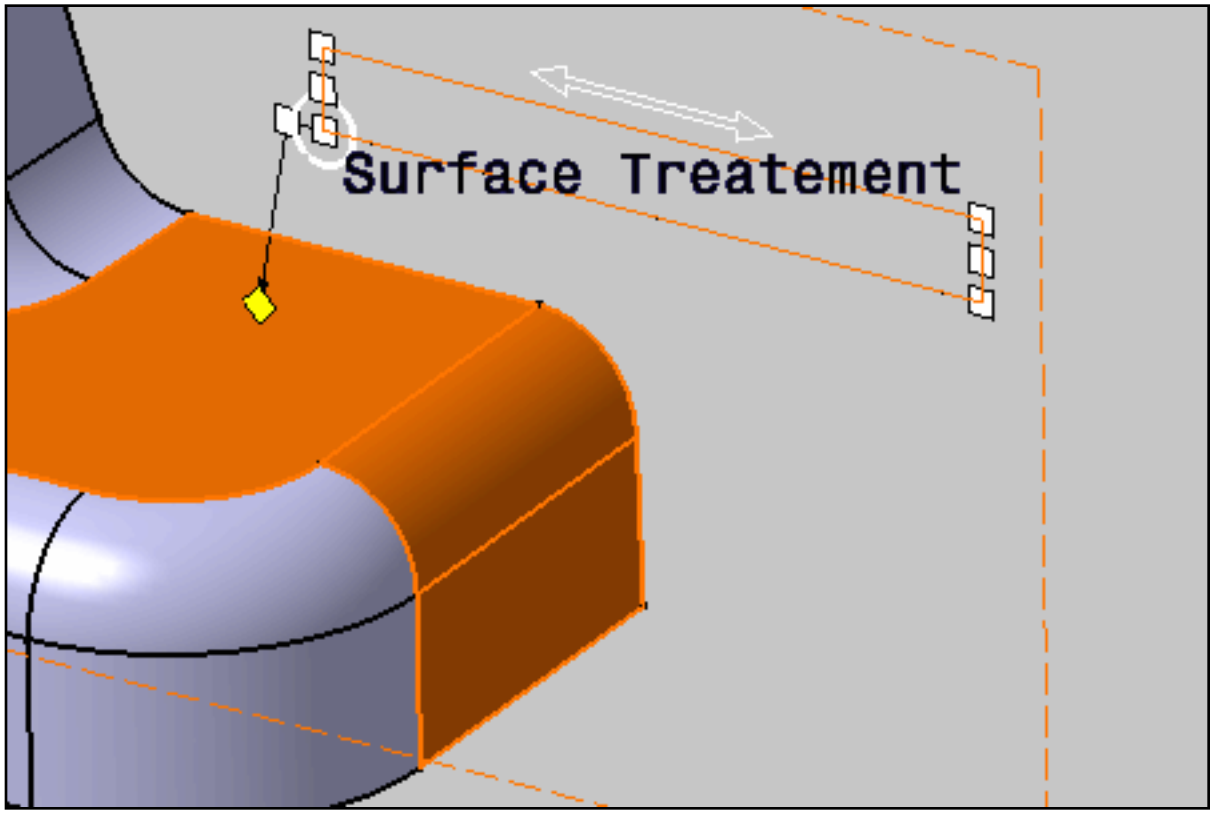


2. Check **Parallel to Screen** option.



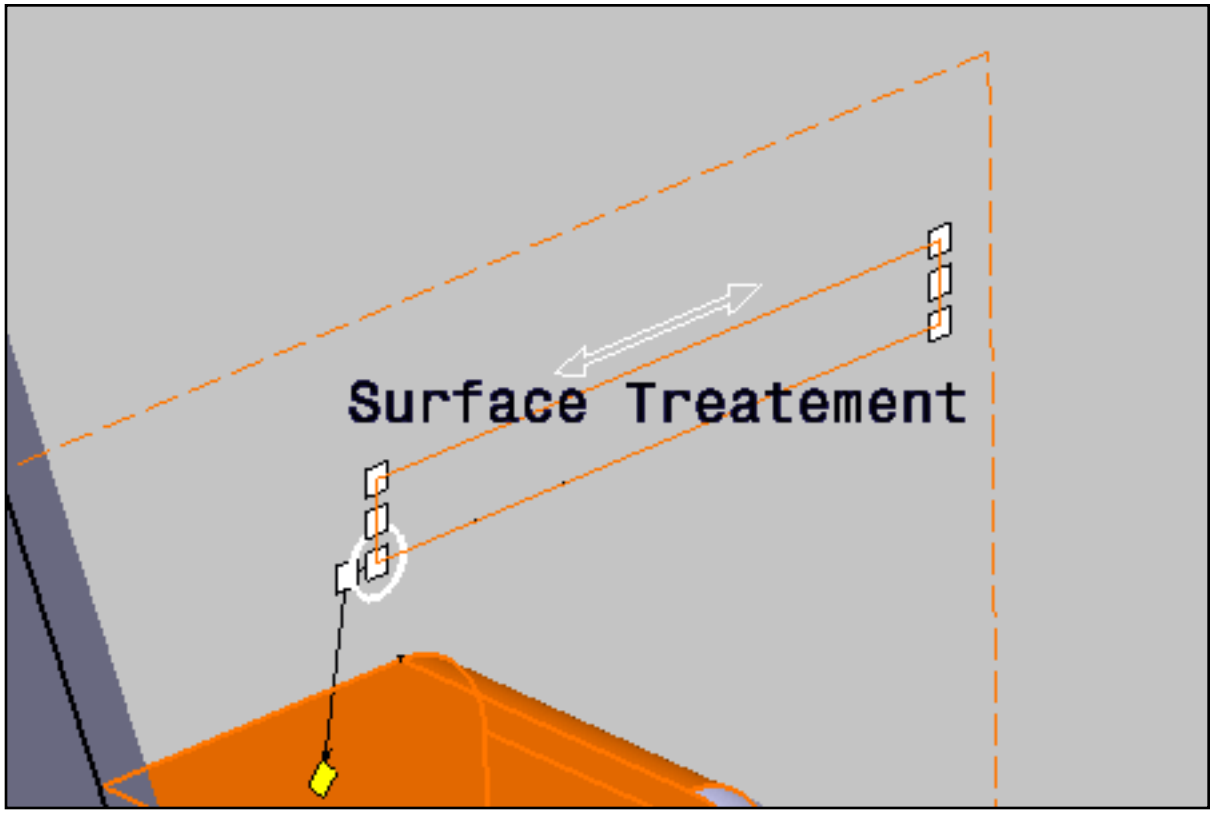
3. Click the **OK**.

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.





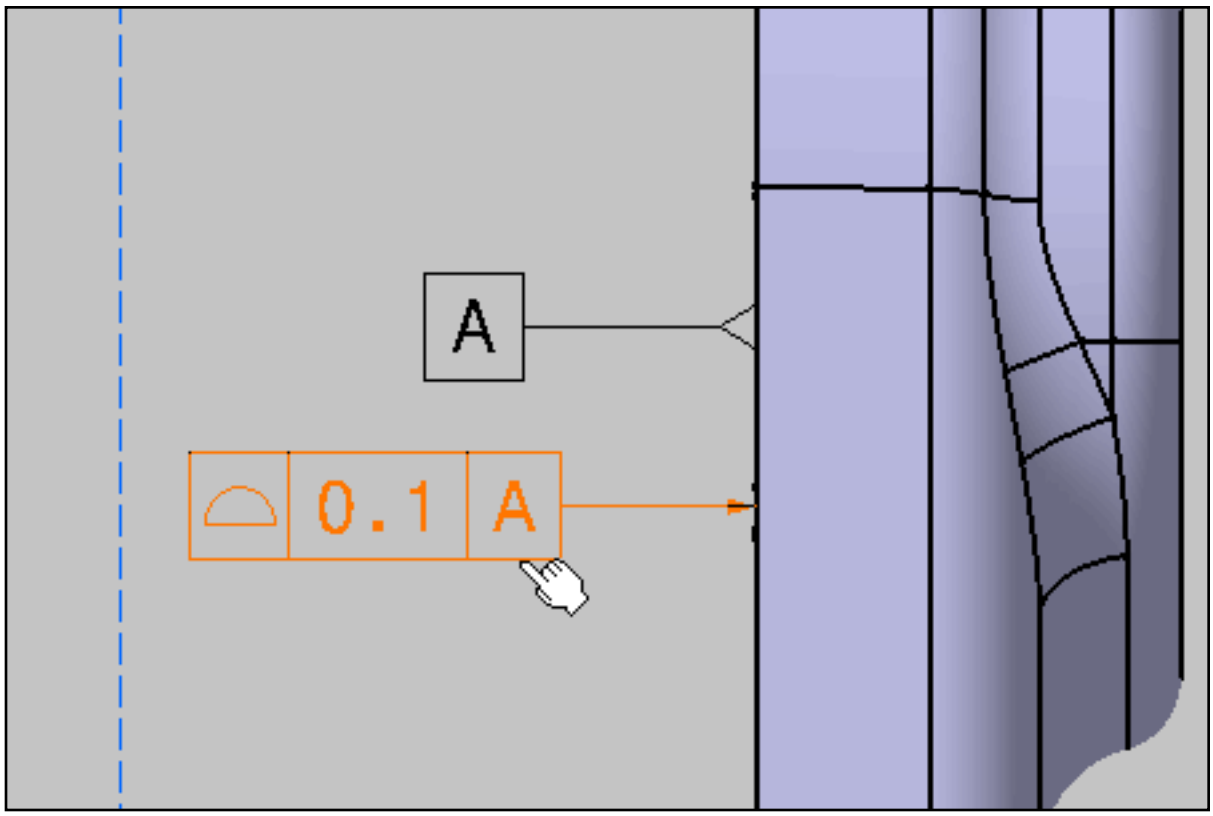
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Replacing a Datum Reference Frame

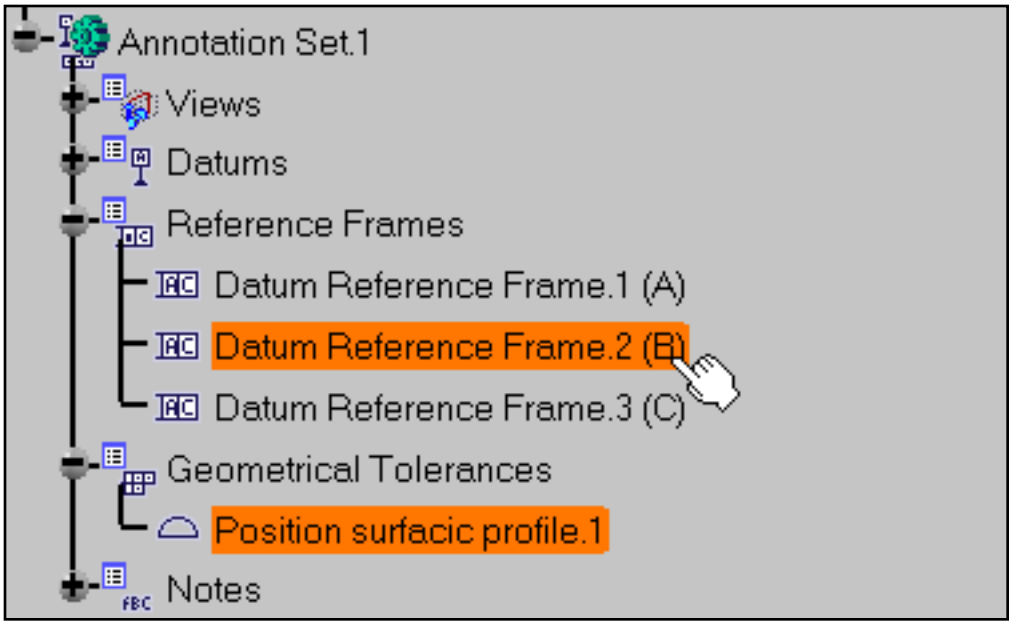
This task shows you how to replace a datum reference frame in an annotation.

Open the Tolerancing_Annotations_04 CATPart document.

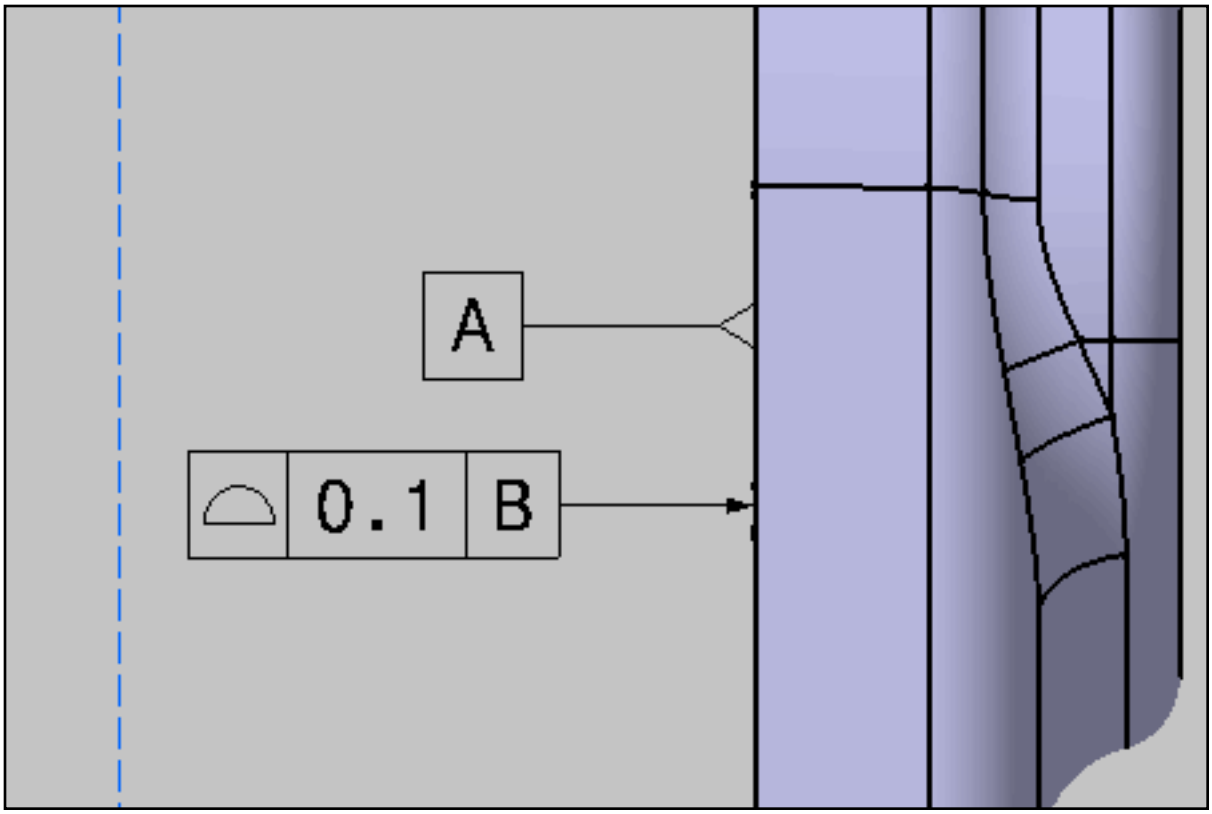
1. Right-click the annotation and select the **Replace Datum Reference Frame** contextual command.



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



The datum reference frame is replaced.



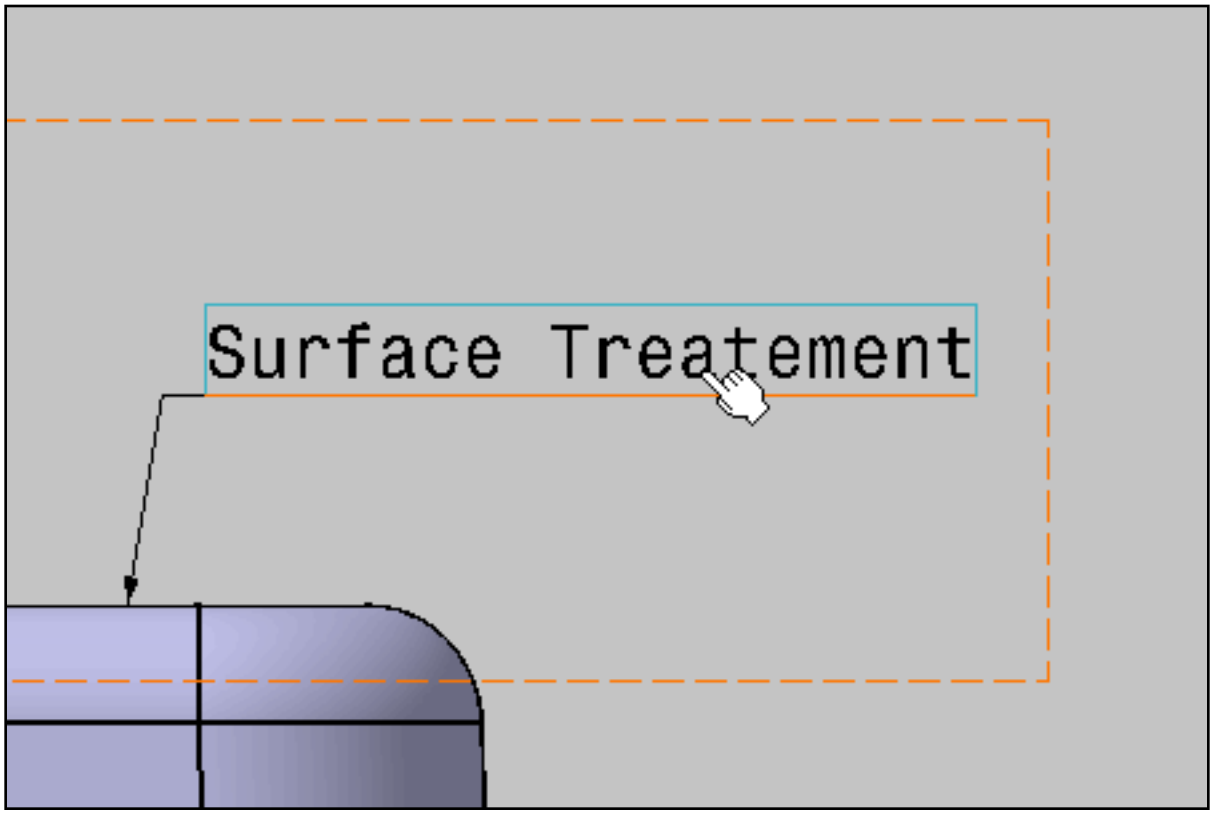
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Using 3D Grid

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.

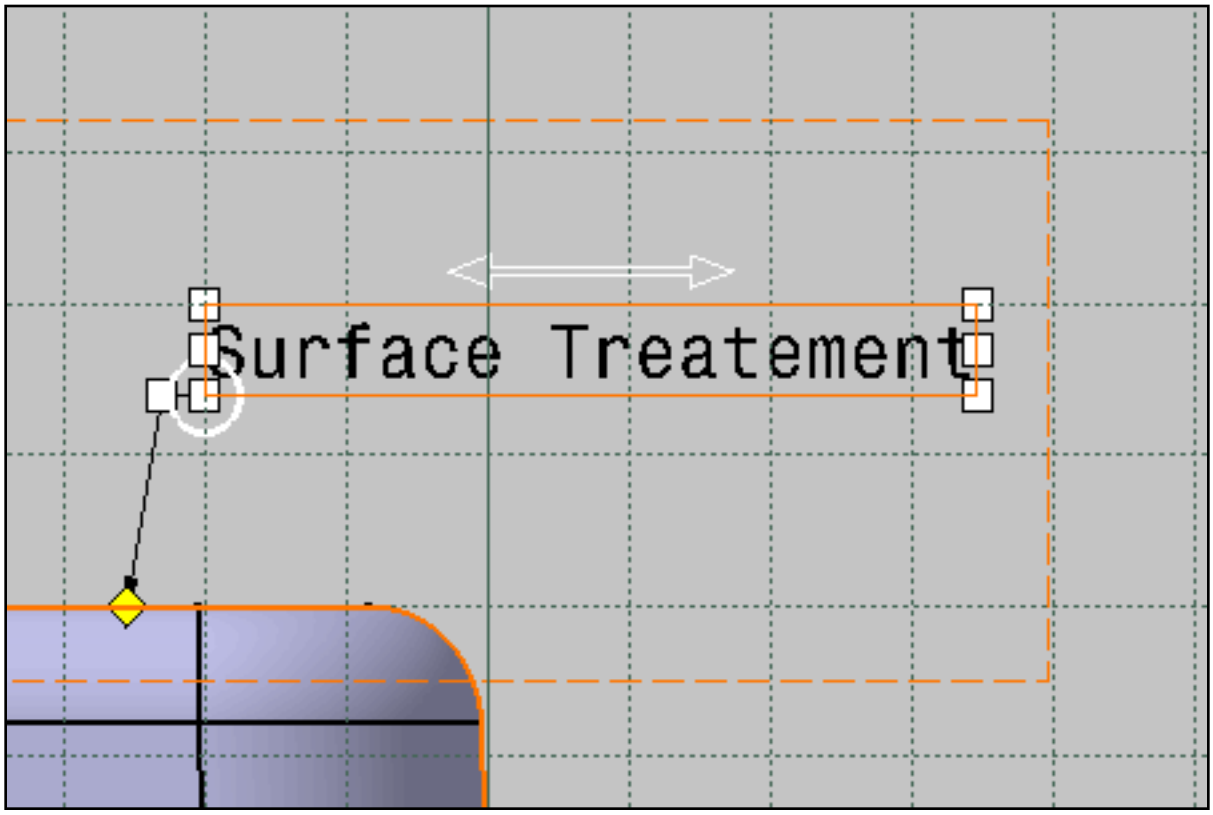
Open the Tolerancing_Annotations_04 CATPart document.

1. Select the annotation.

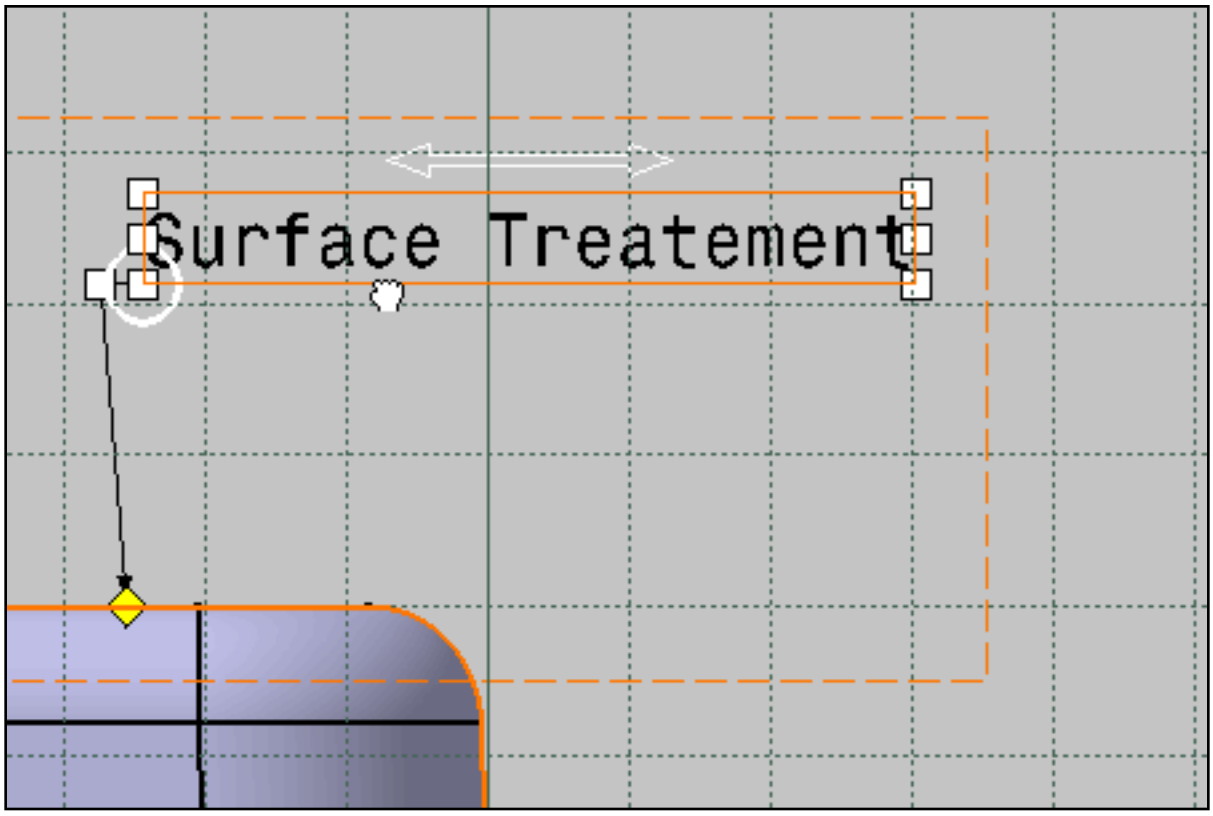


2. Click the **Display 3D Grid** icon:

The 3D grid appears.



3. Drag anywhere the annotation.

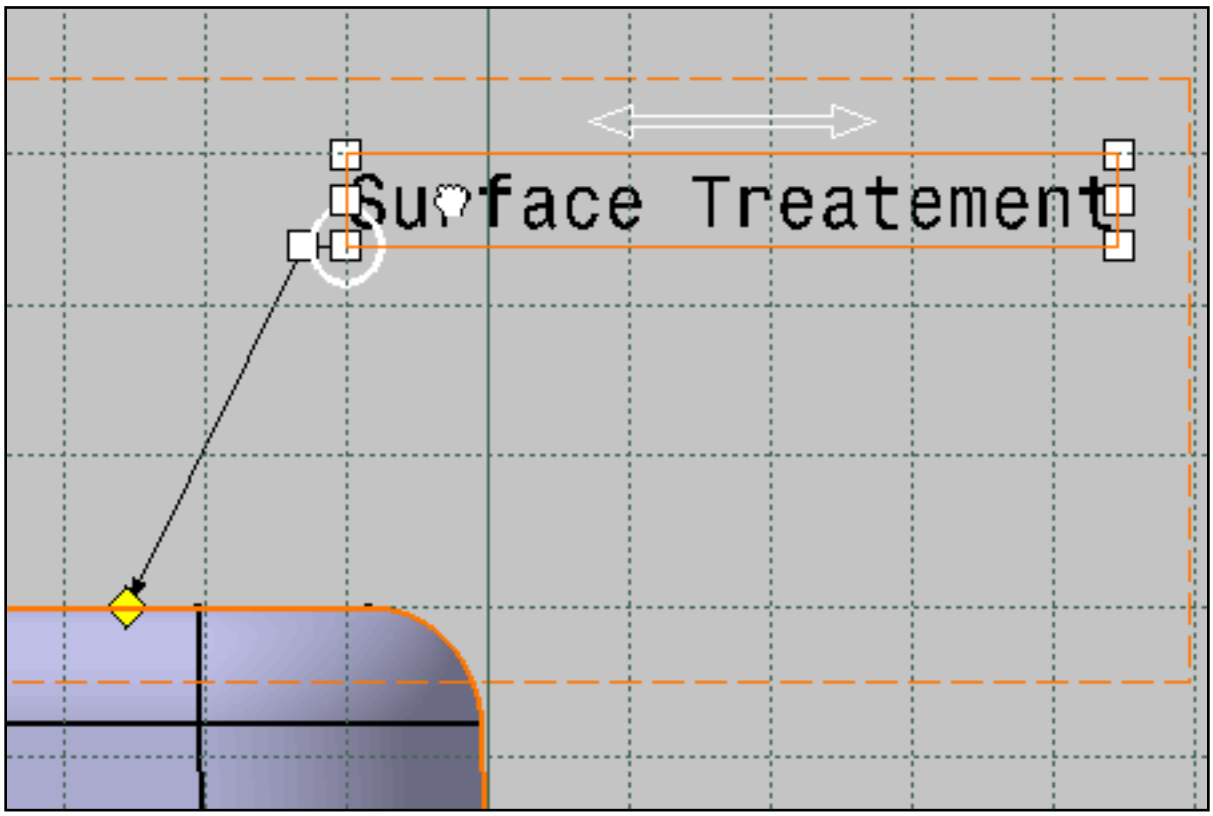


While the **Snap to Point** icon is deactivated , the annotation may be moved without any connection to the grid.

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.

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Managing Annotation Leaders

Adding Leaders and Using Breakpoint
Editing the Shape of an End Manipulator
Moving the End Manipulator of a Leader
Adding the All Around Symbol
Setting Perpendicular a Leader
Adding an Interruption Leader

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Adding Leaders and Using Breakpoints

This task shows you how to add:

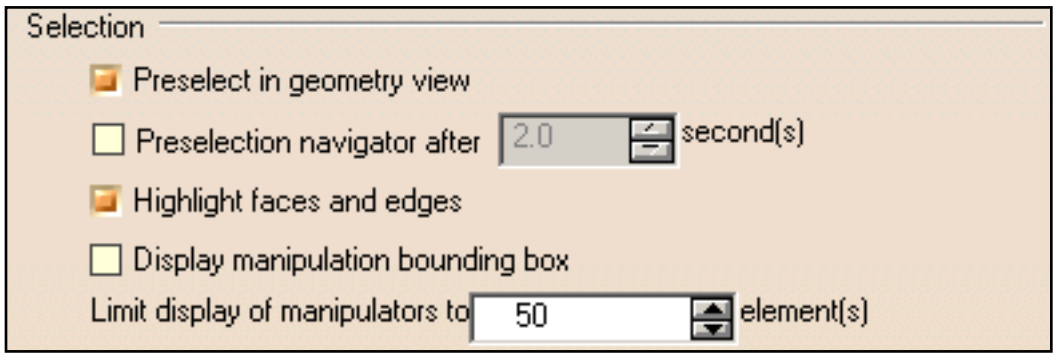
A leader to an annotation.

A breakpoint on the created leader.

A leader from the created breakpoint.

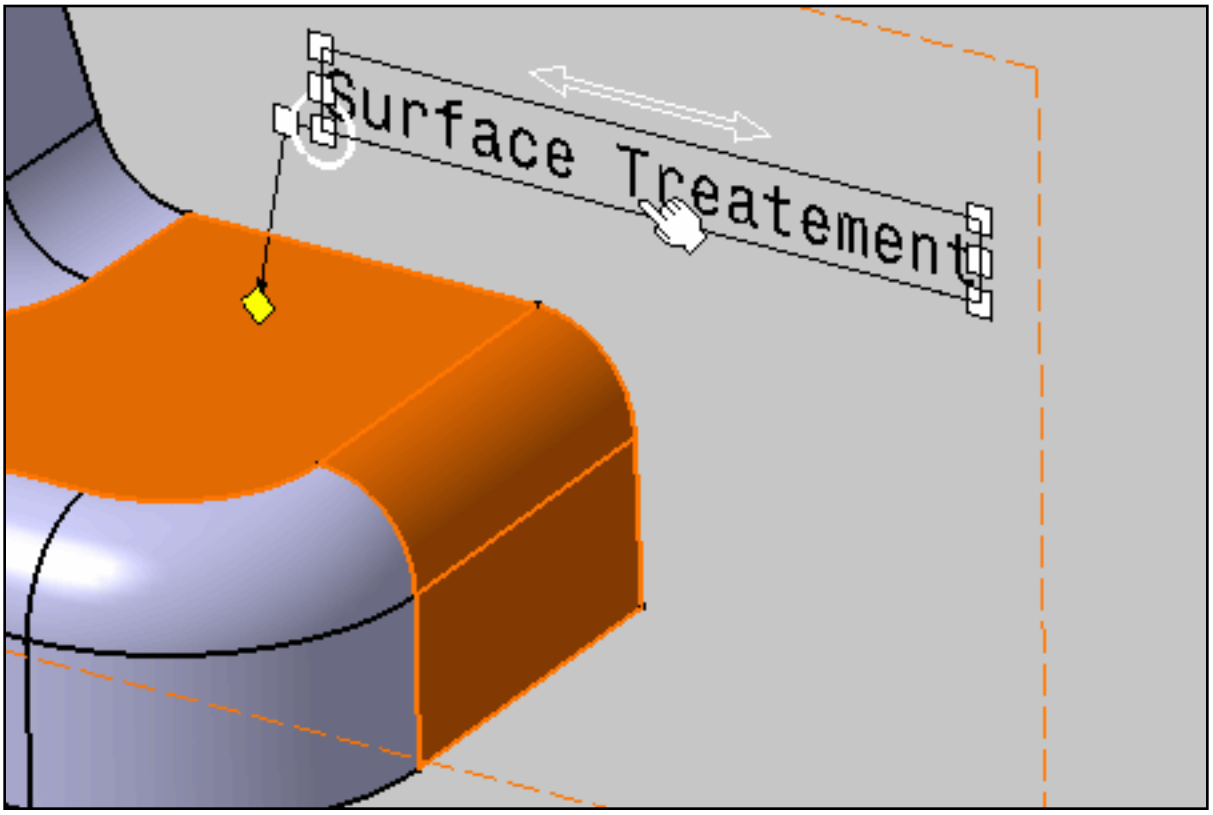
You can add a leader to geometrical elements associated with the annotation only.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

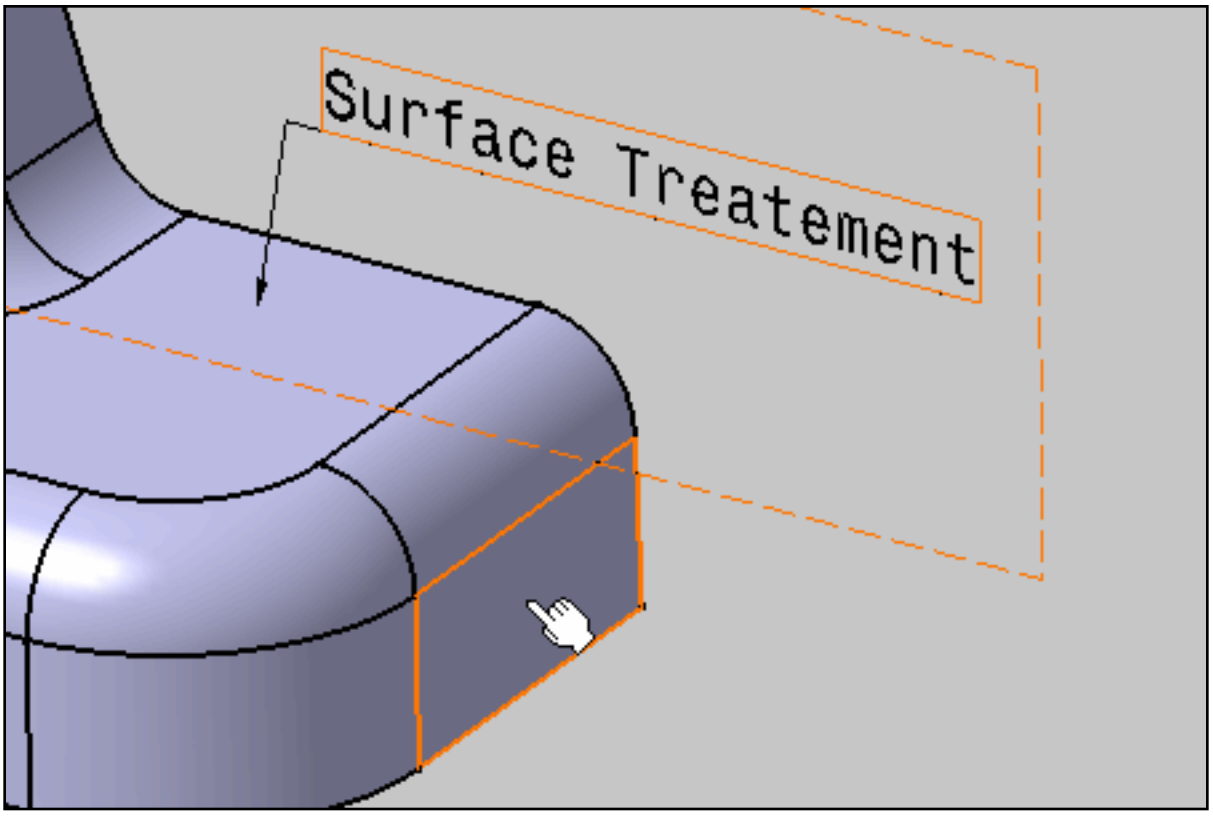


Open the Tolerancing_Annotations_04 CATPart document.

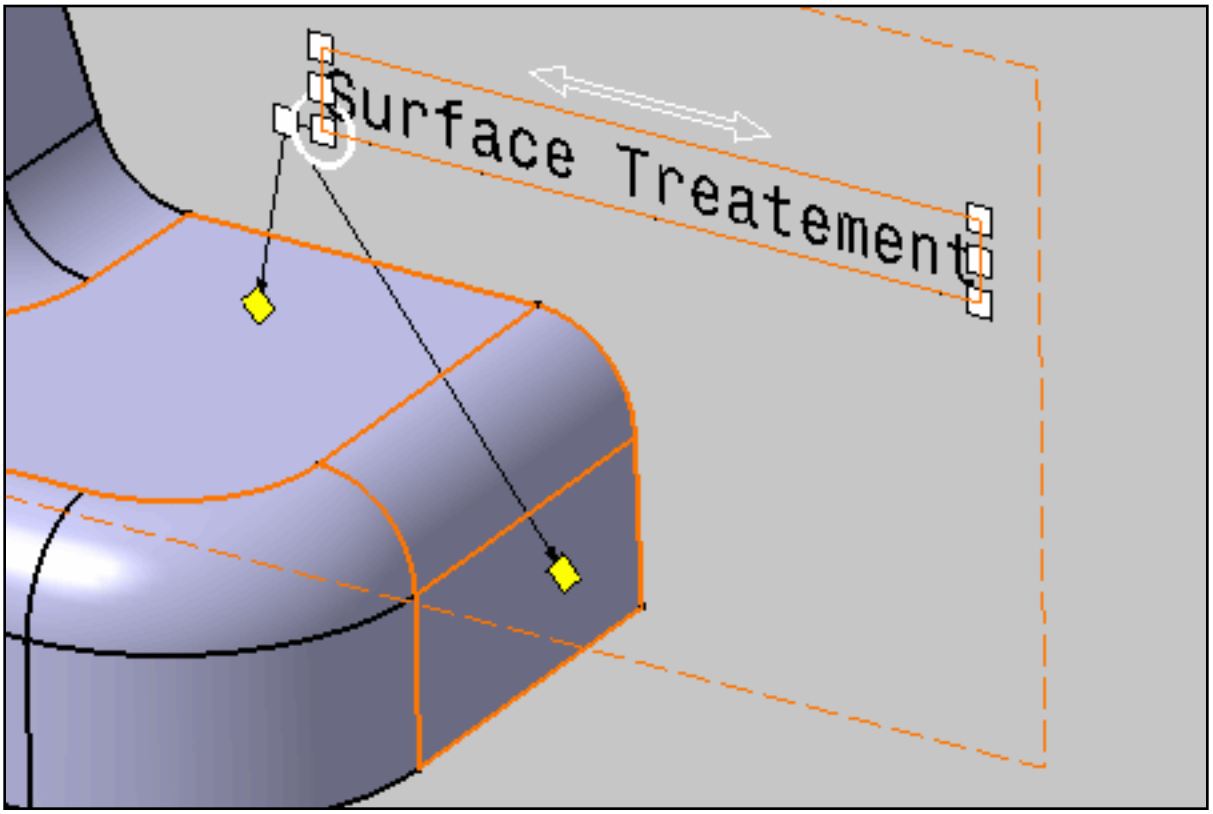
1. Right-click the annotation text and select **Add Leader** from the contextual menu.



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

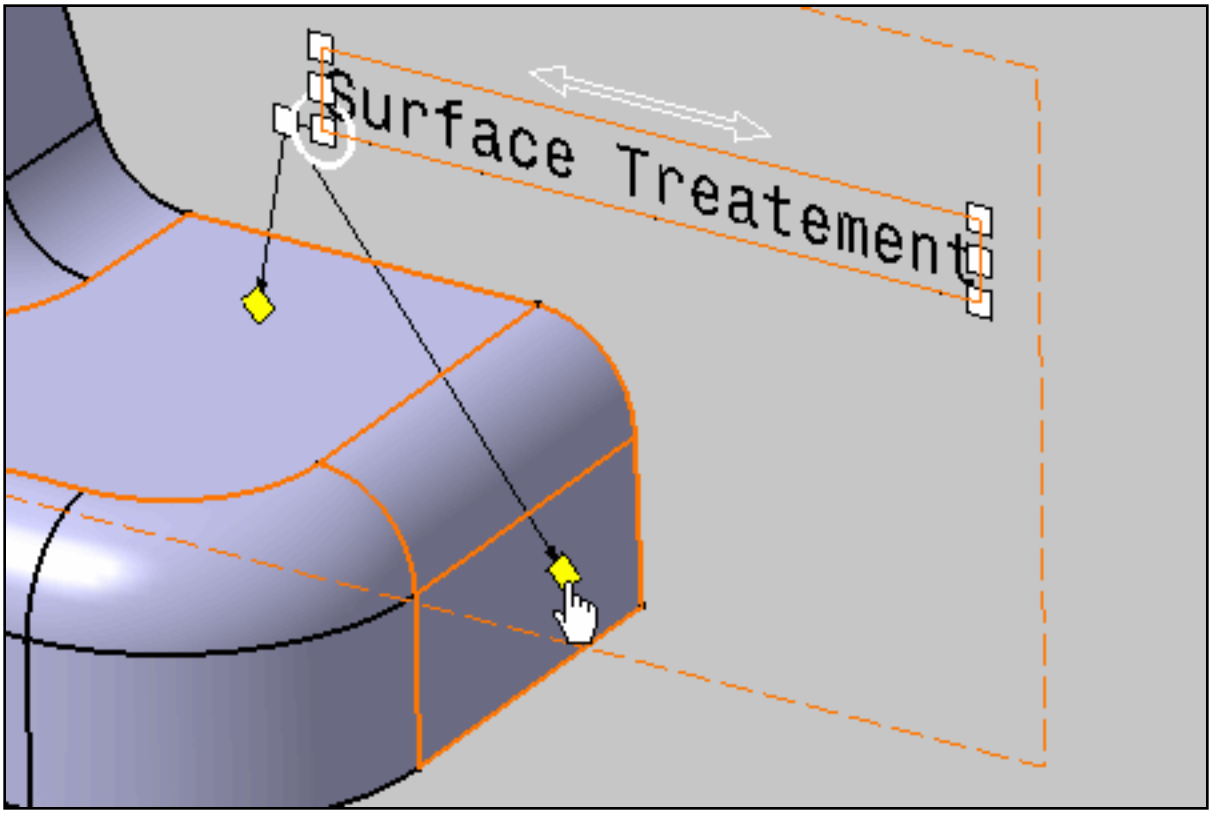


Then new leader appears.

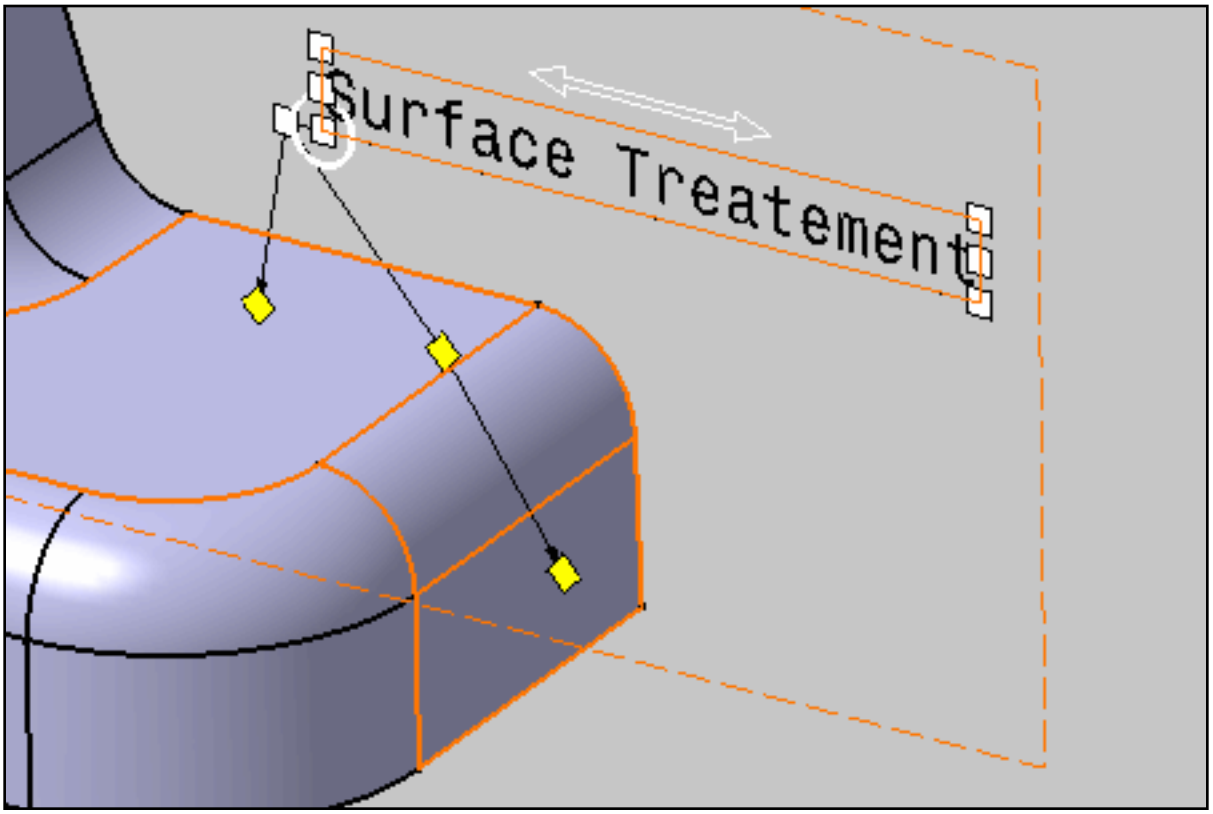


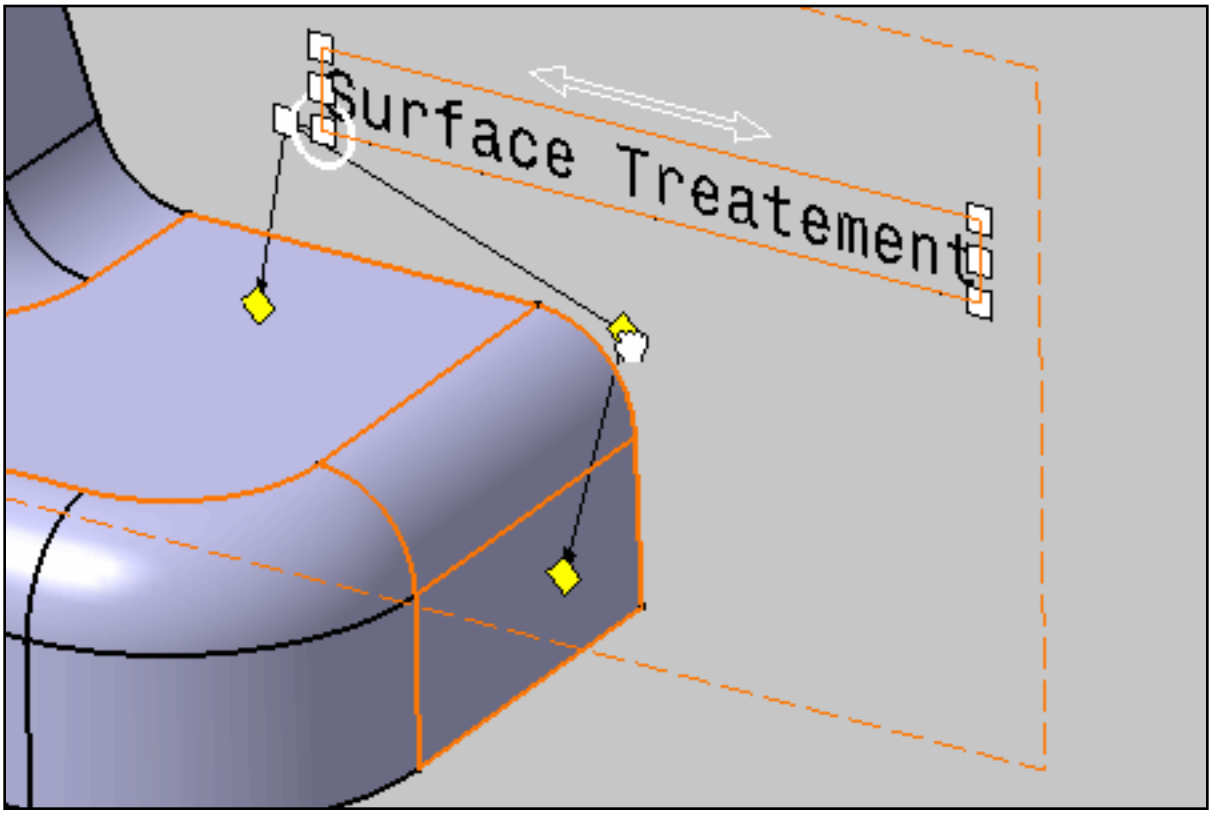
3. If needed, position the leader at the desired location by dragging it.

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.

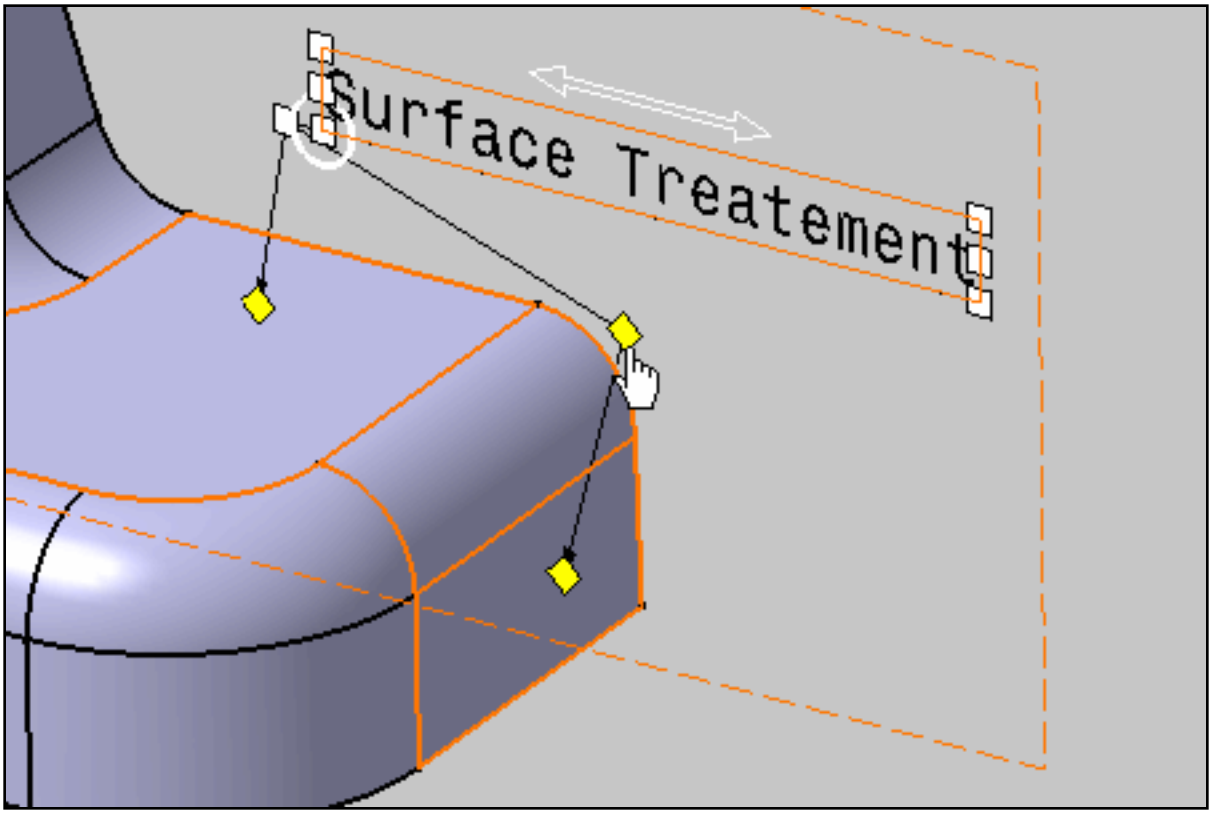




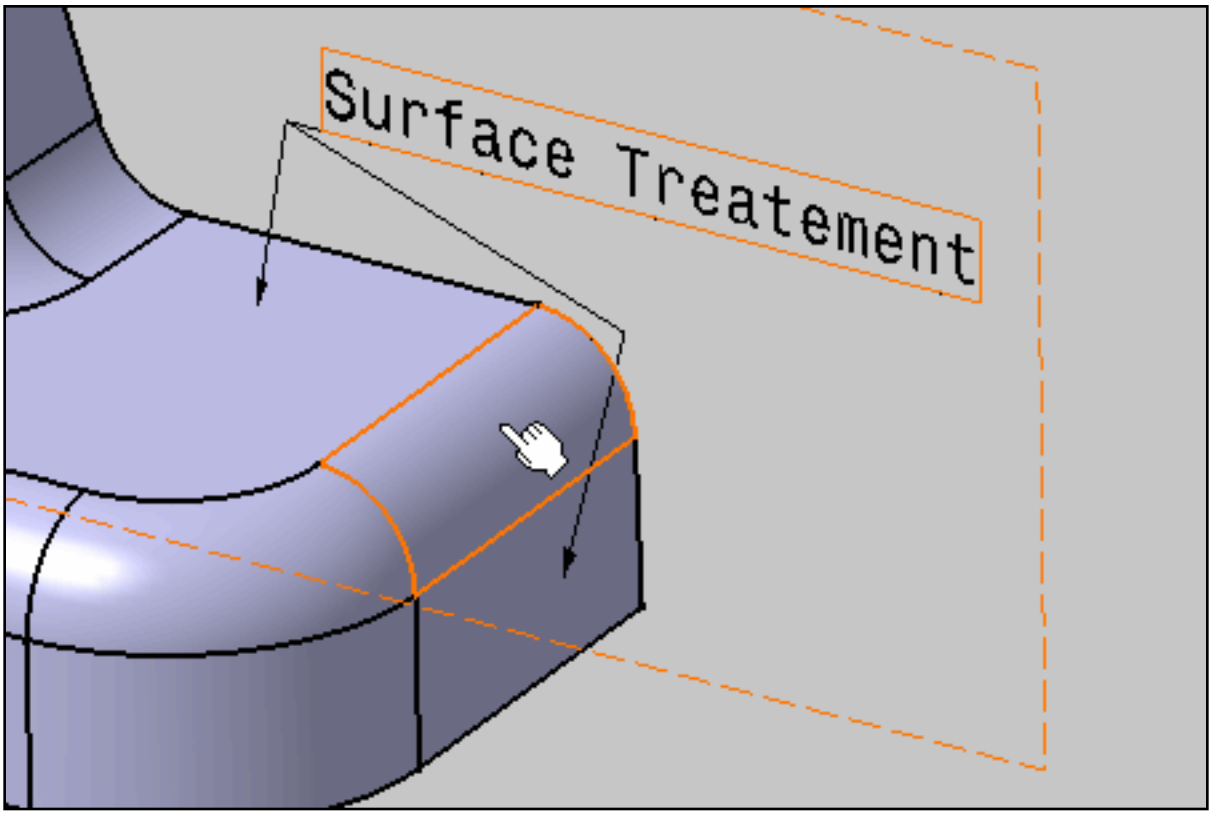
5. To add a leader from the breakpoint, select the breakpoint and select the **Add an Extremity** contextual command.



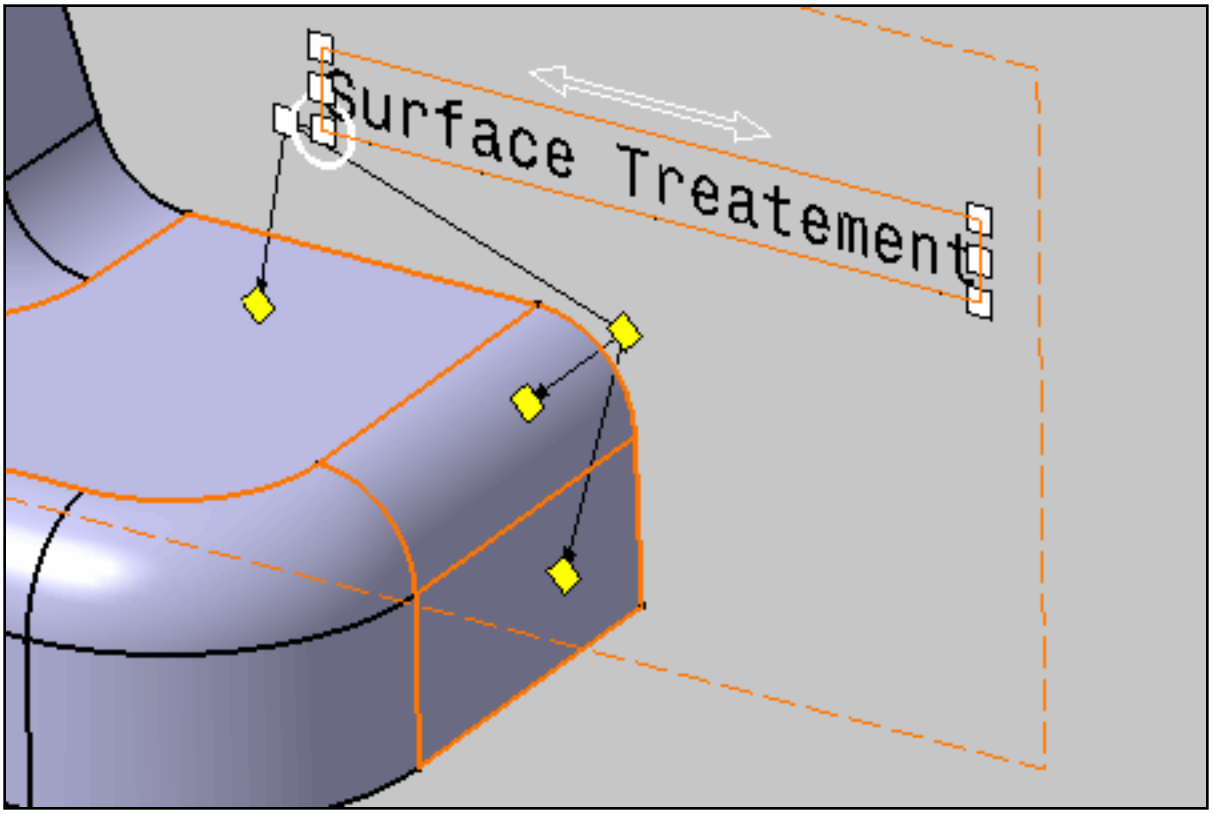
This command is only available for text and flag note annotations.



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



The leader appears.





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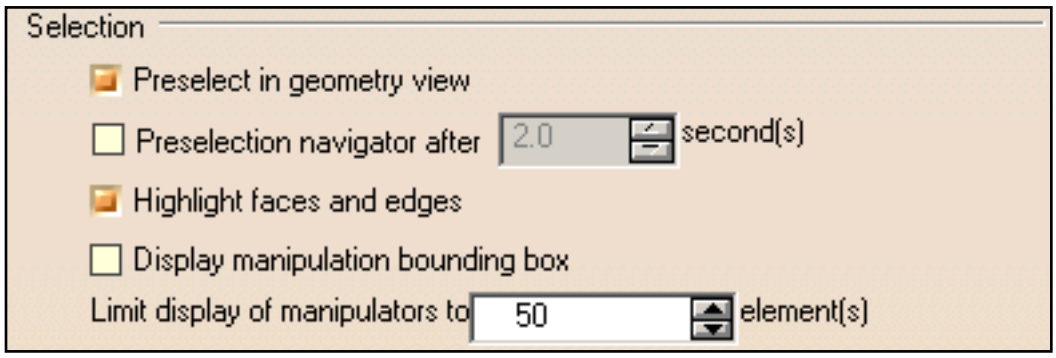
Editing the Shape of an End Manipulator

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

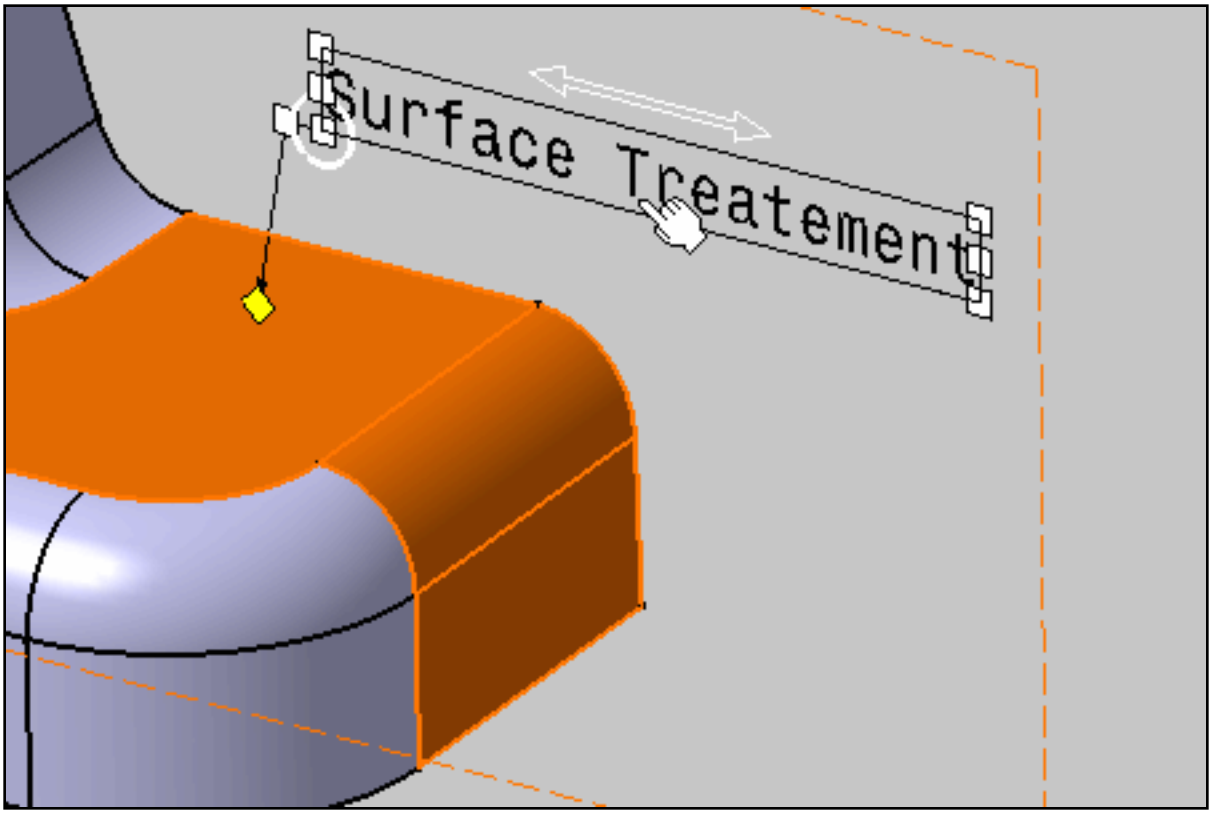
You can edit the shape of end manipulators only.

Open the Tolerancing_Annotations_04 CATPart document.

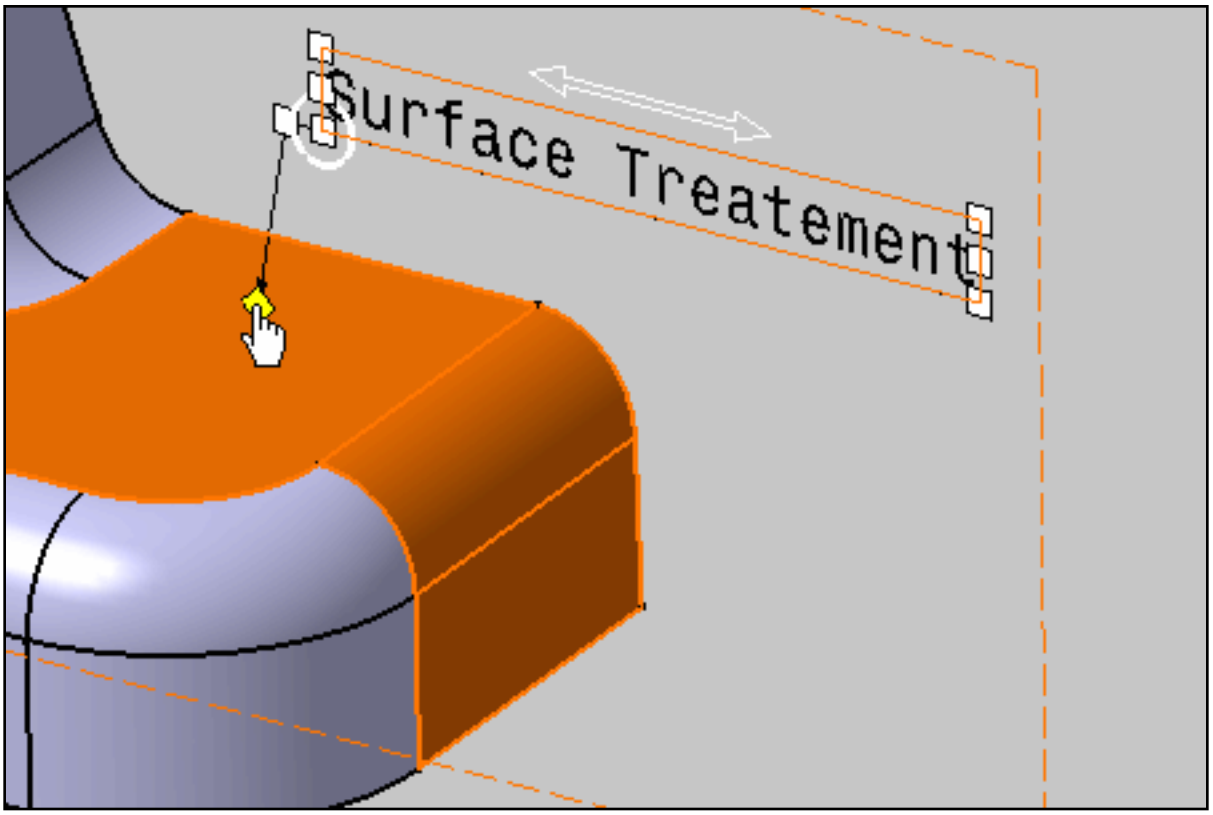
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



1. Click the annotation text.

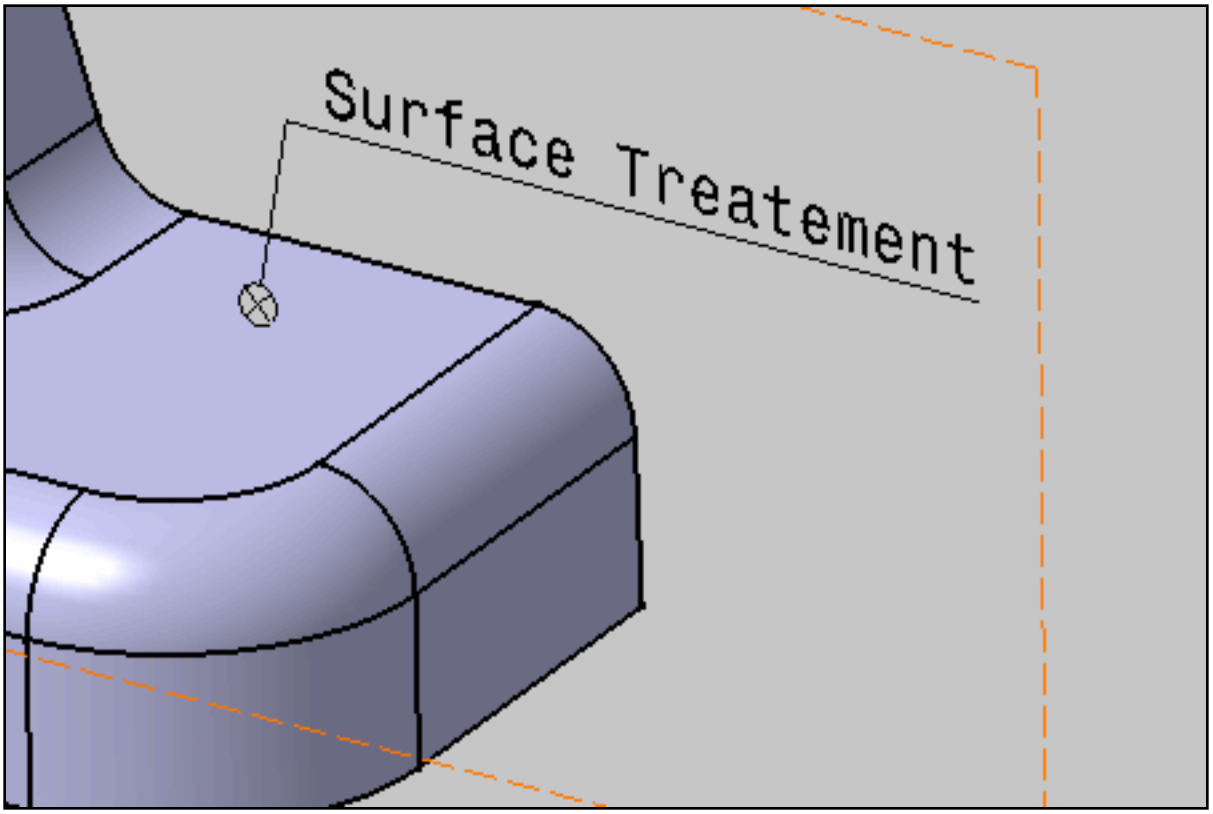


2. Right-click the end manipulator and select **Symbol Shape** from the contextual menu.



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

You obtain this result.



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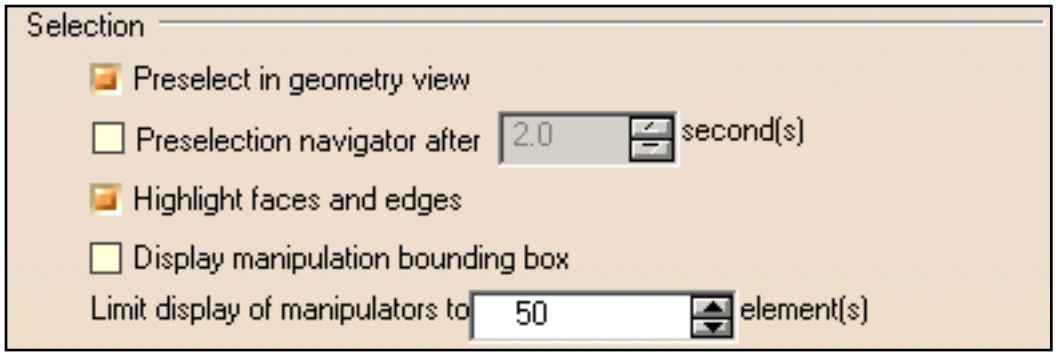
Moving the End Manipulator of a Leader

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

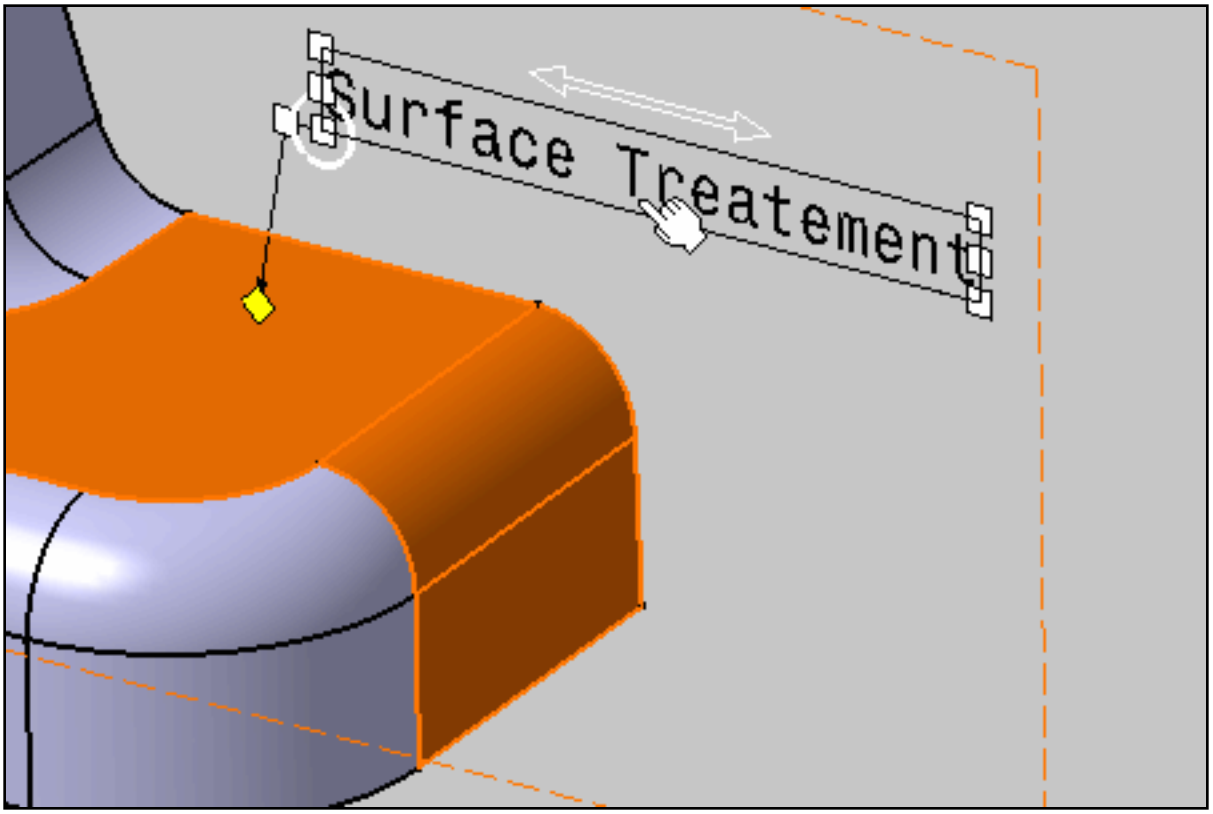
You can move a end manipulator of a leader on geometrical elements associated with the annotation only, a yellow line plots the route on them where the end manipulator is moveable.

Open the Tolerancing_Annotations_04 CATPart document.

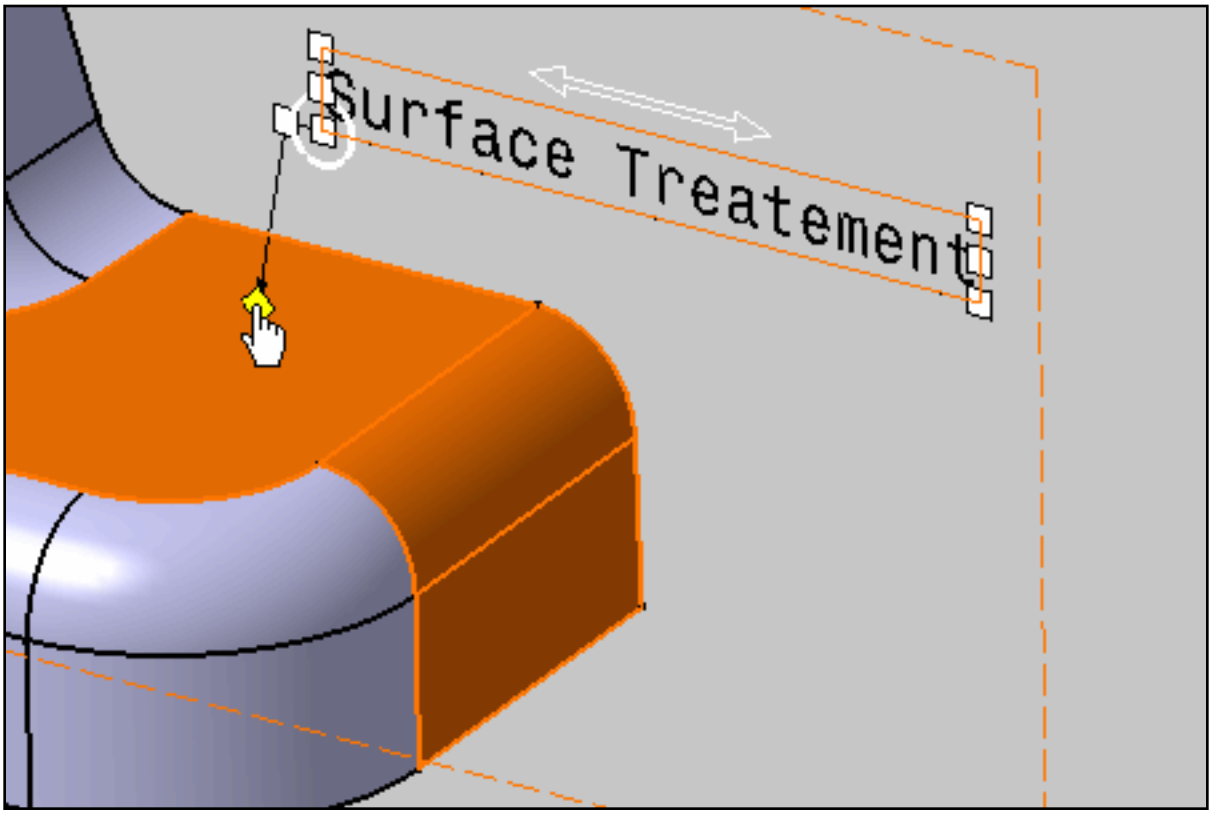
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



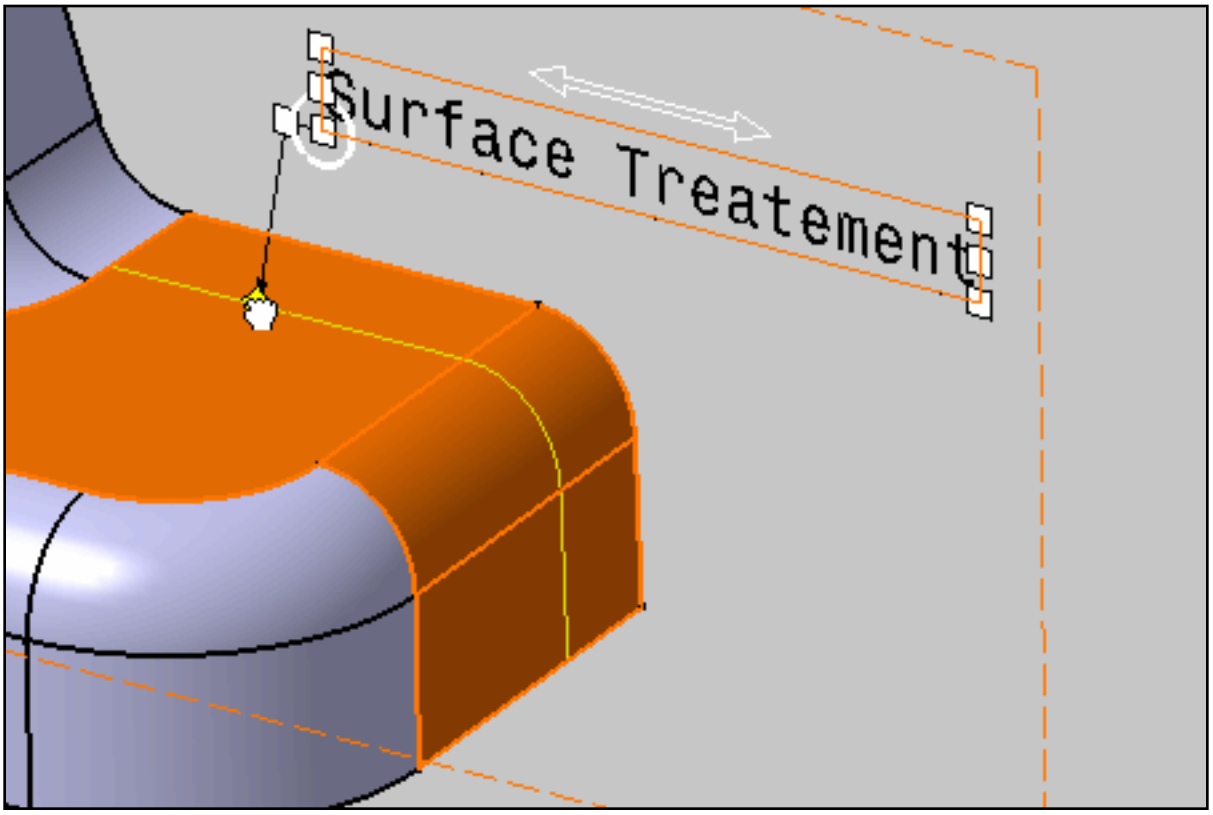
1. Click the annotation text.



2. Select the end manipulator to be moved.

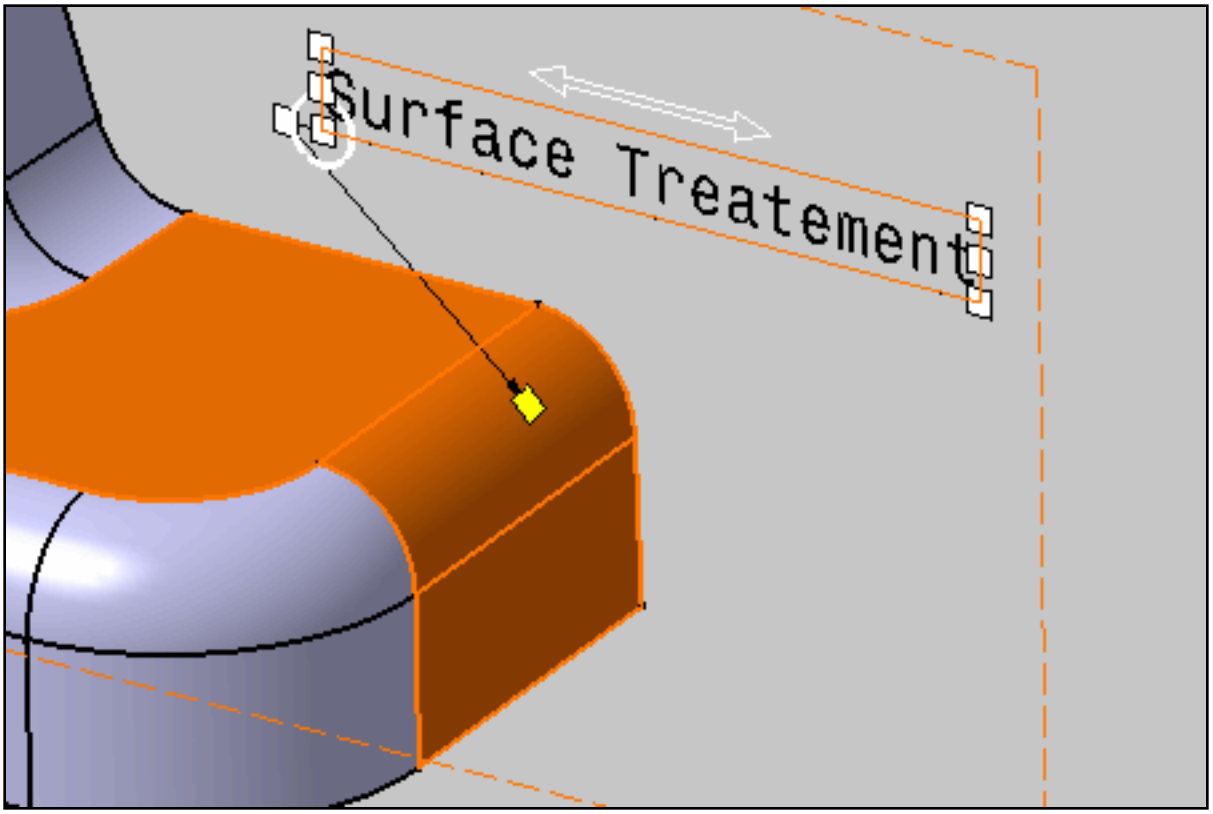


3. Drag the end manipulator along the yellow line.



4. Release the end manipulator.

You obtain this result.





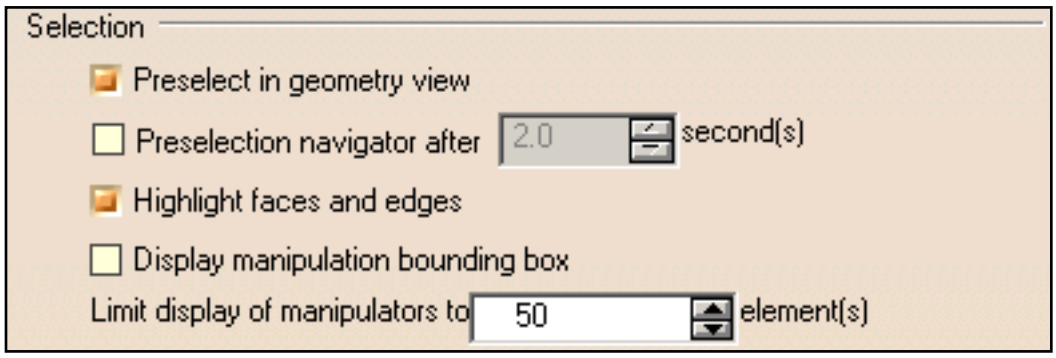
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Adding the All Around Symbol

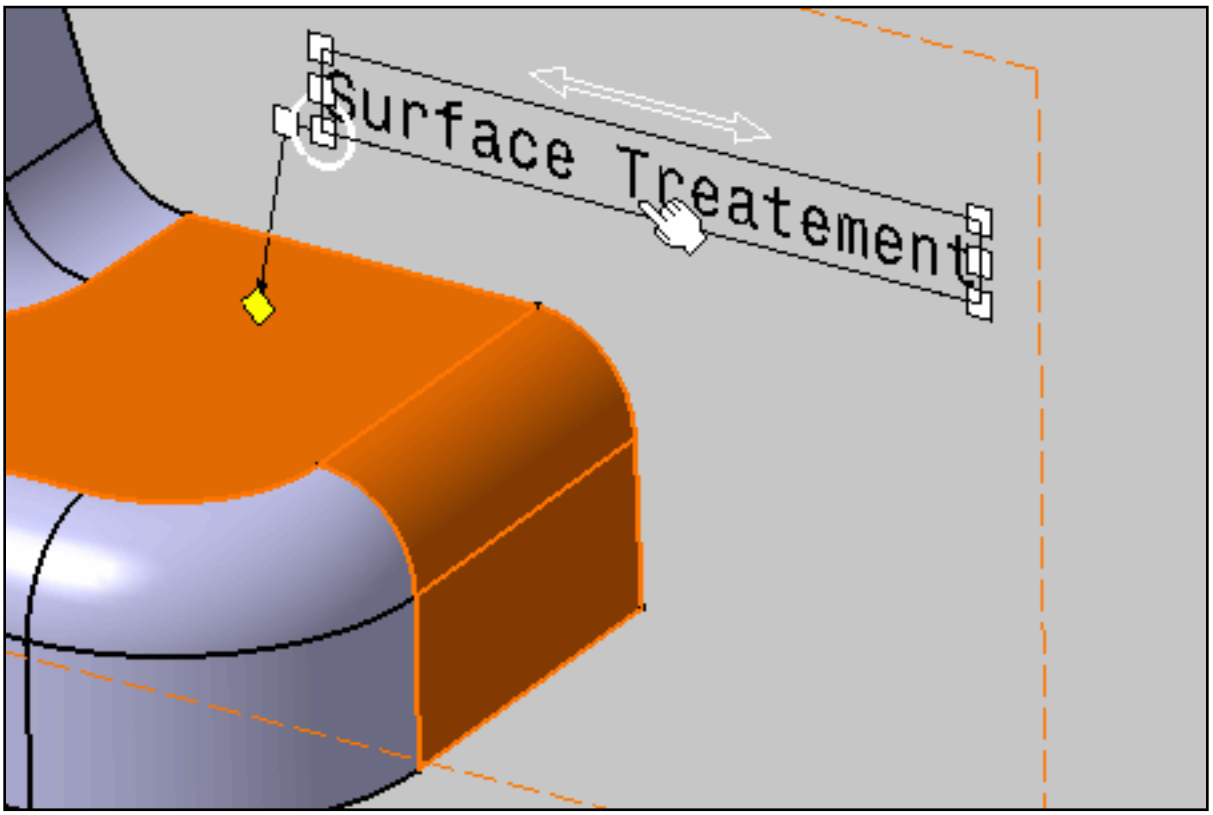
This task shows you how to add the All Around symbol to an annotation.

Open the Tolerancing_Annotations_04 CATPart document.

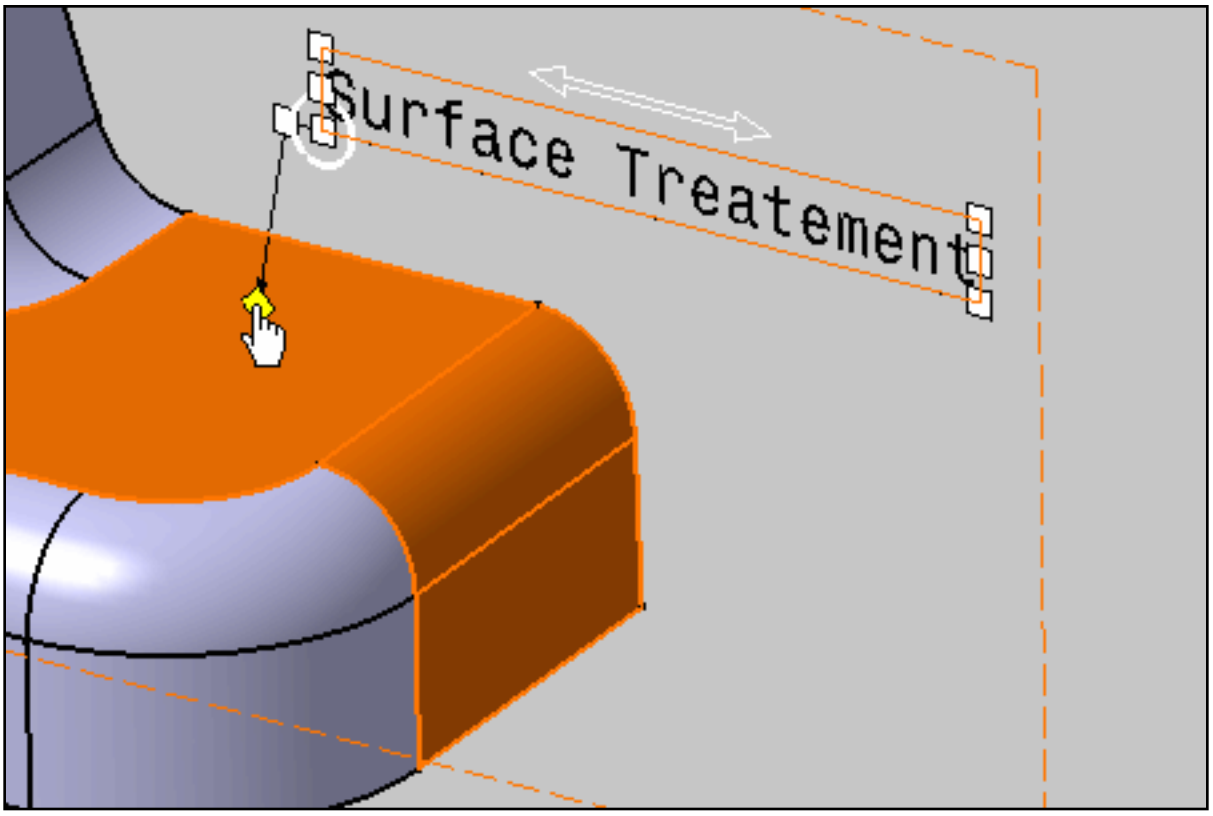
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



1. Click the annotation text.

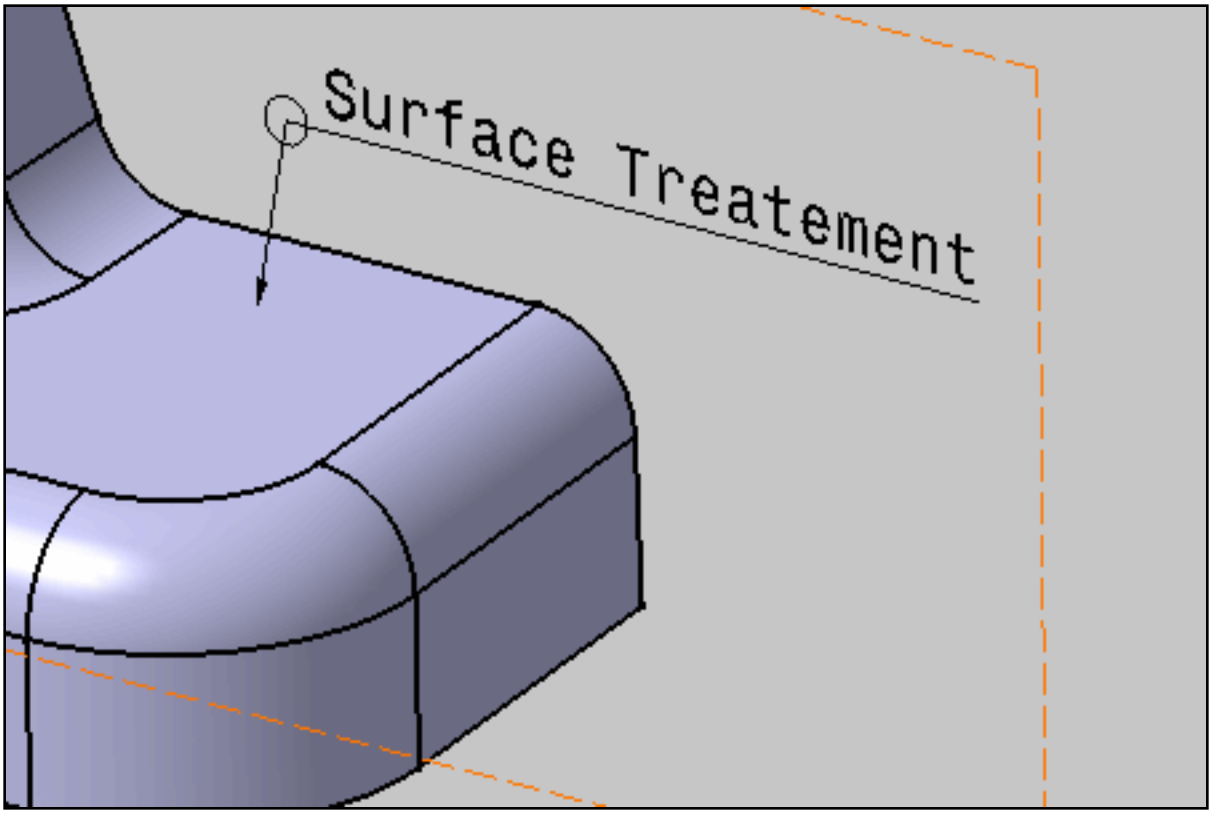


2. Right-click any manipulator and check **All Around** from the contextual menu.



3. Un-select the annotation.

You obtain this result.



Back Up Next

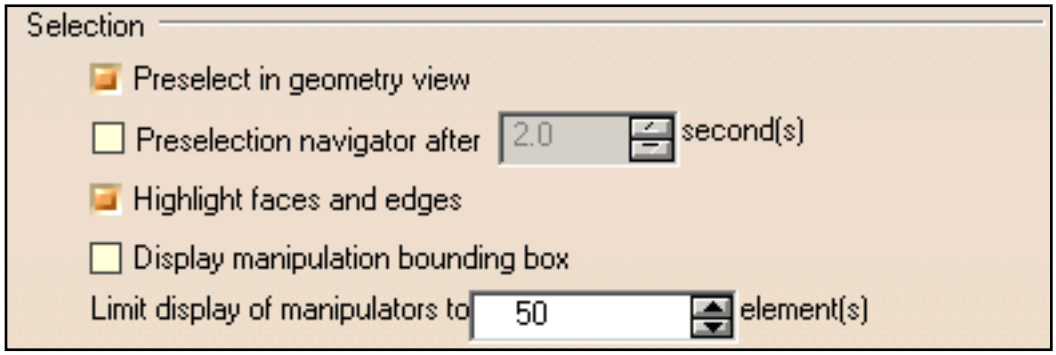
Setting Perpendicular a Leader

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

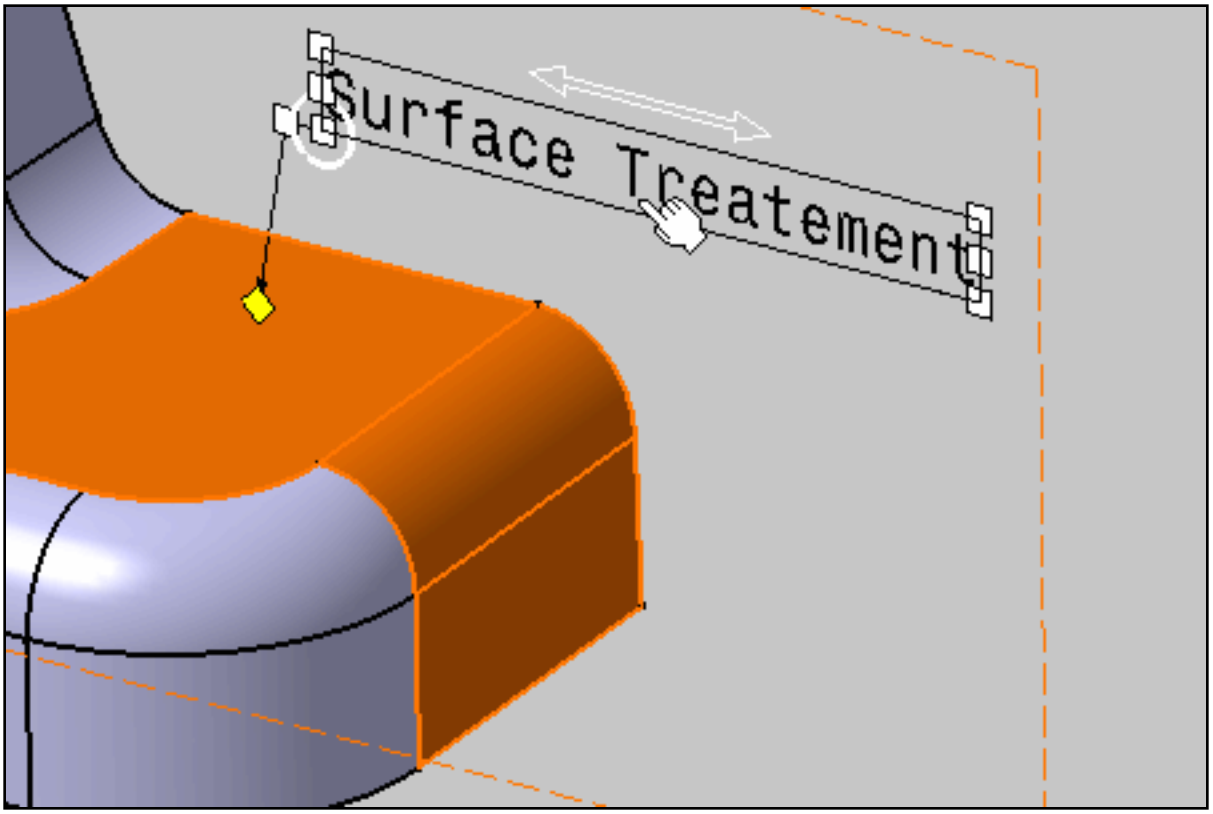
You can set the leader perpendicularity during the creation using the Leader associativity to the geometry option.

Open the Tolerancing_Annotations_04 CATPart document.

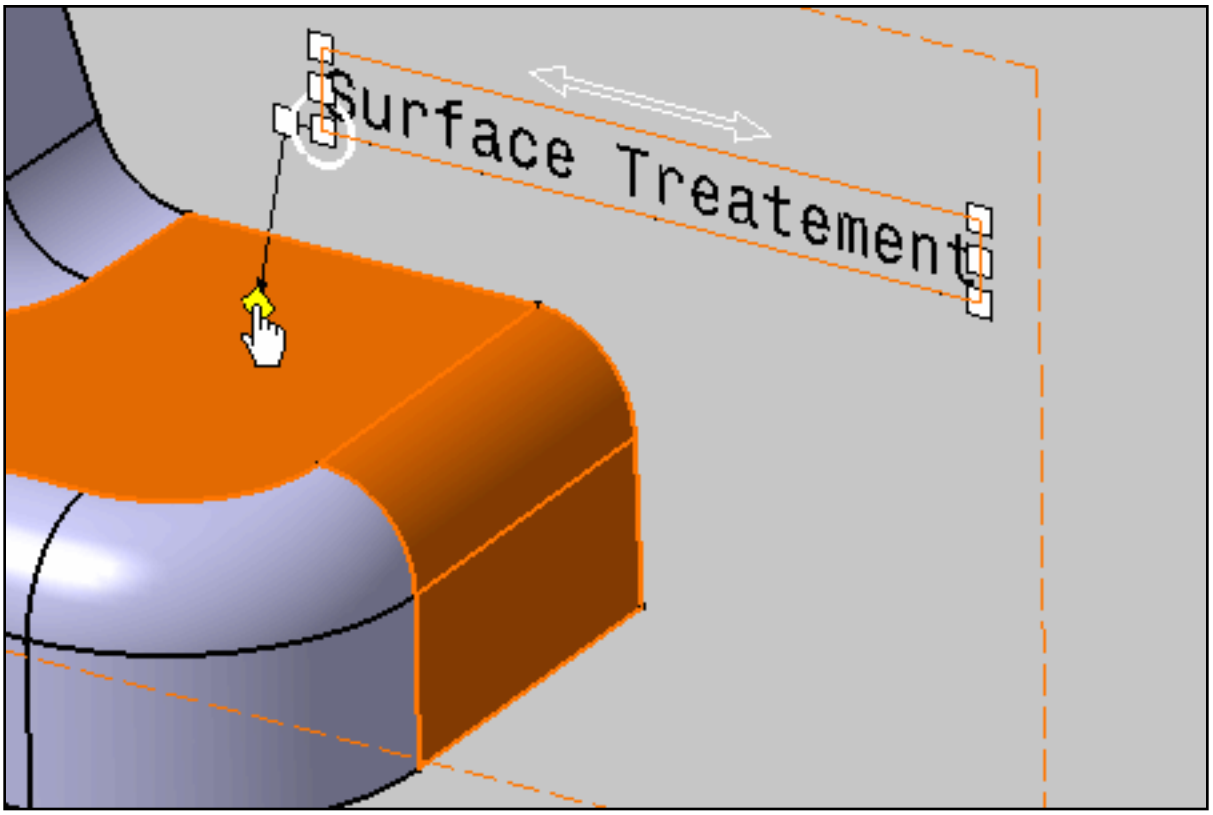
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



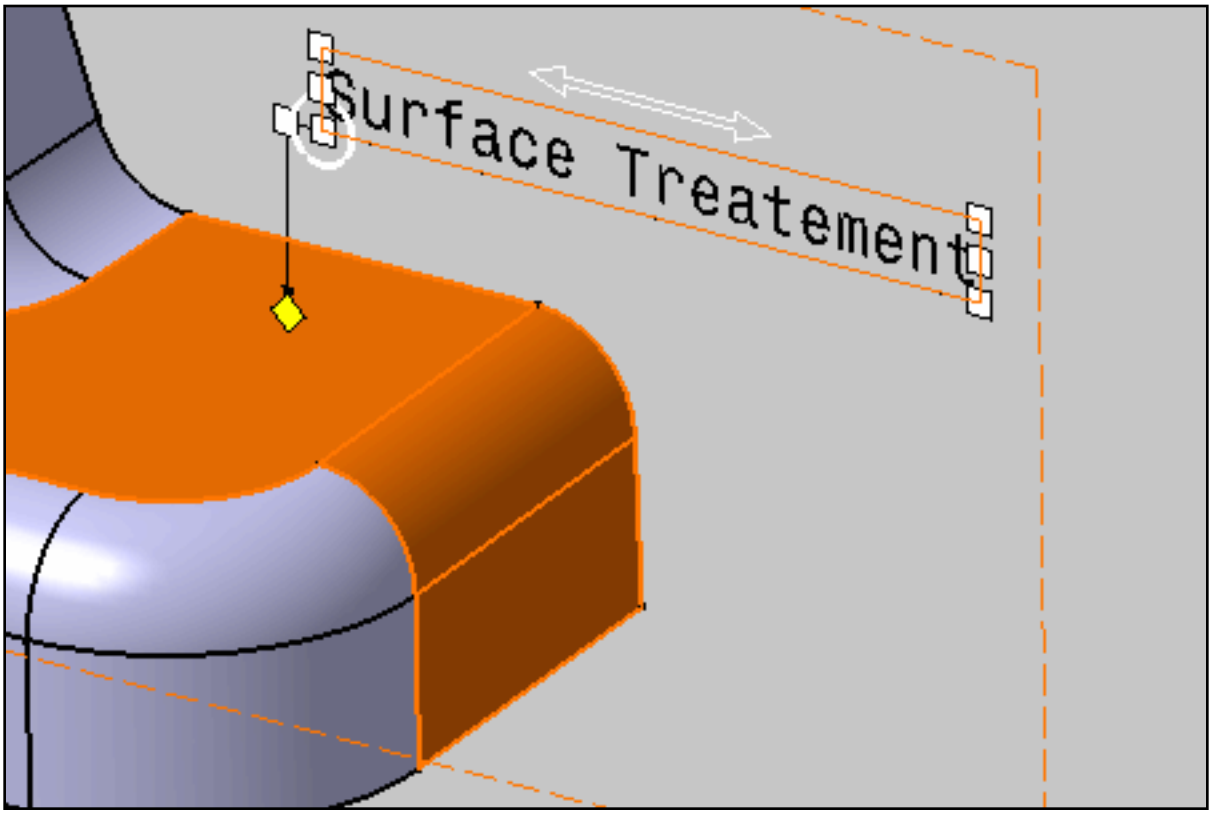
1. Click the annotation text.



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

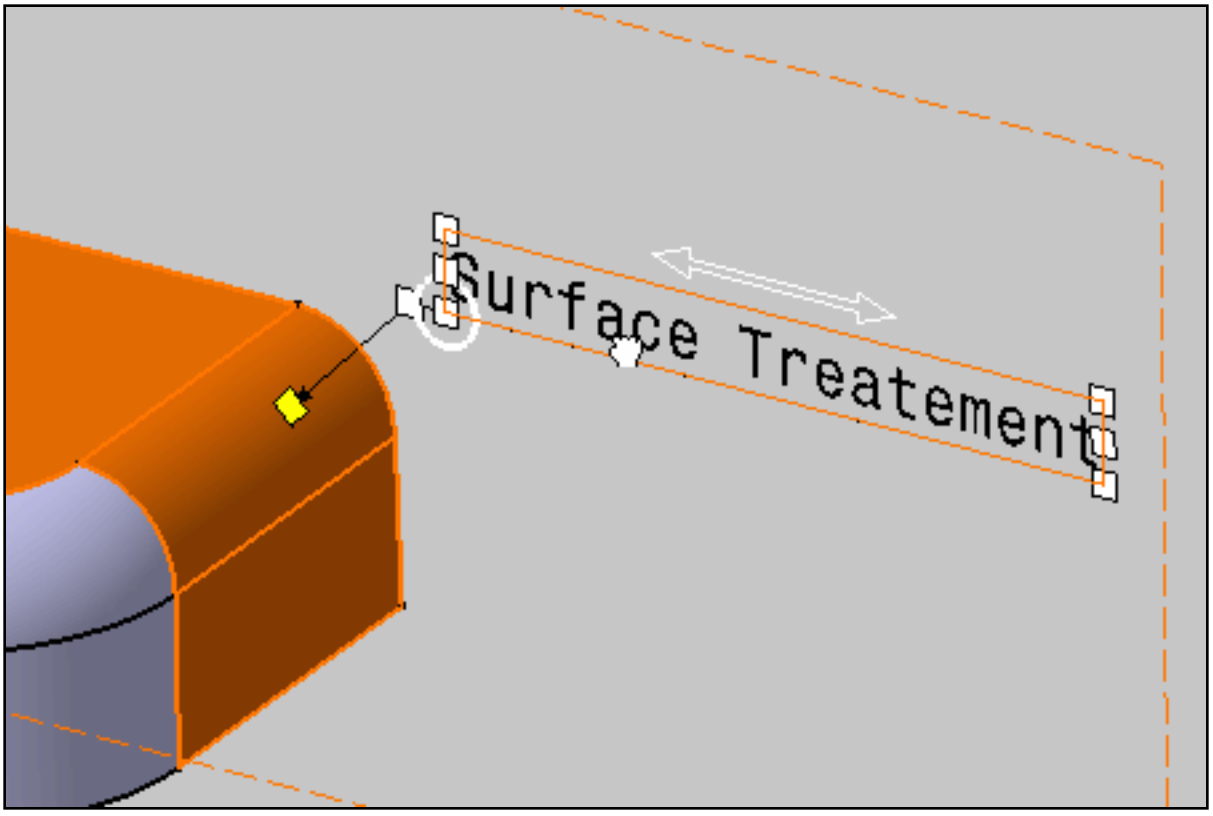


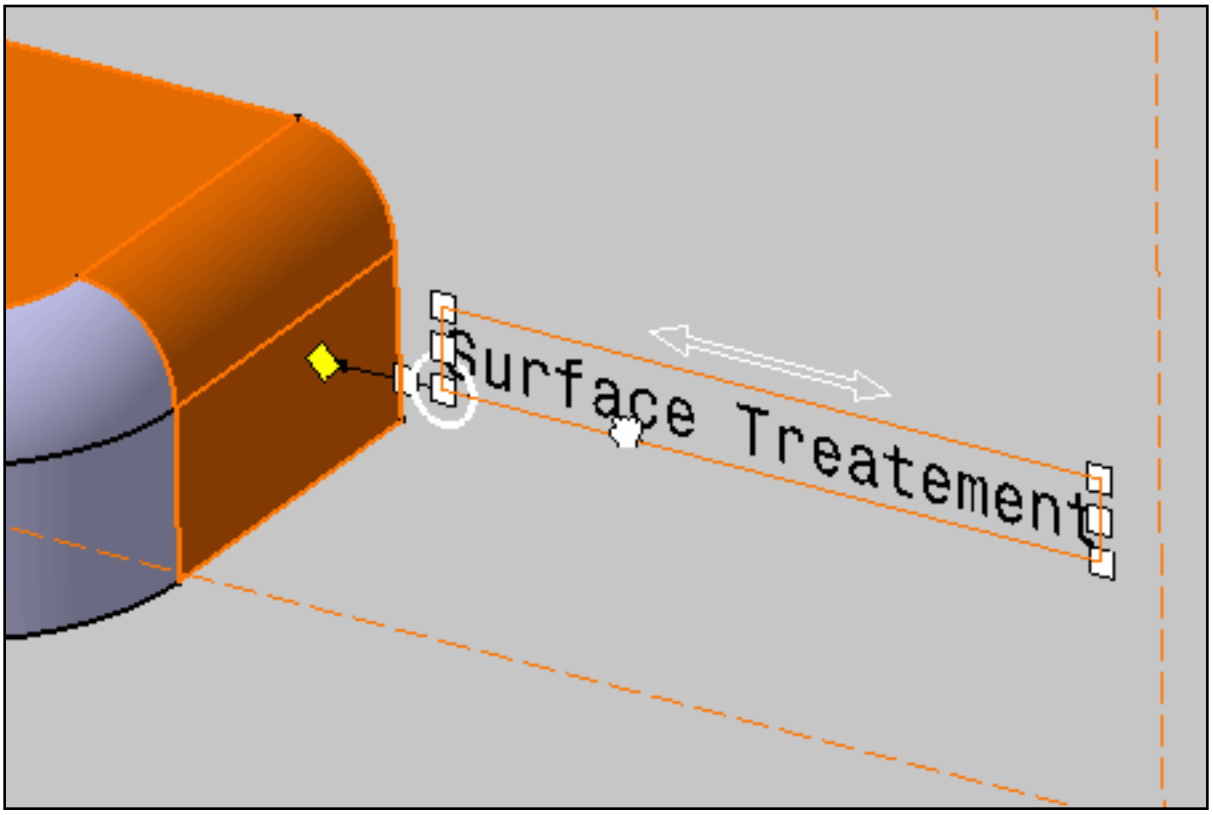
You obtain this result.



3. Drag the annotation.

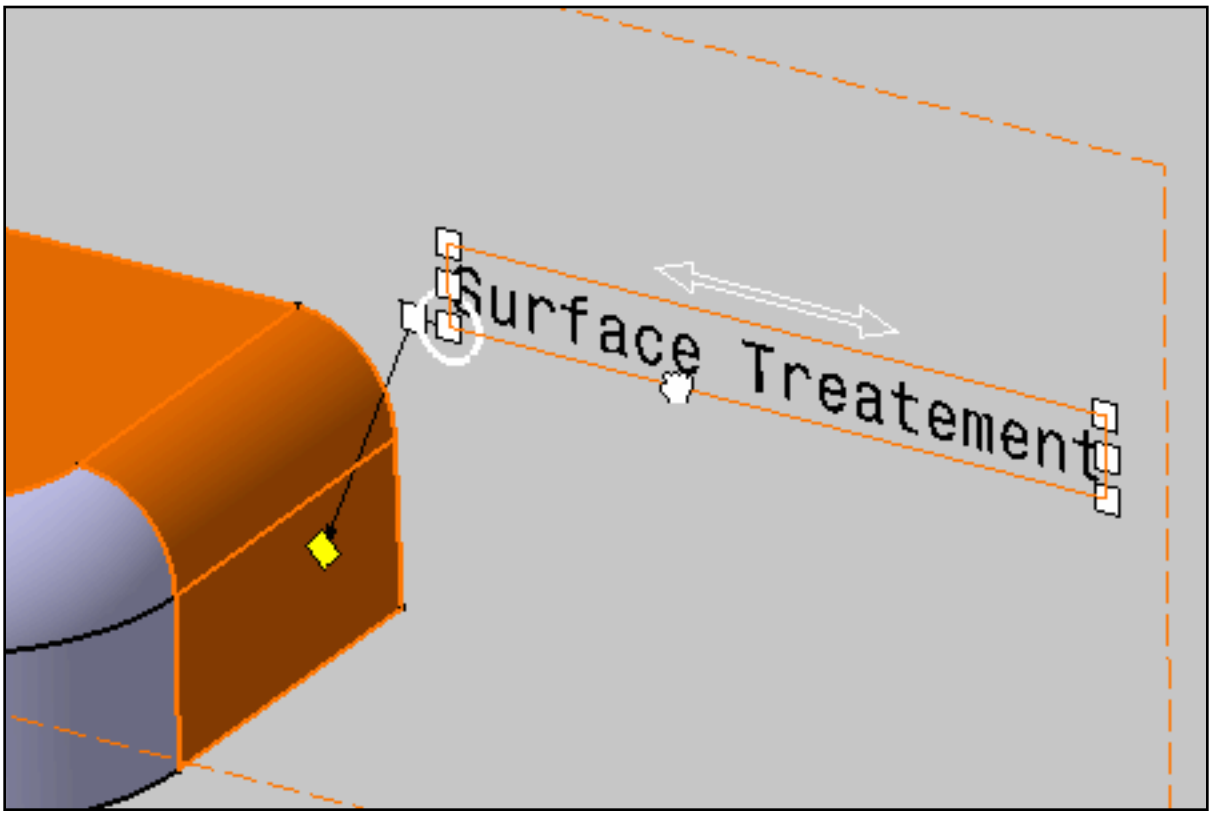
The annotation leader still perpendicular to any associated geometrical element with the annotation.





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.





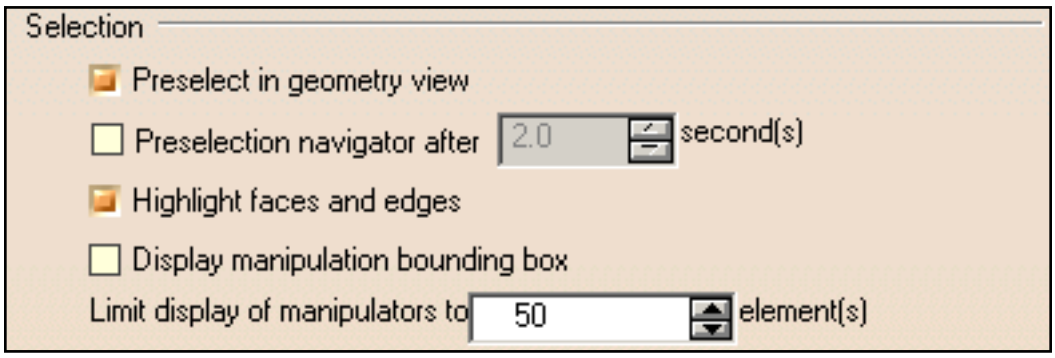
Back Up Next

Adding an Interruption Leader

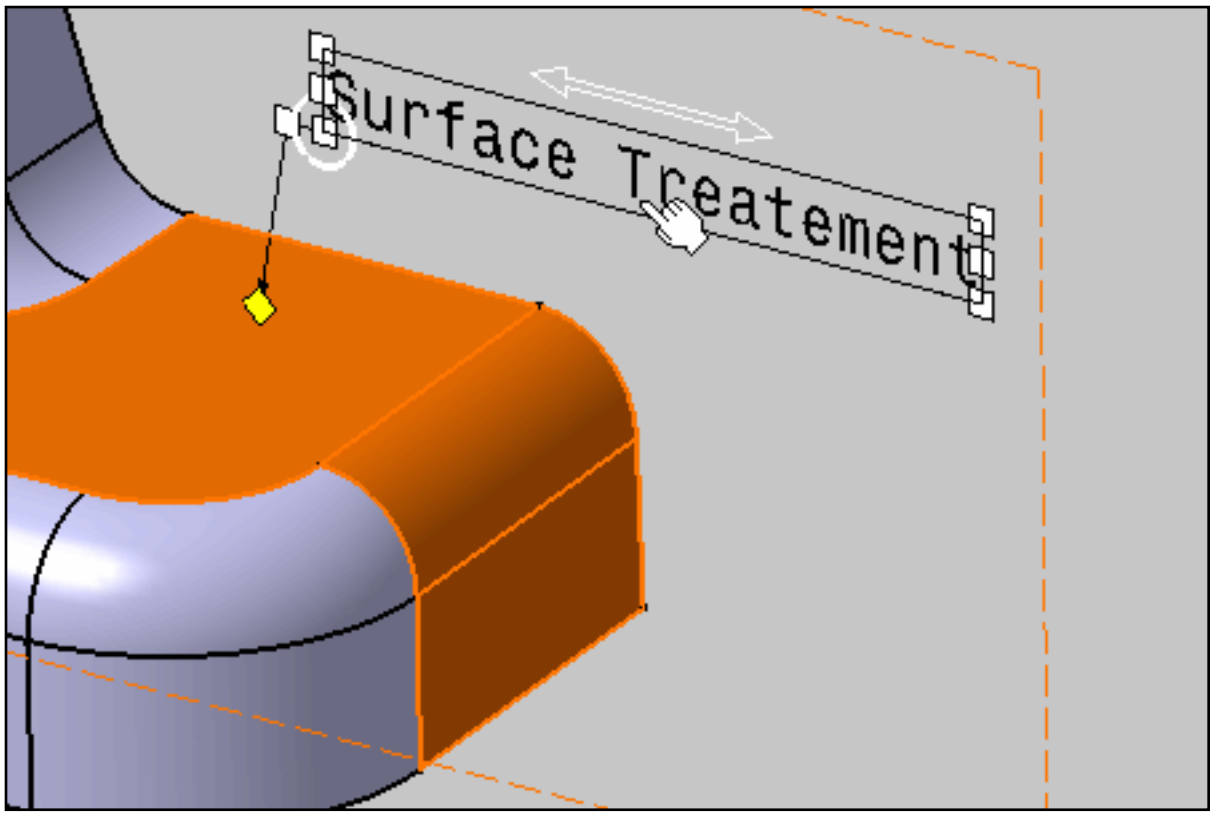
This task shows you how to add an interruption leader.

Open the Tolerancing_Annotations_04 CATPart document.

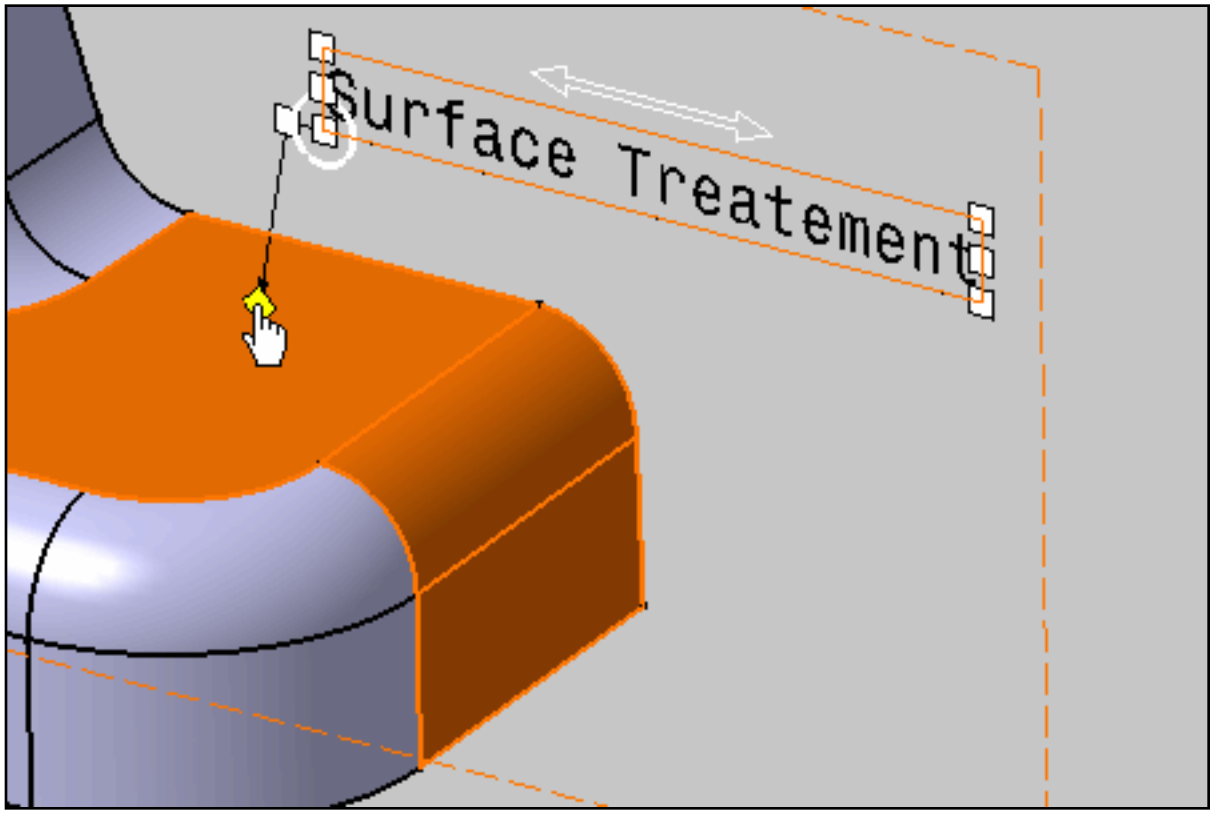
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



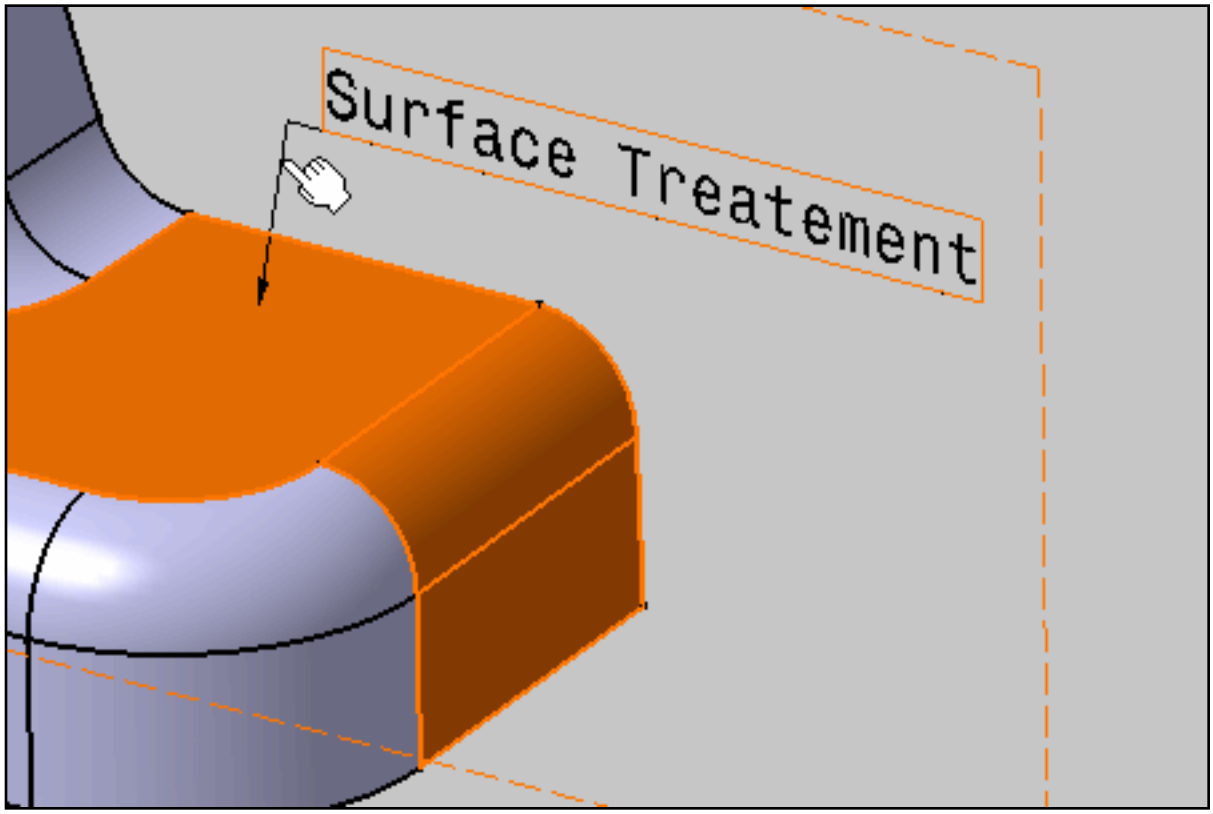
1. Click the annotation text.



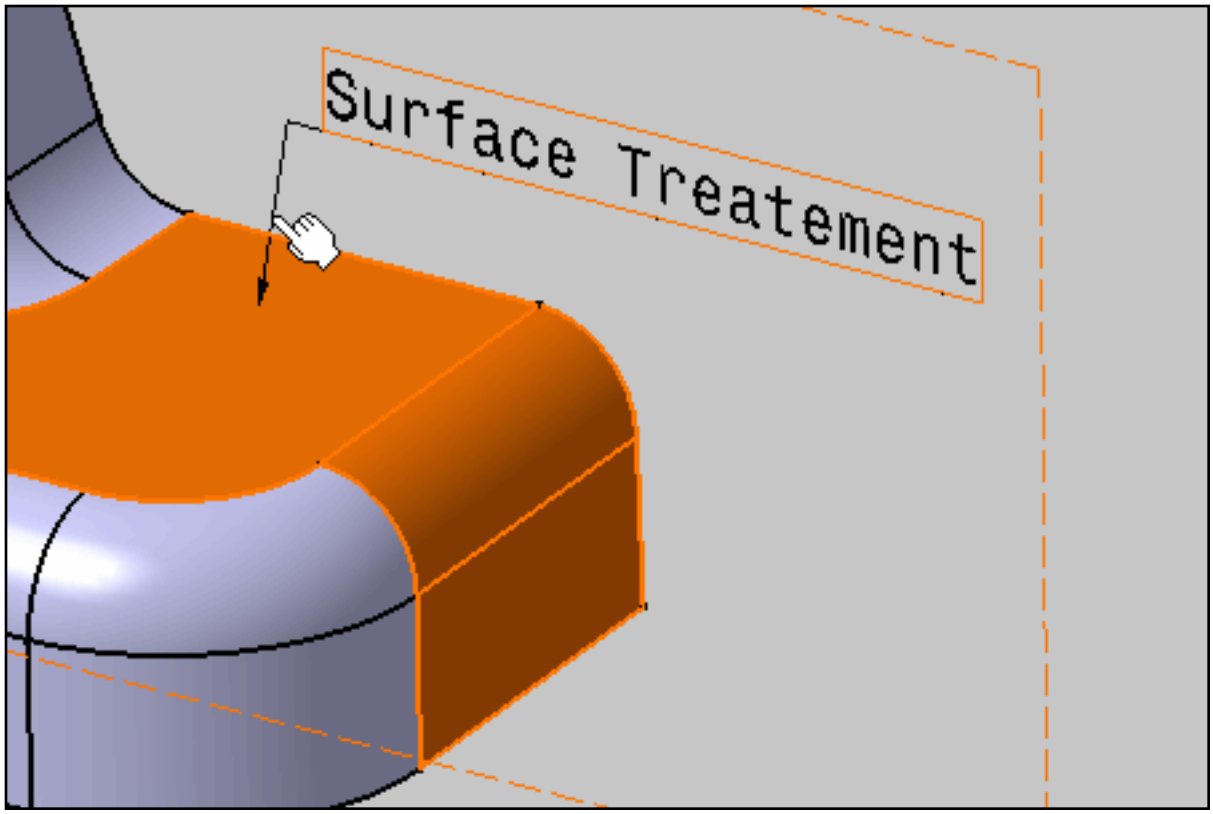
2. Right-click the end manipulator and select **Add an Interruption** from the contextual menu.



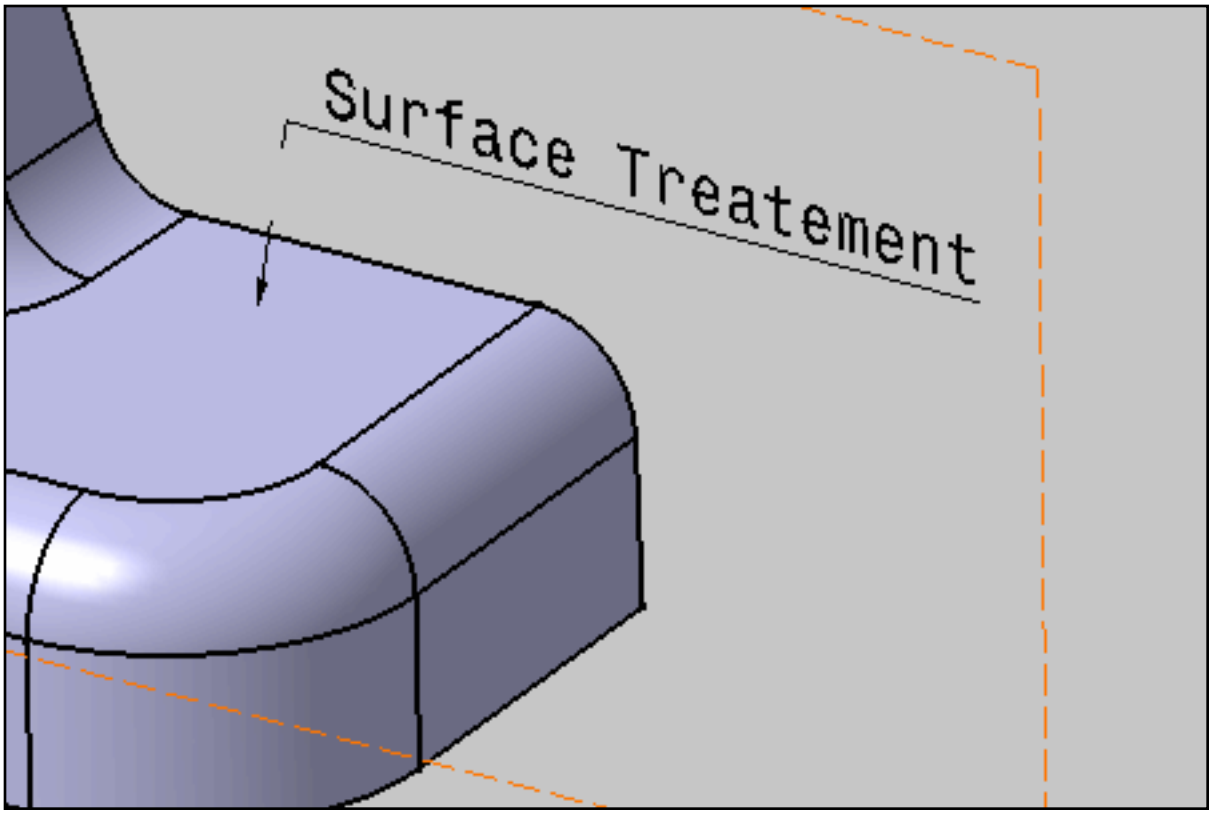
3. Indicate the start point of the interruption on the leader.



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.

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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.

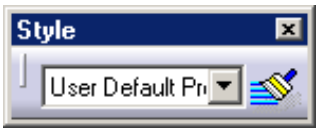
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Setting Basic Graphical Properties

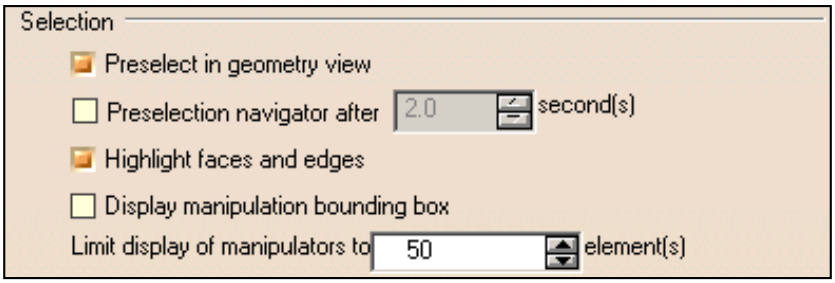
This task shows you how to set graphical properties from the **Text Properties** toolbar for a textual annotation.
Note that the operating mode described here is also valid for datum, datum targets and geometrical tolerances.

Open the Tolerancing_Annotations_04 CATPart document.

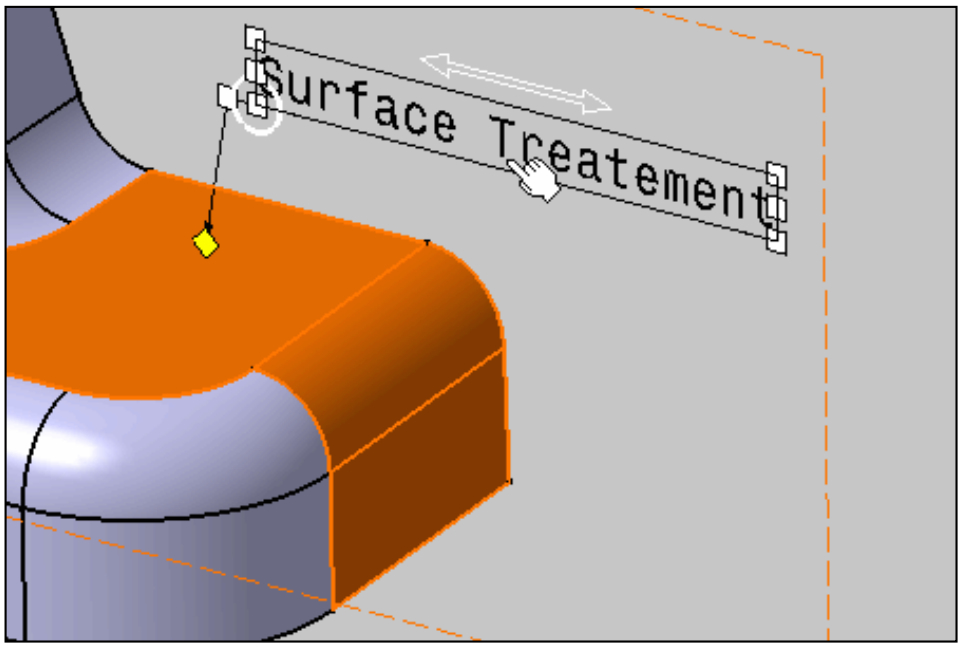
Set the **Application Default Properties** or **User Default Properties** option in the **Style** toolbar if needed.



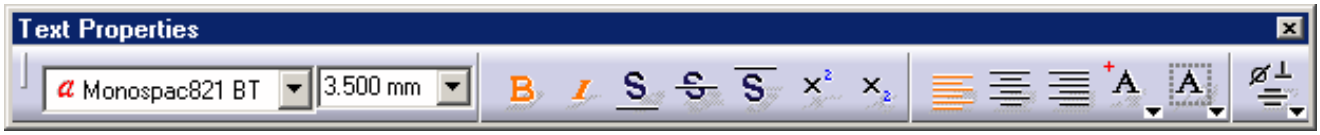
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category, then the **Navigation** tab, and check the option **Highlight faces and edges**.



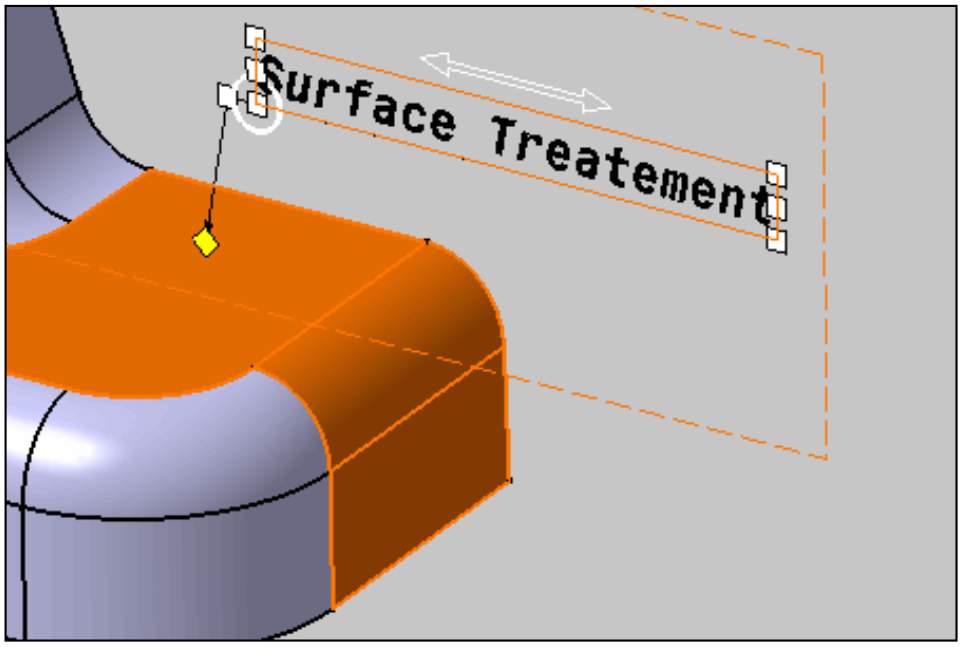
1. Select the annotation text.



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

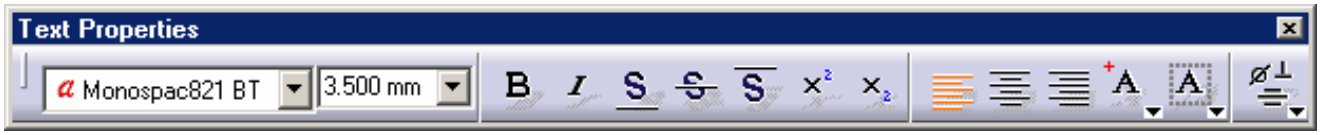


The text is modified accordingly.

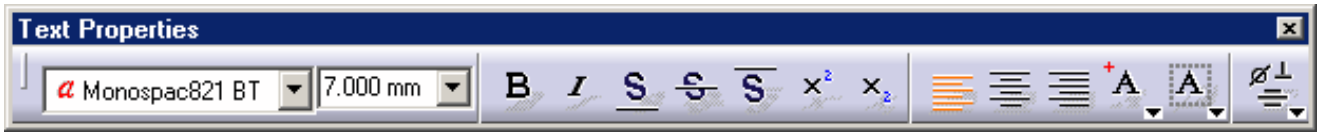


3. De-select the text.

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

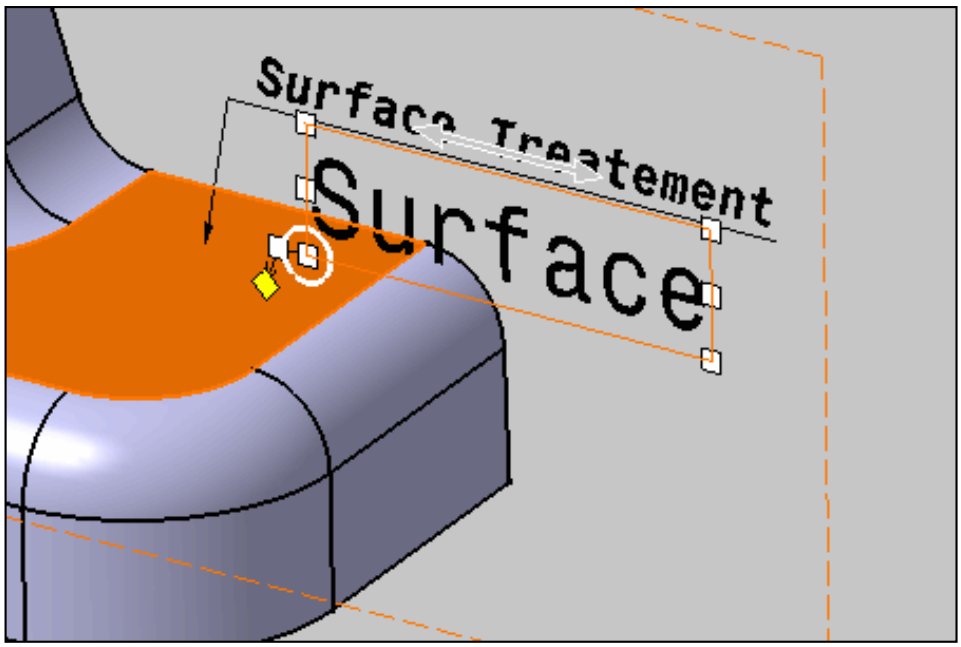


4. Set the **Font Size** to 7 mm from the **Text Properties** toolbar.



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.

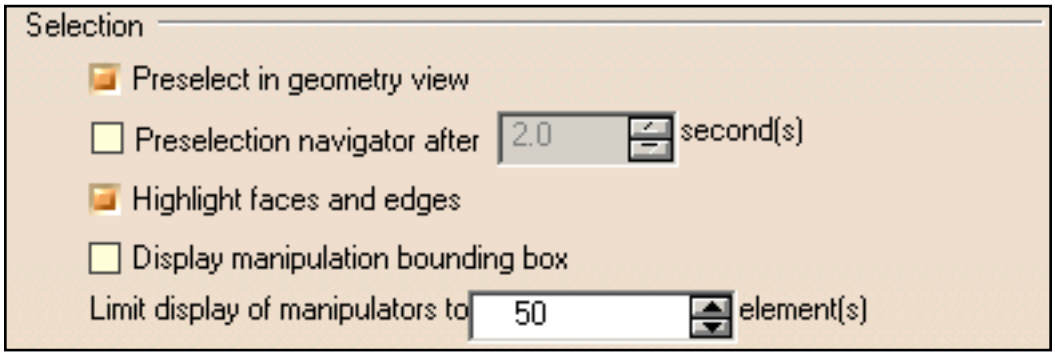
Up Next

Setting Advanced Graphical Properties

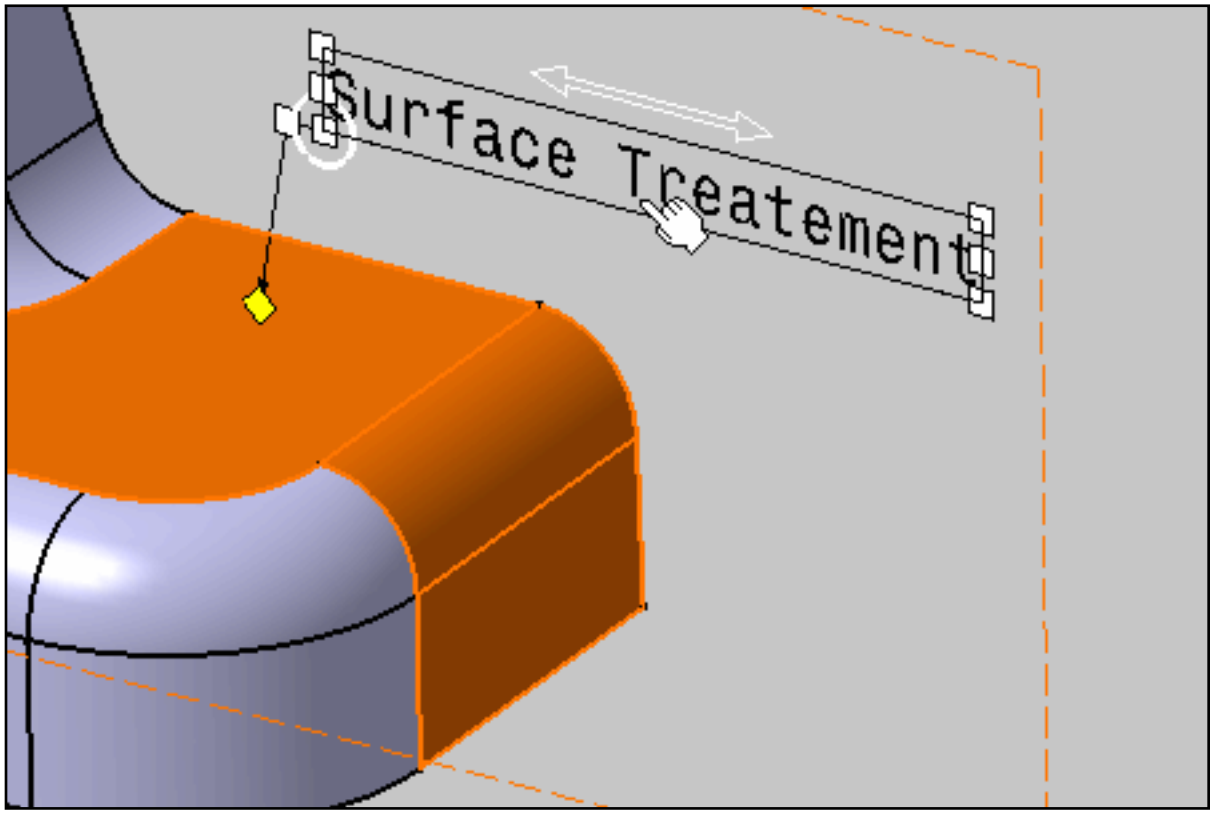
This task shows you how to set graphical properties from the **Properties** dialog box for a textual annotation.
Note that the operating mode described here is valid for datum elements, datum targets and geometrical tolerances too.

Open the Tolerancing_Annotations_04 CATPart document.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

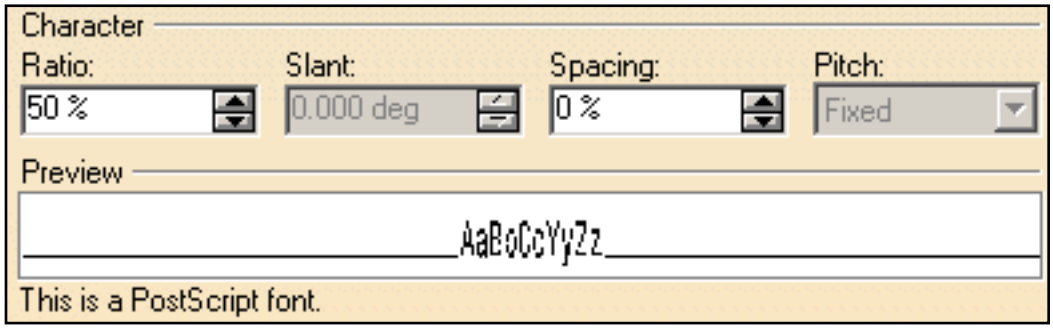


1. Select the annotation text.



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

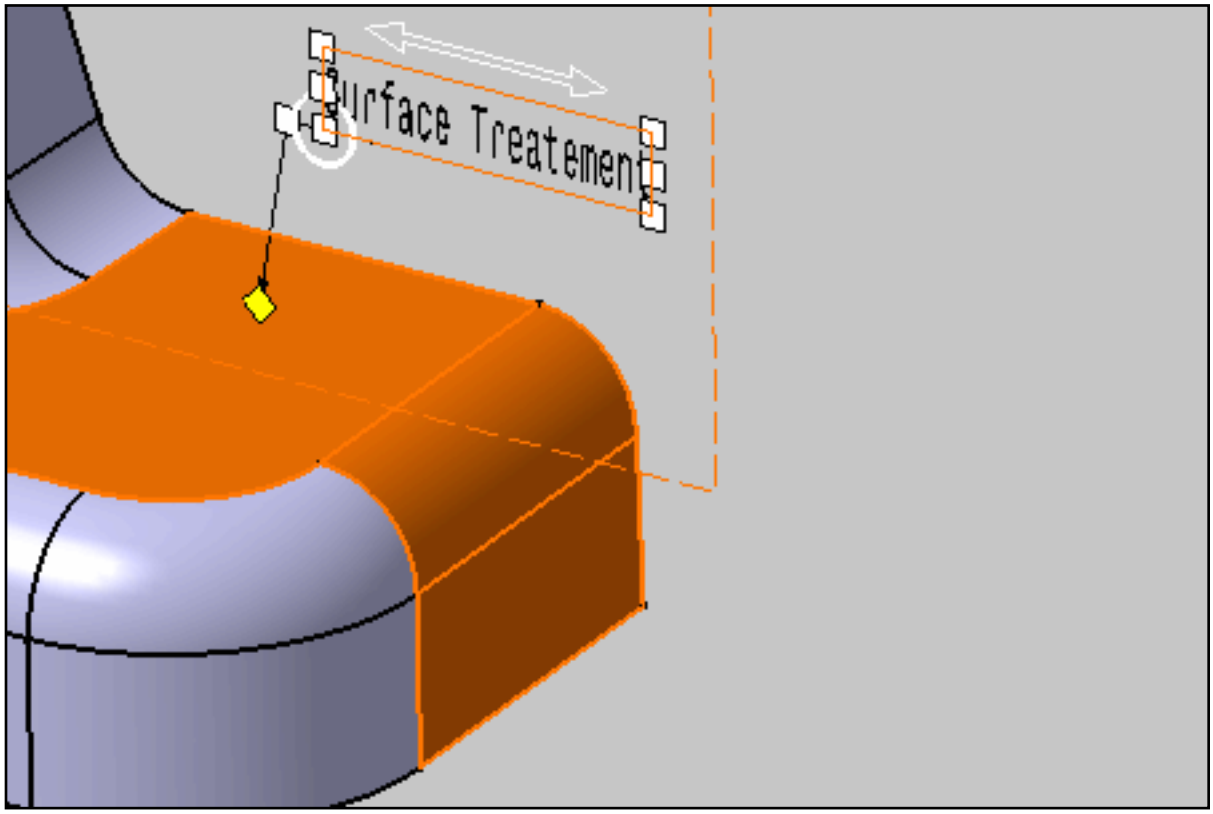
3. Select the **Font** tab and set the **Character Ratio** to 50%.



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.



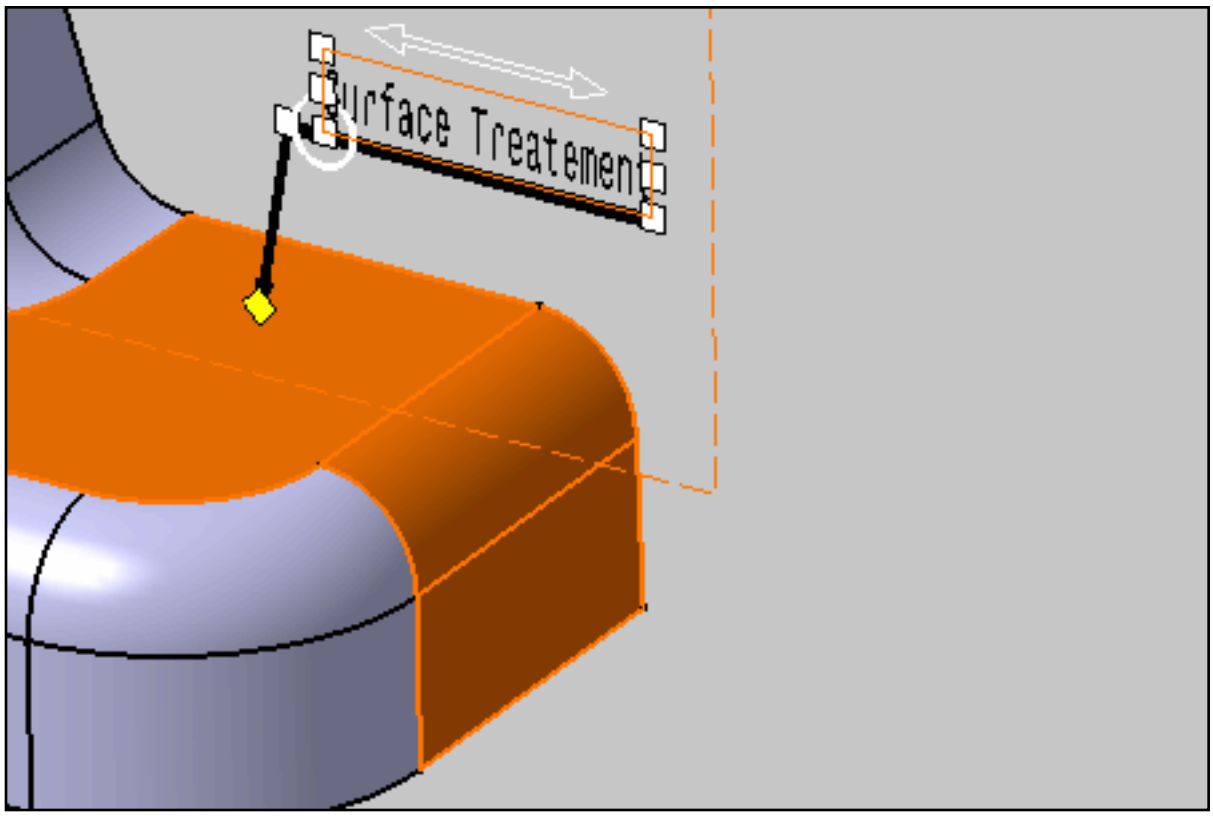
5. Select the **Text** tab and set the **Thickness** option to 2.00 mm to make the leader more visible.



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.



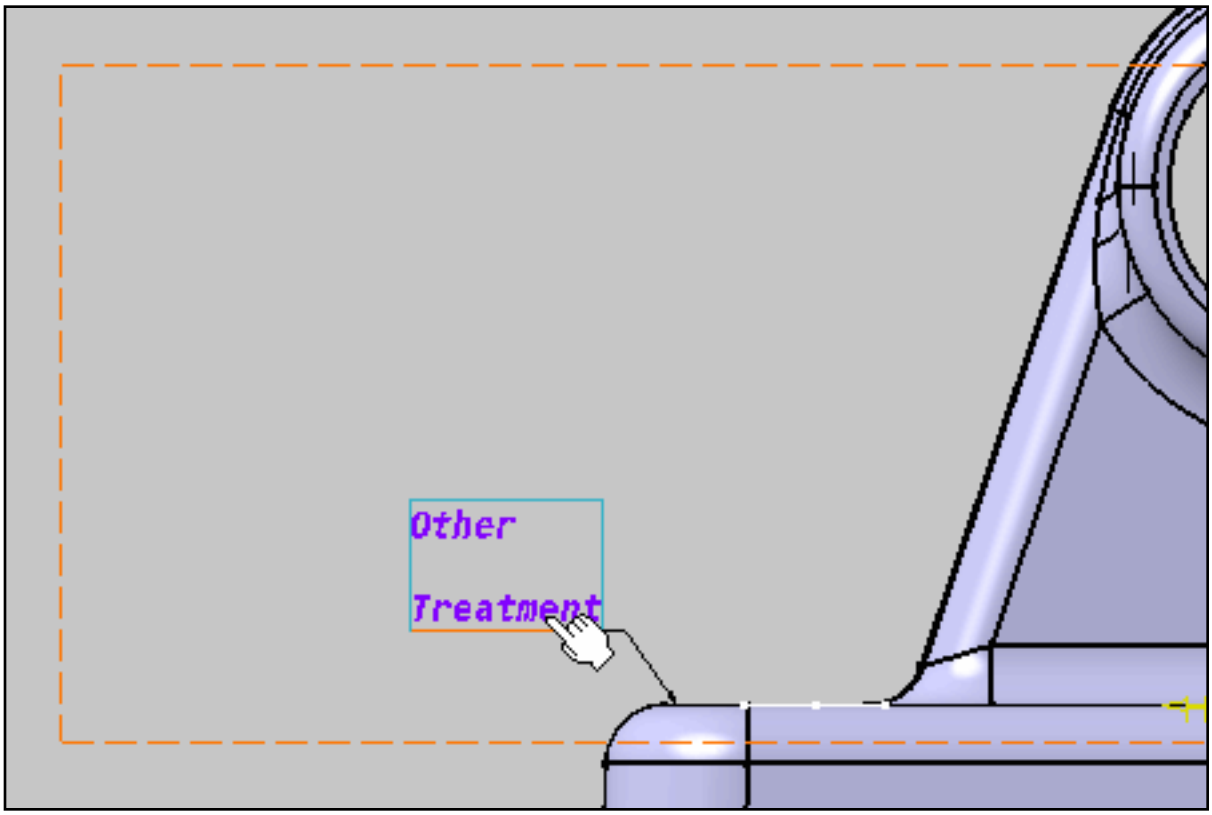
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Setting Graphical Properties as Default

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 Tolerancing_Annotations_04 CATPart document.

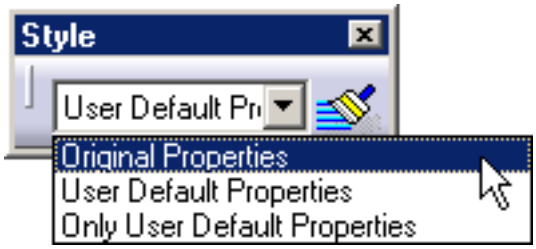
1. Select the text.



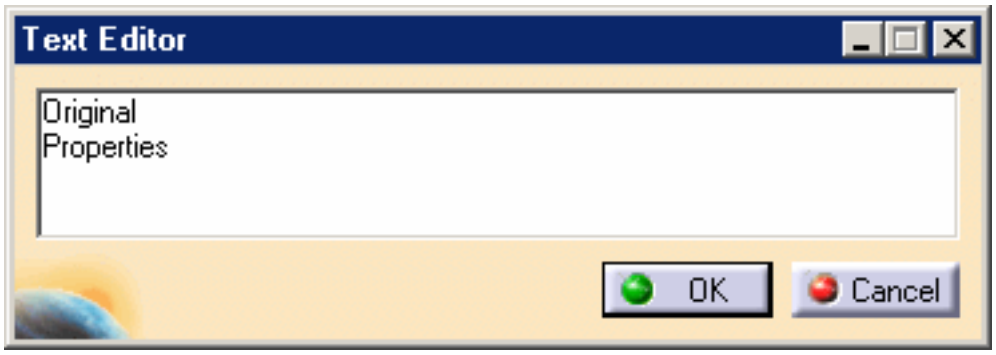
2. Right-click and select the **Set as Default** contextual option.

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.

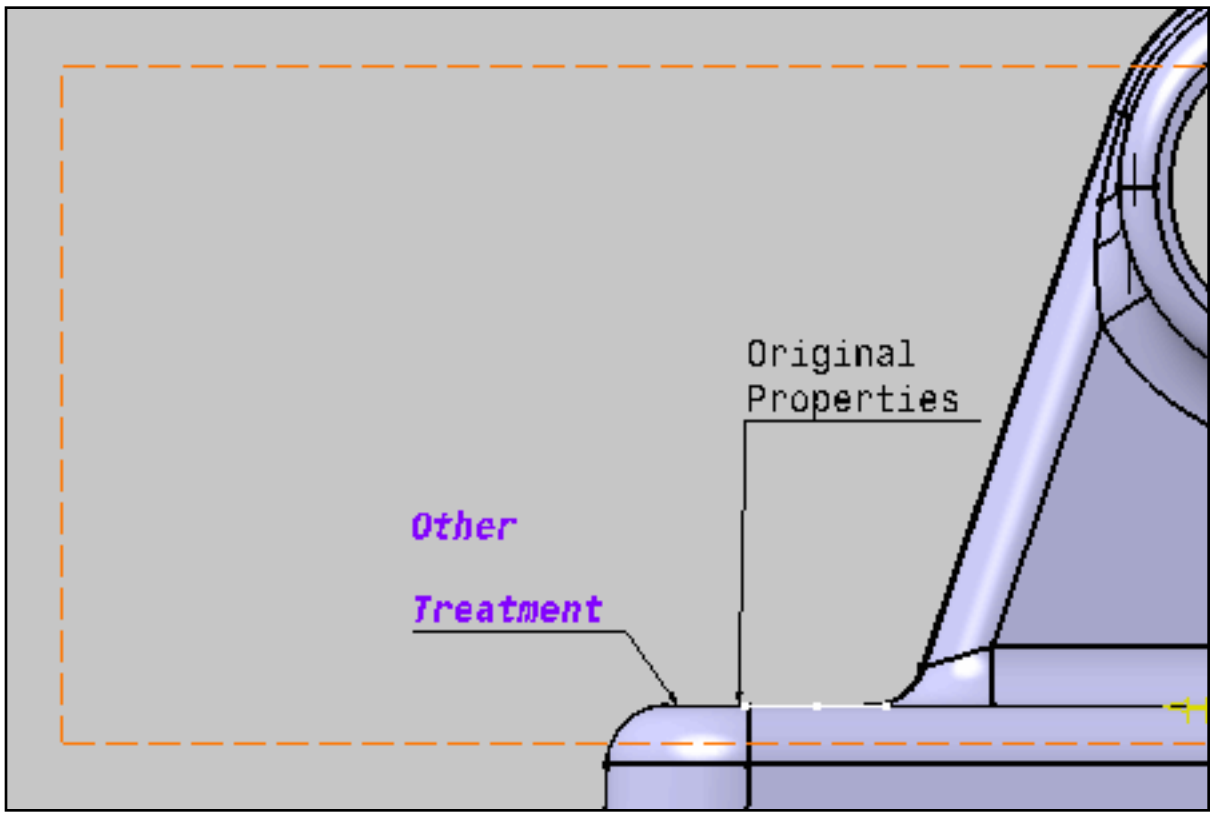
3. Select the **Original Properties** option in the **Style** toolbar.



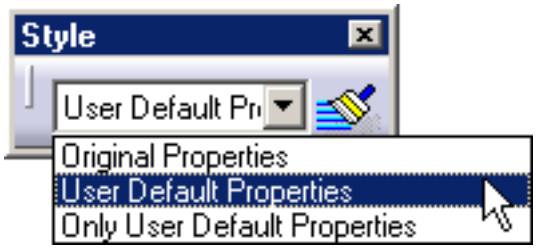
4. Create a new text and enter the following text: Original + Press Enter + Properties
   See Creating an Annotation Text.



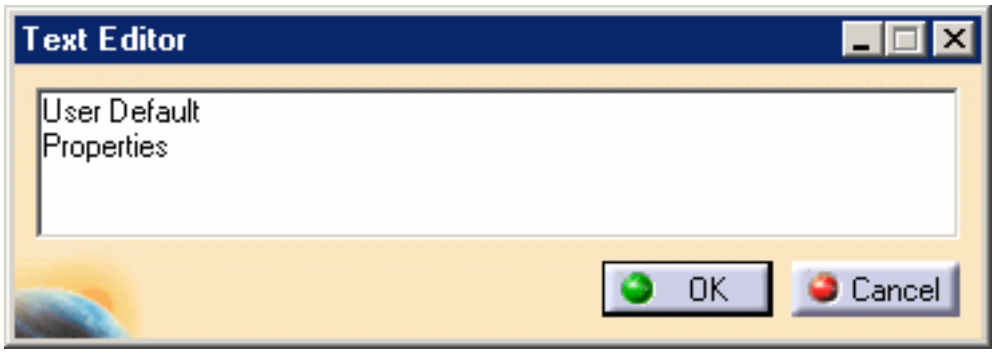
The text appears in the geometry and takes only the application default properties.



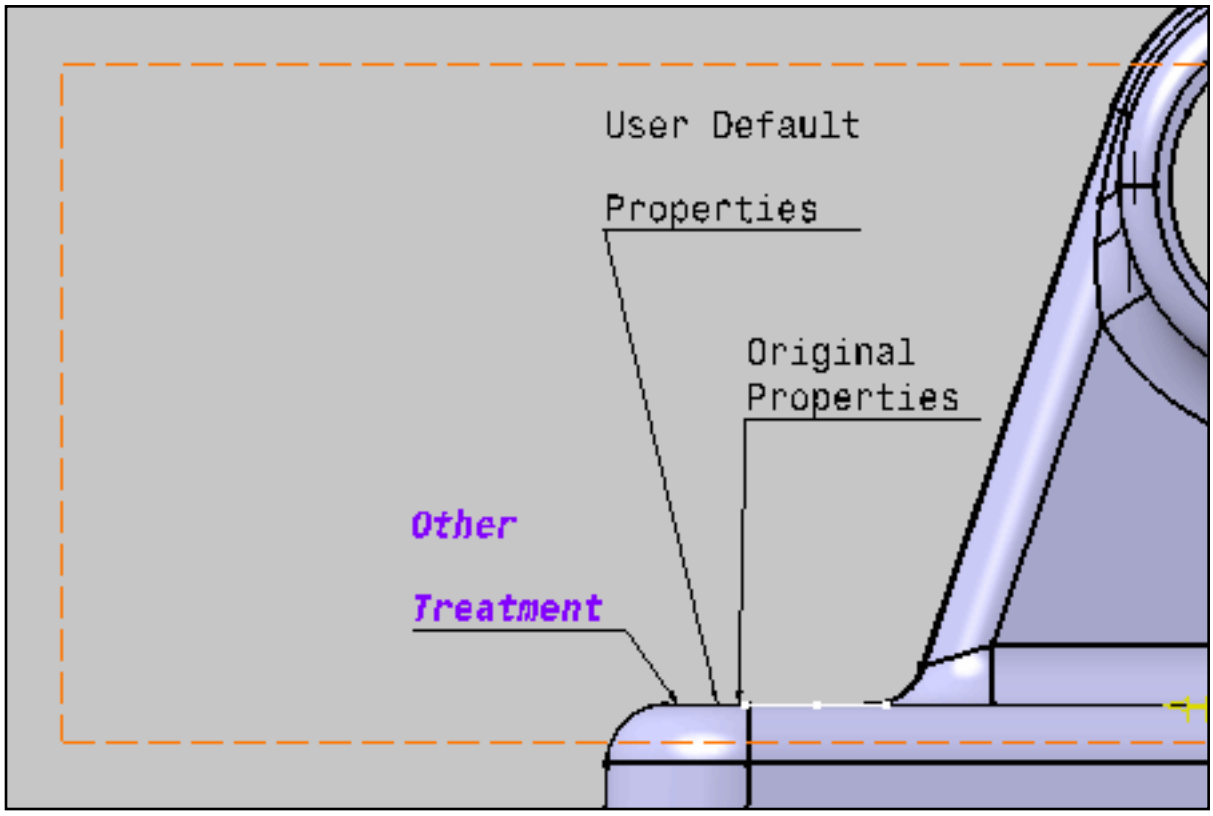
5. Select the **User Default Properties** option in the **Style** toolbar.



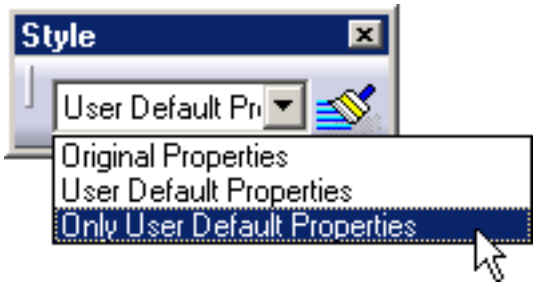
6. Create a new text and enter the following text: User Default + Press Enter + Properties



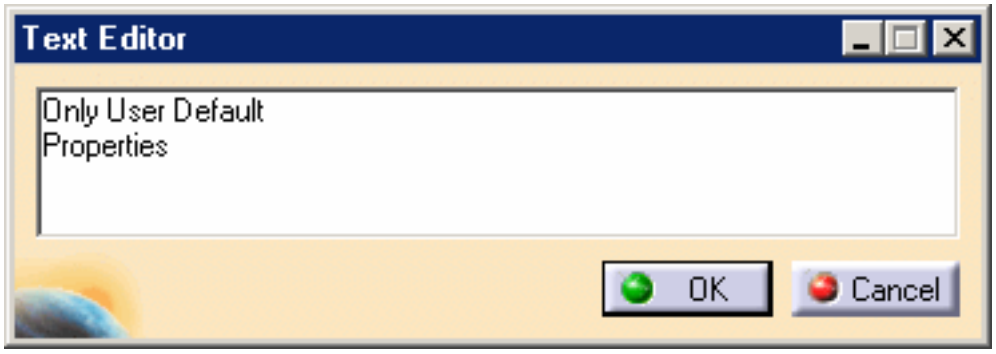
The text appears in the geometry and takes the application default properties and the **Text: Line Spacing** property from the first text.



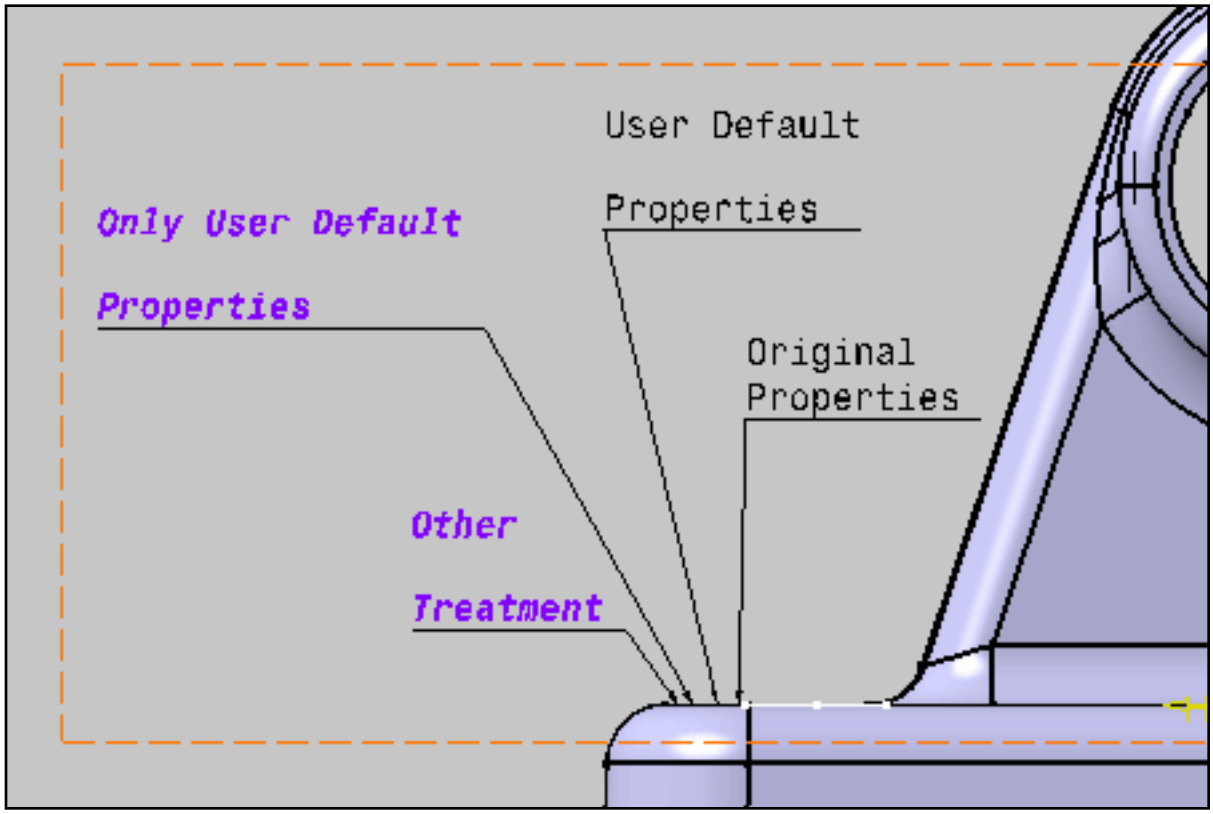
7. Select the **Only User Default Properties** option in the **Style** toolbar.



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.



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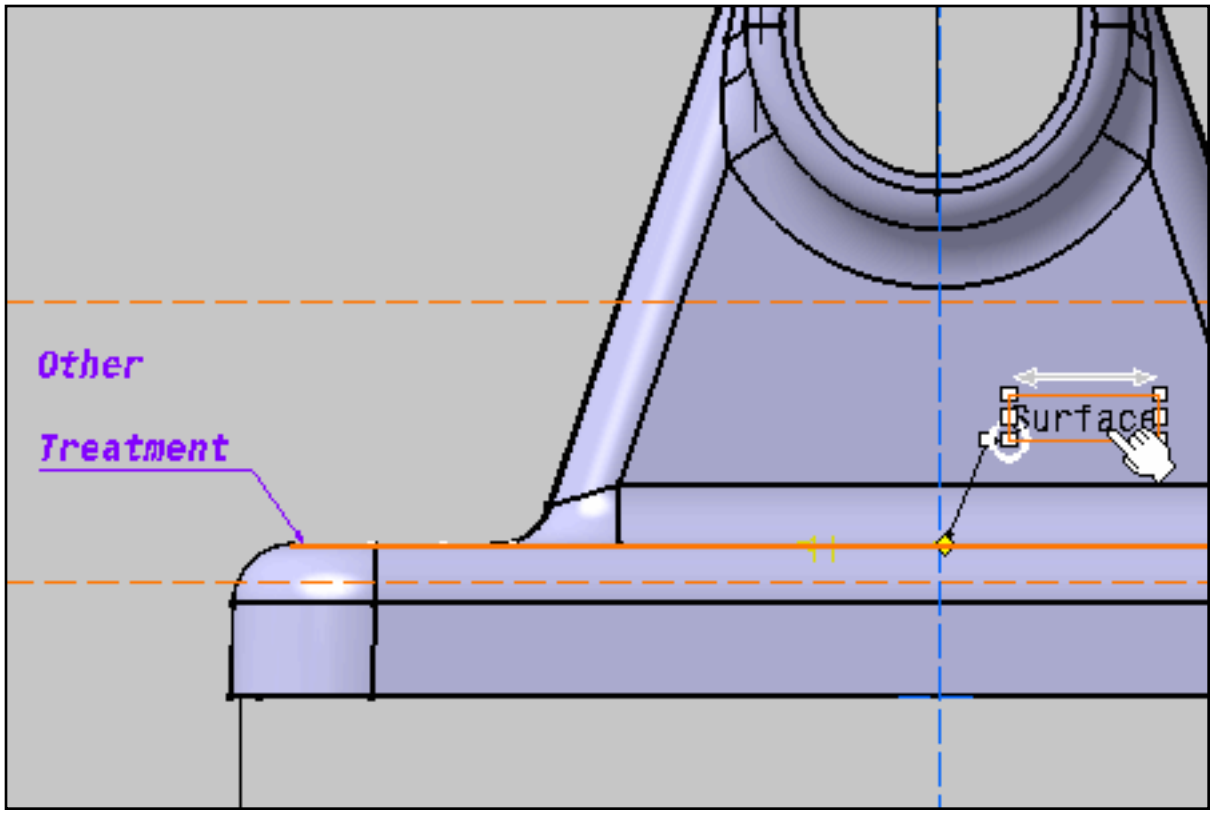
Copying Graphical Properties

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.

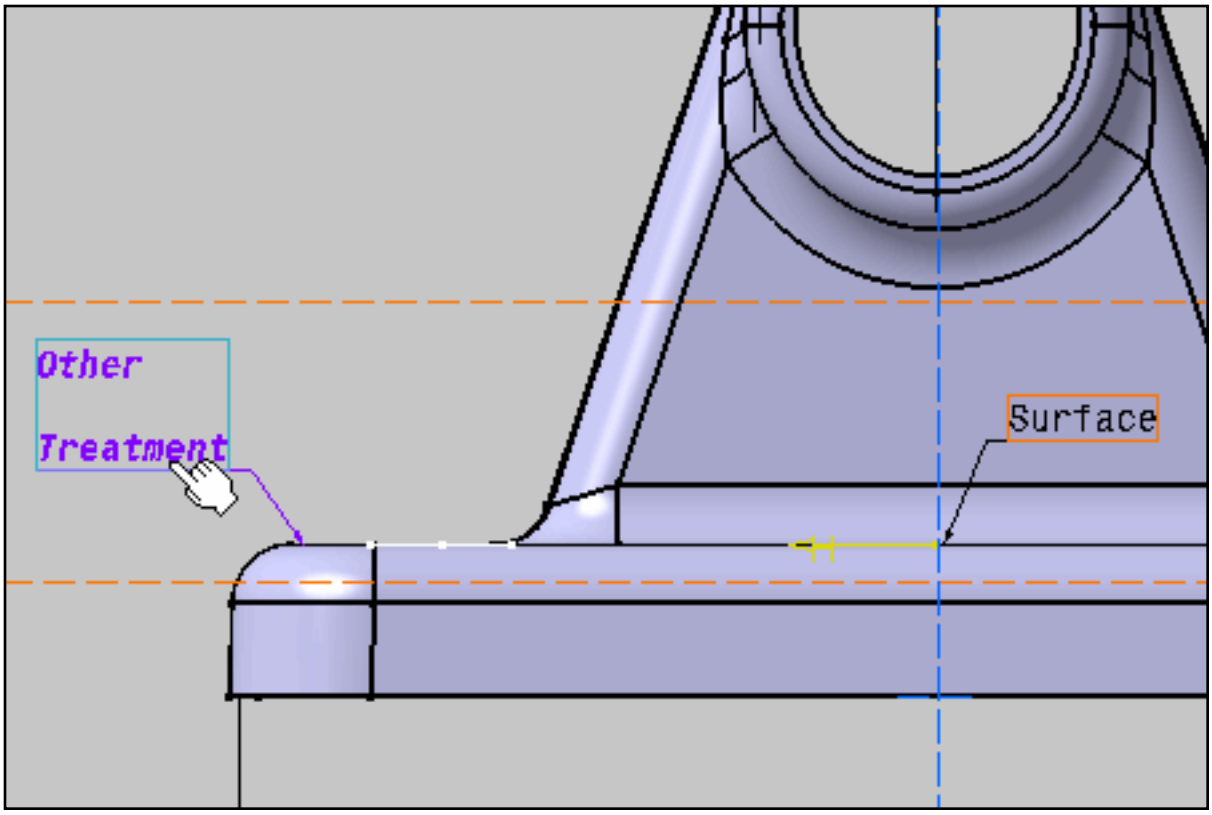
Open the Tolerancing_Annotations_04 CATPart document.

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

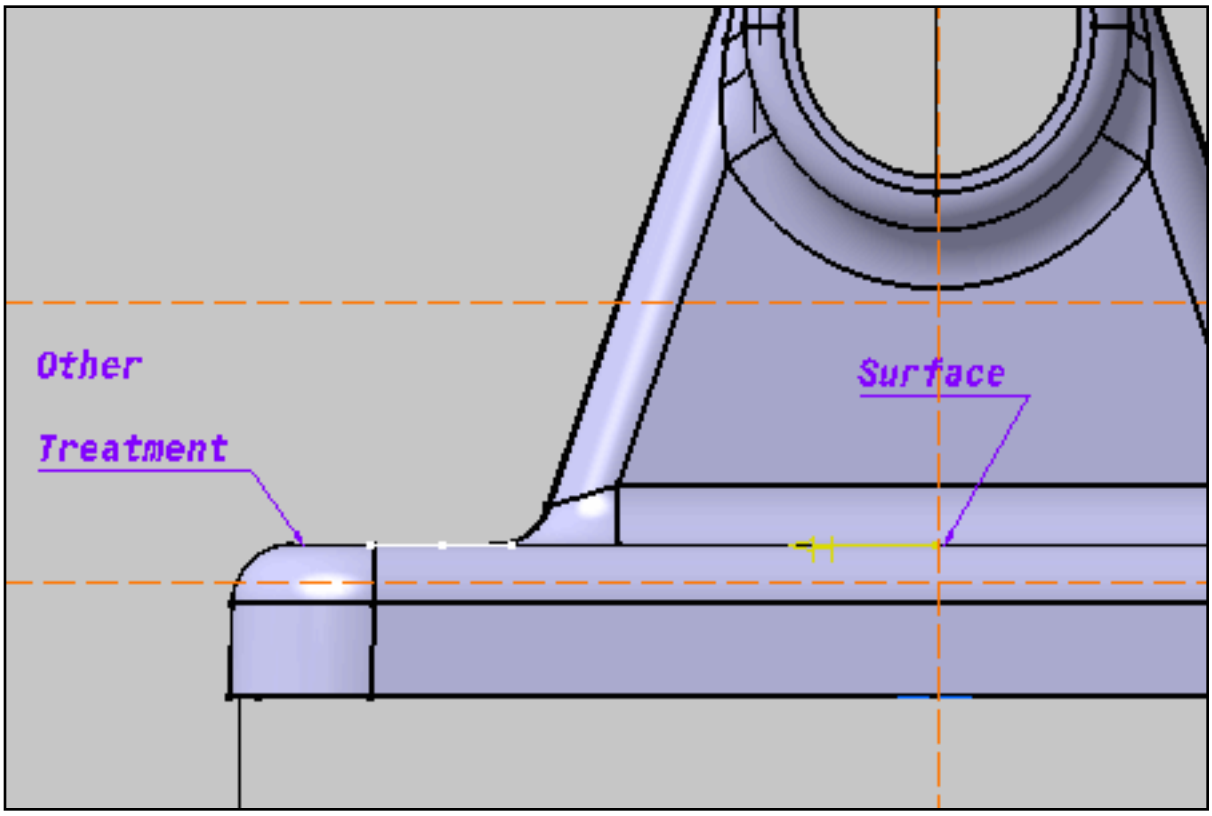


2. Click the **Copy Object Format** icon 

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.



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Filtering Annotations

	Use Annotation Filter: 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.

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Using Annotation Filters

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

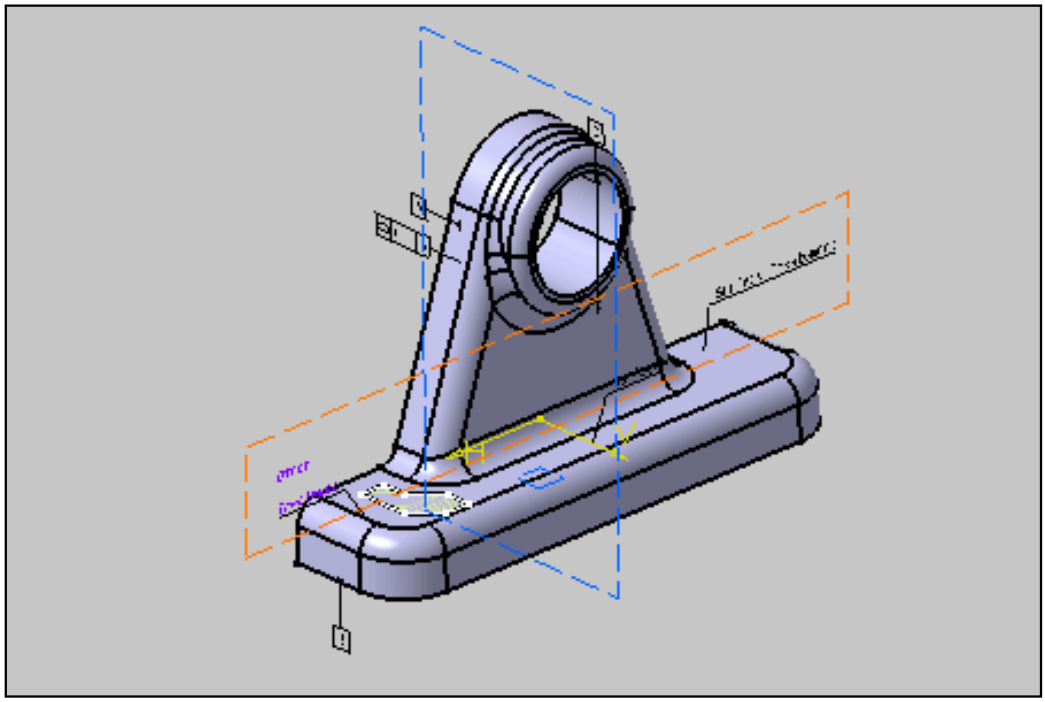
You can filter annotations display though the following features:

- Views/annotation planes
- Annotation sets
- Geometrical elements
- 3D annotations

- Any Part Design feature
- Any Generative Shape Design feature
- Restricted areas

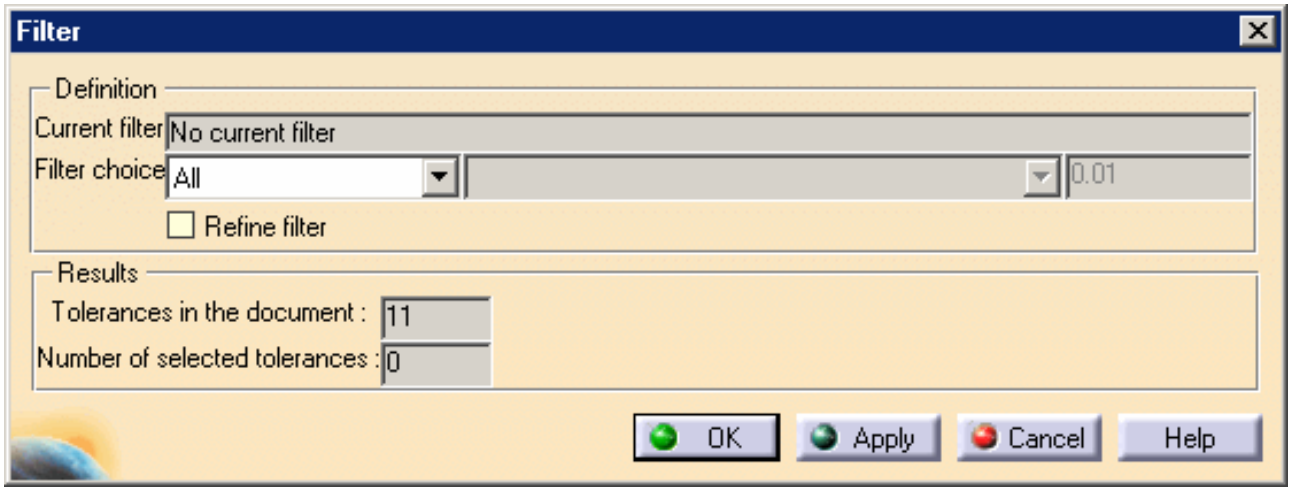
In the case of Part Design or Generative Shape Design features, only the annotations that are directly or indirectly applied to the geometrical elements which compose the feature will be displayed when applying the filter. In the case of restricted areas, only the annotations that are directly or indirectly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

Open the Tolerancing_Annotations_04 CATPart document.



1. Click the **Filter** icon: 

   The **Filter** dialog box is displayed.



You can filter the display of annotations in the 3D viewer using the following criteria:

- All
- None
- 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 default filter, **All**, displays all the geometrical tolerance annotations.

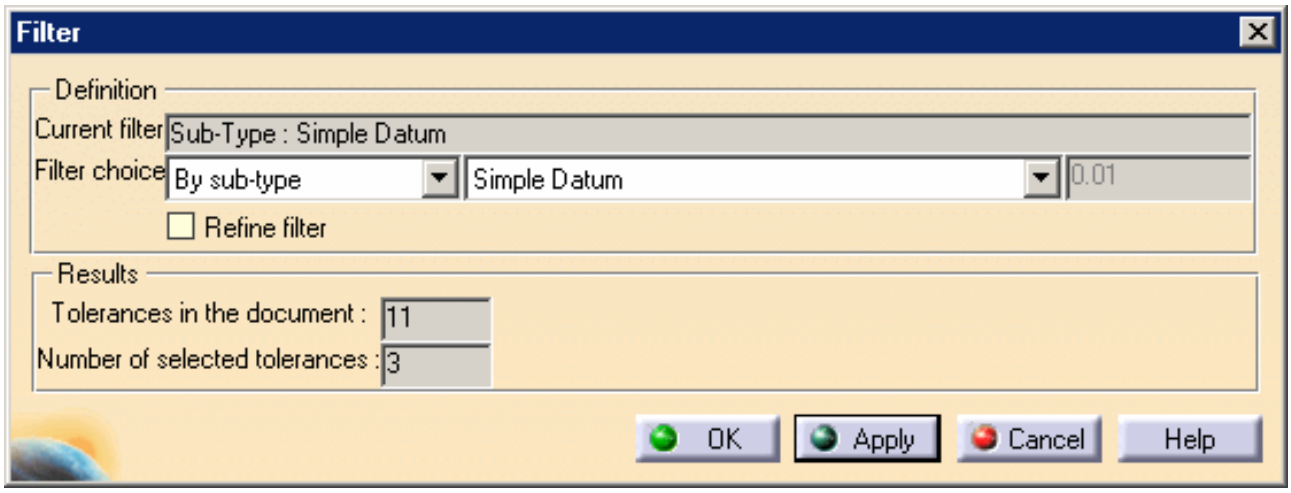
The **Results** area provides the following information:

- 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.

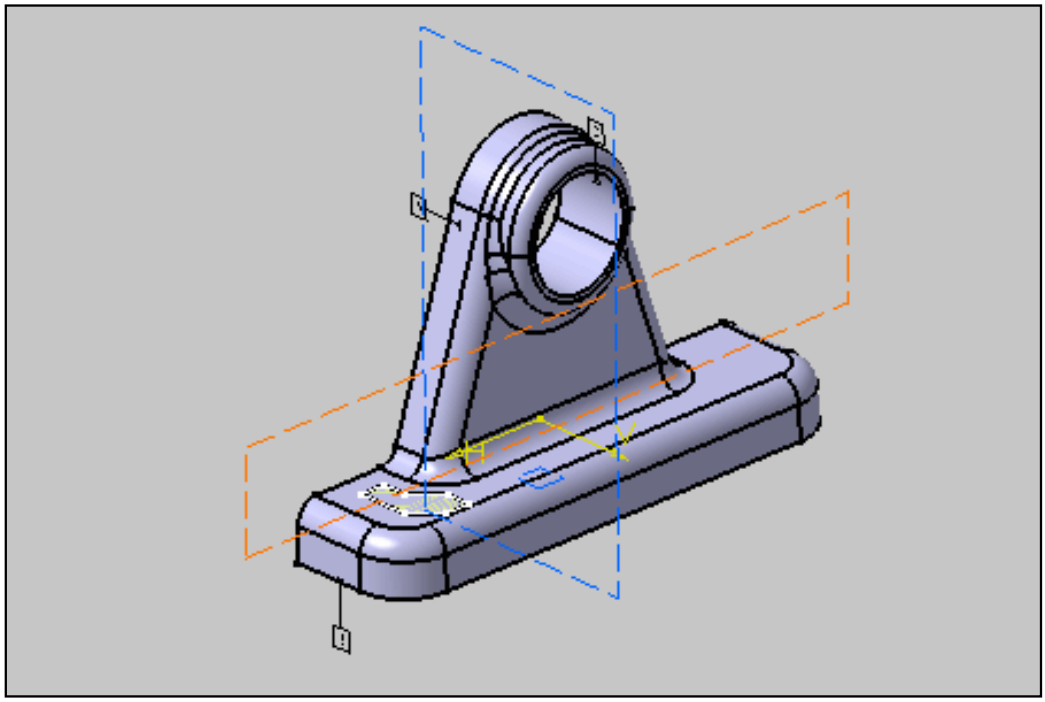
2. Set the **Filter** choice field to "By sub-type".

3. Set the **Simple Datum** sub-type. The Number of selected tolerances field displays "2".



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

4. Click **Apply**. Only simple datum are displayed.



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

Up Next

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.

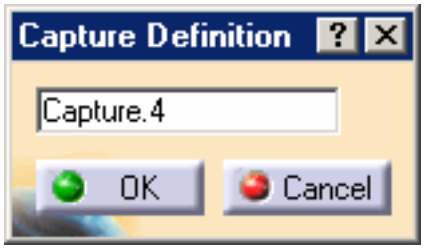
2D sheet equivalent.

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.



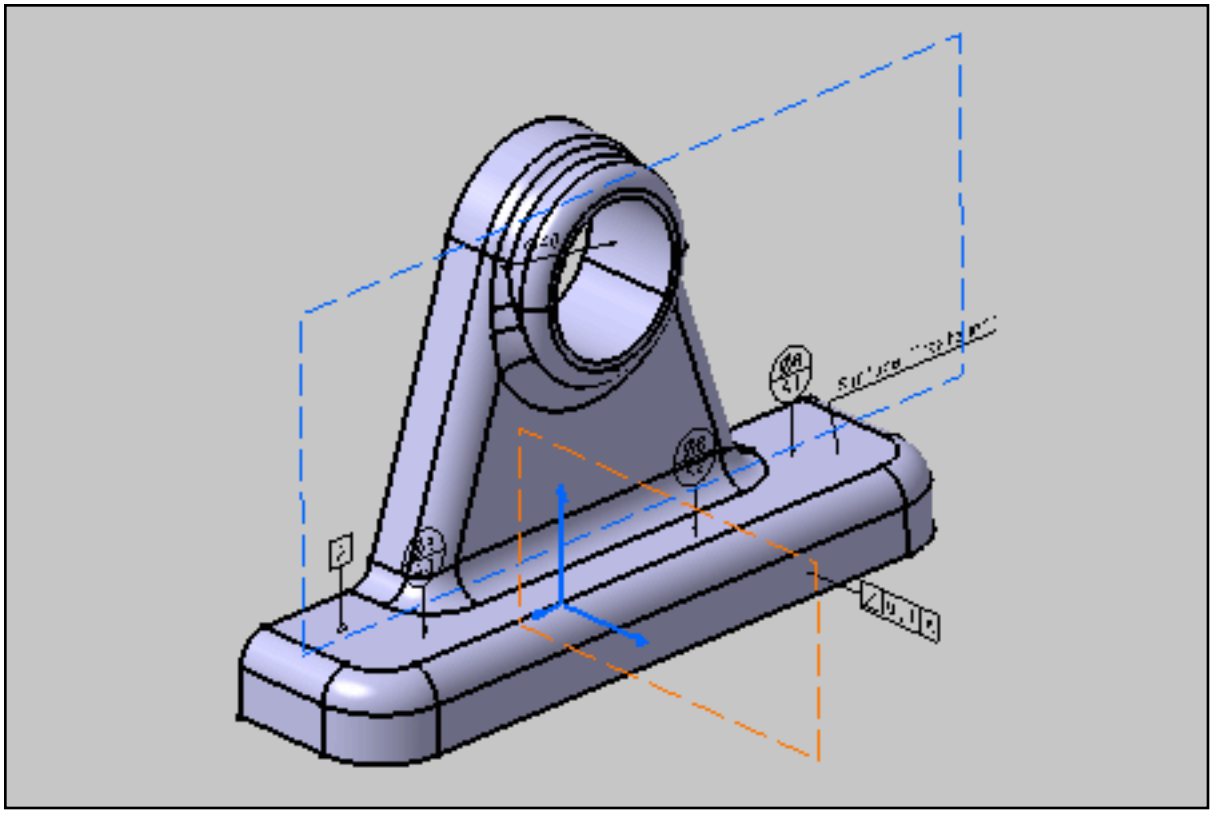
Back Up Next

Displaying a Tolerancing Capture

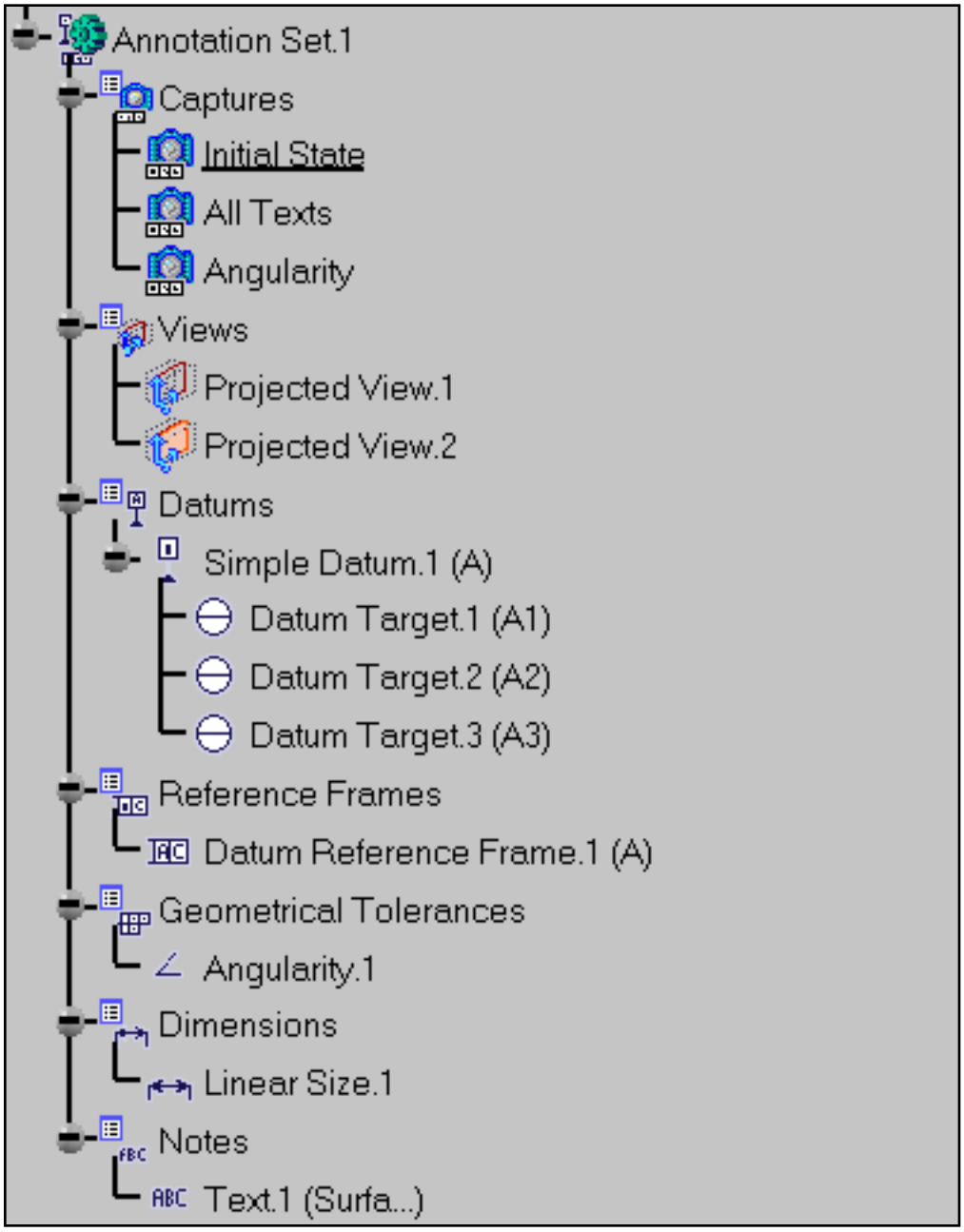
This task shows you how to display a tolerancing capture.
See Managing Capture Options.

Open the Tolerancing_Annotations_05 CATPart document.

The geometry looks like this.



The specification tree looks like this.



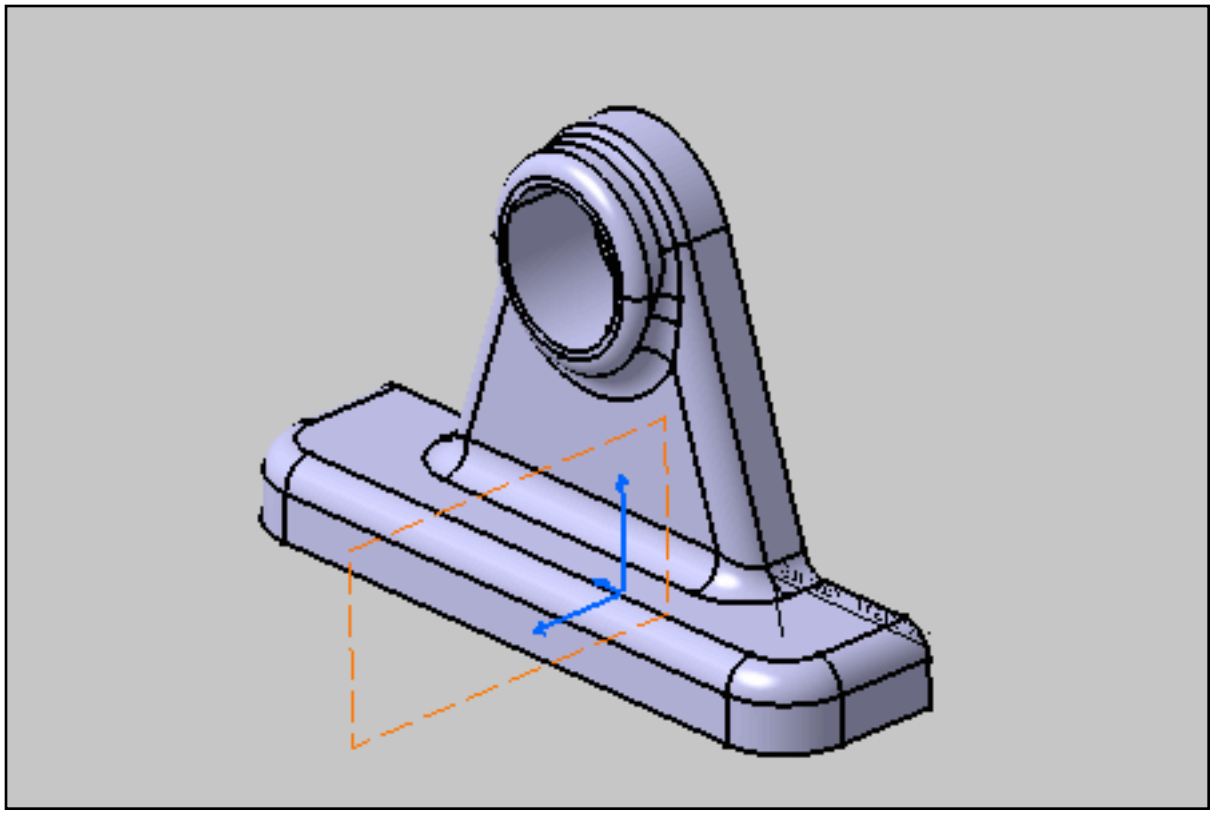
1. Right-click the **All Texts** capture and select **Display Capture** command from the contextual menu.

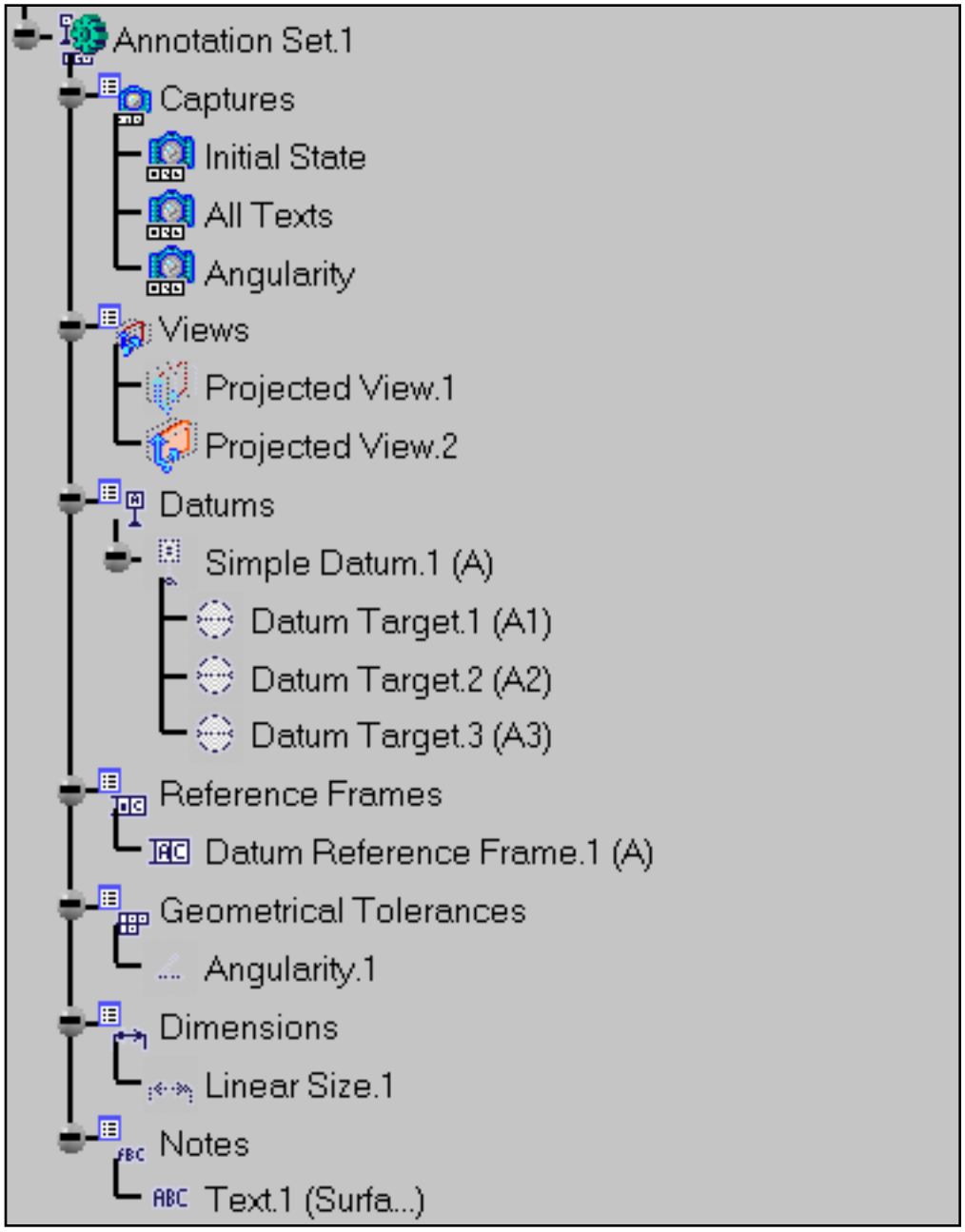
The geometry and specification tree look like this:

The geometry is zoomed and rotated.

All annotations are hidden except notes.

Projection View.1 is hidden.





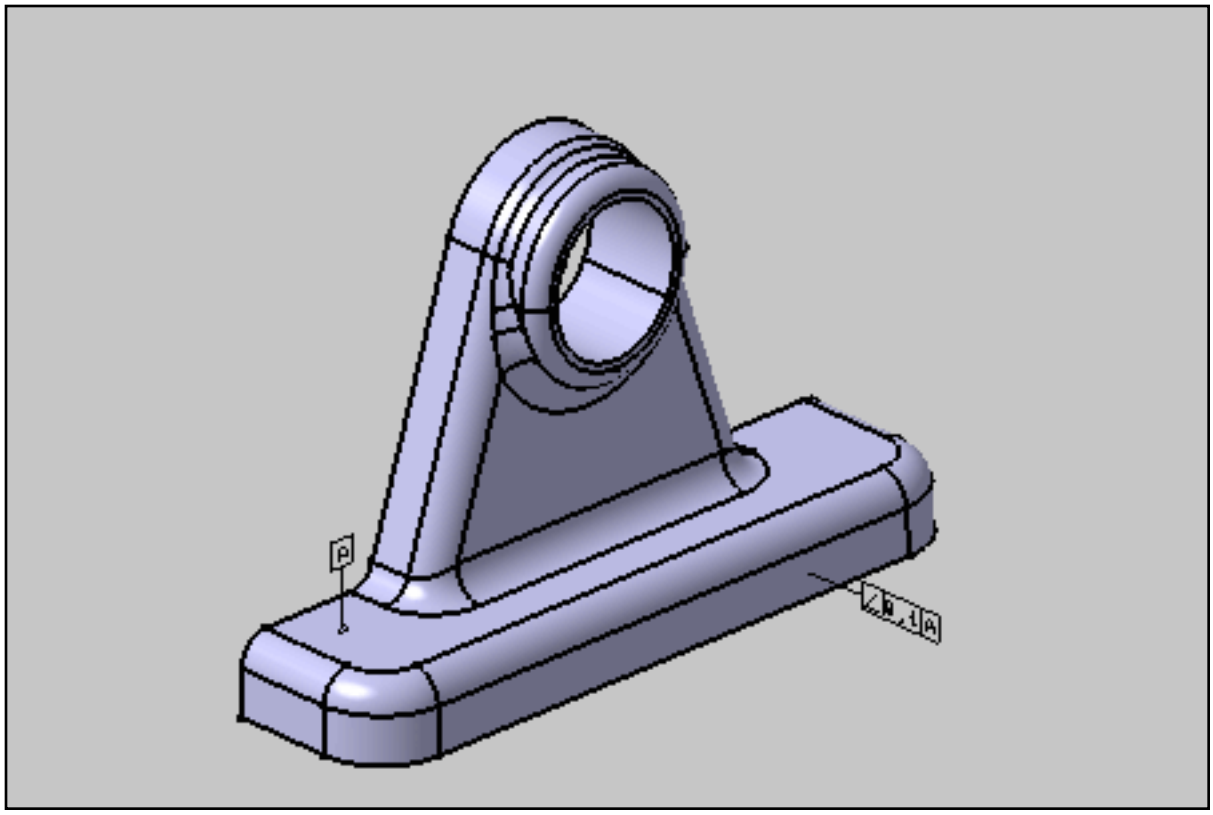
2. Right-click the **Angularity** capture and select **Display Capture** form the contextual menu.

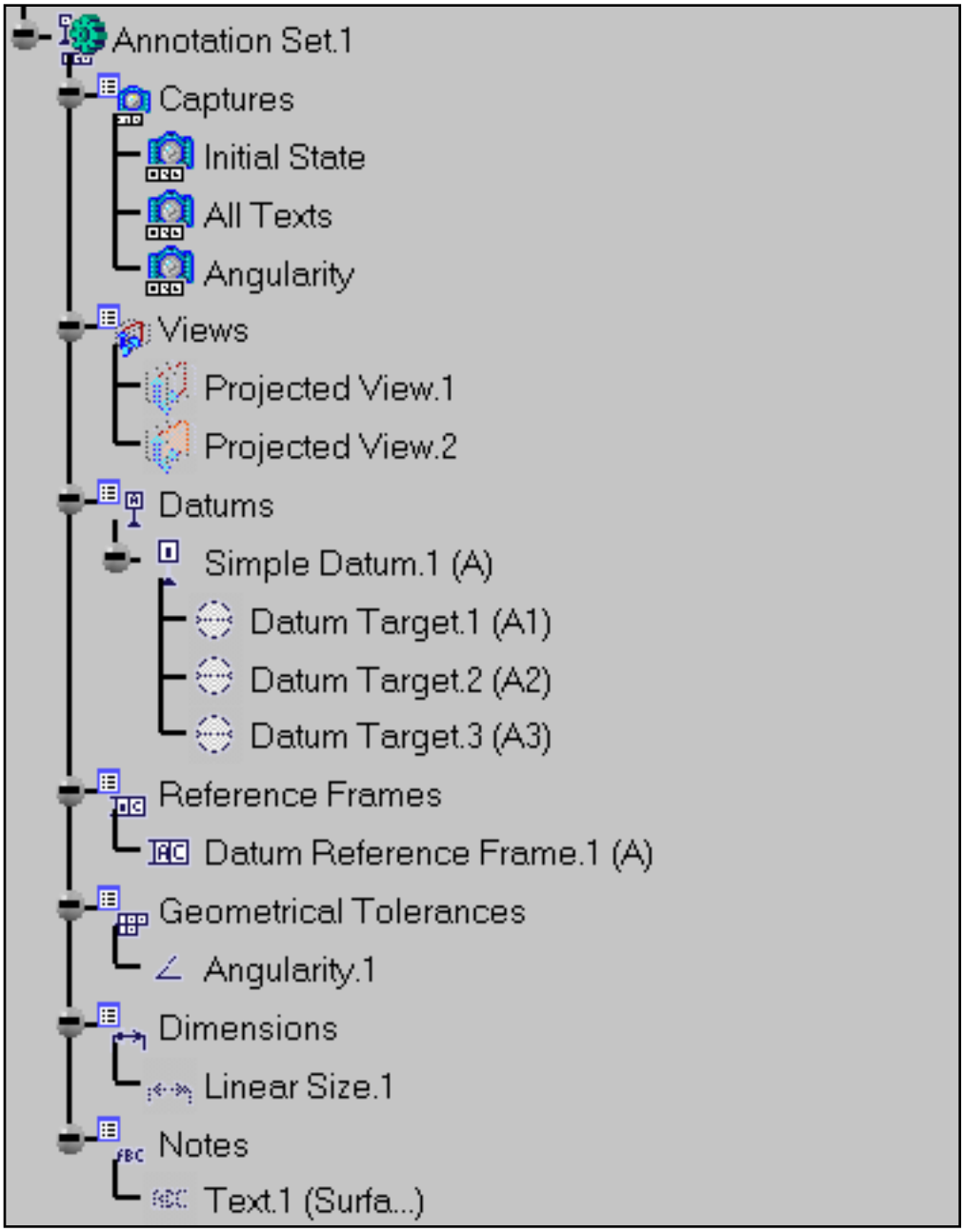
The geometry and specification tree look like this:

The geometry is moved and rotated.

Annotation planes are hidden.

Only the datum and the geometrical tolerance are shown.


A
A


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...)

Back Up Next

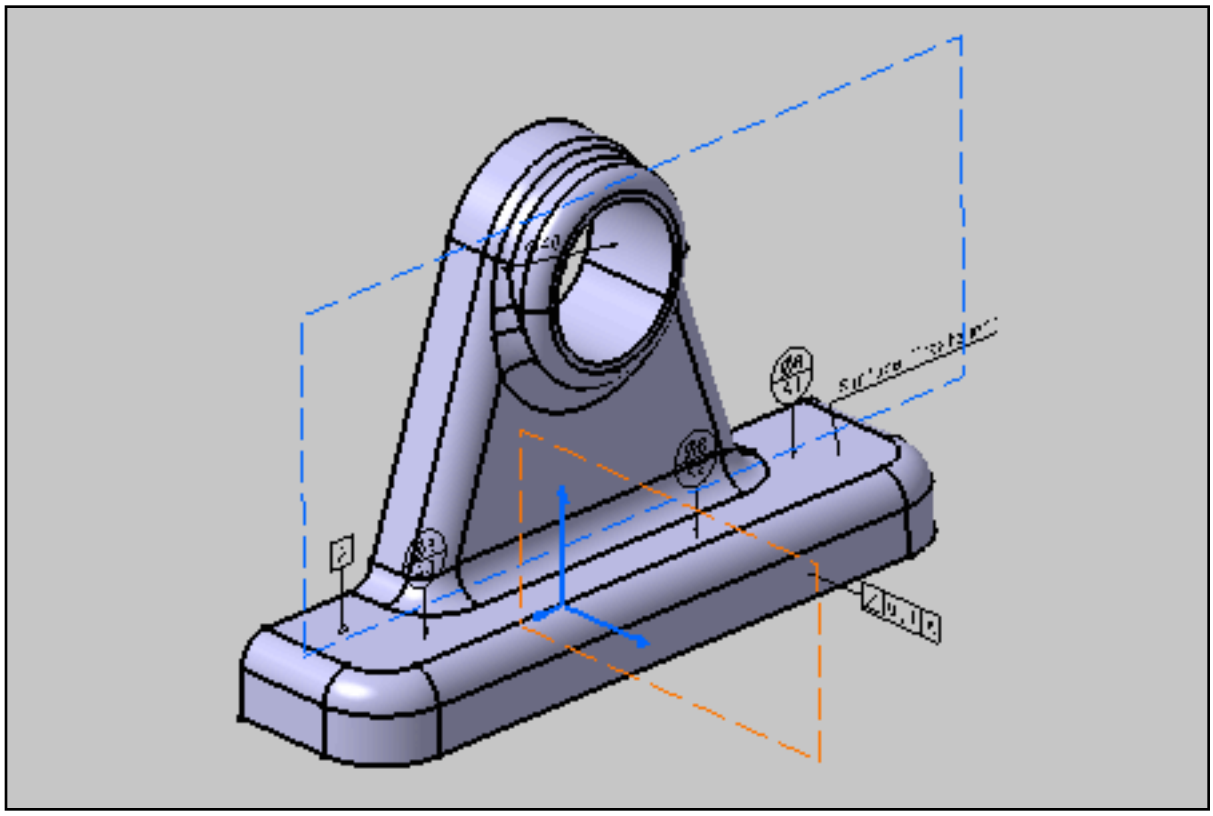
Managing Tolerancing Capture Options

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

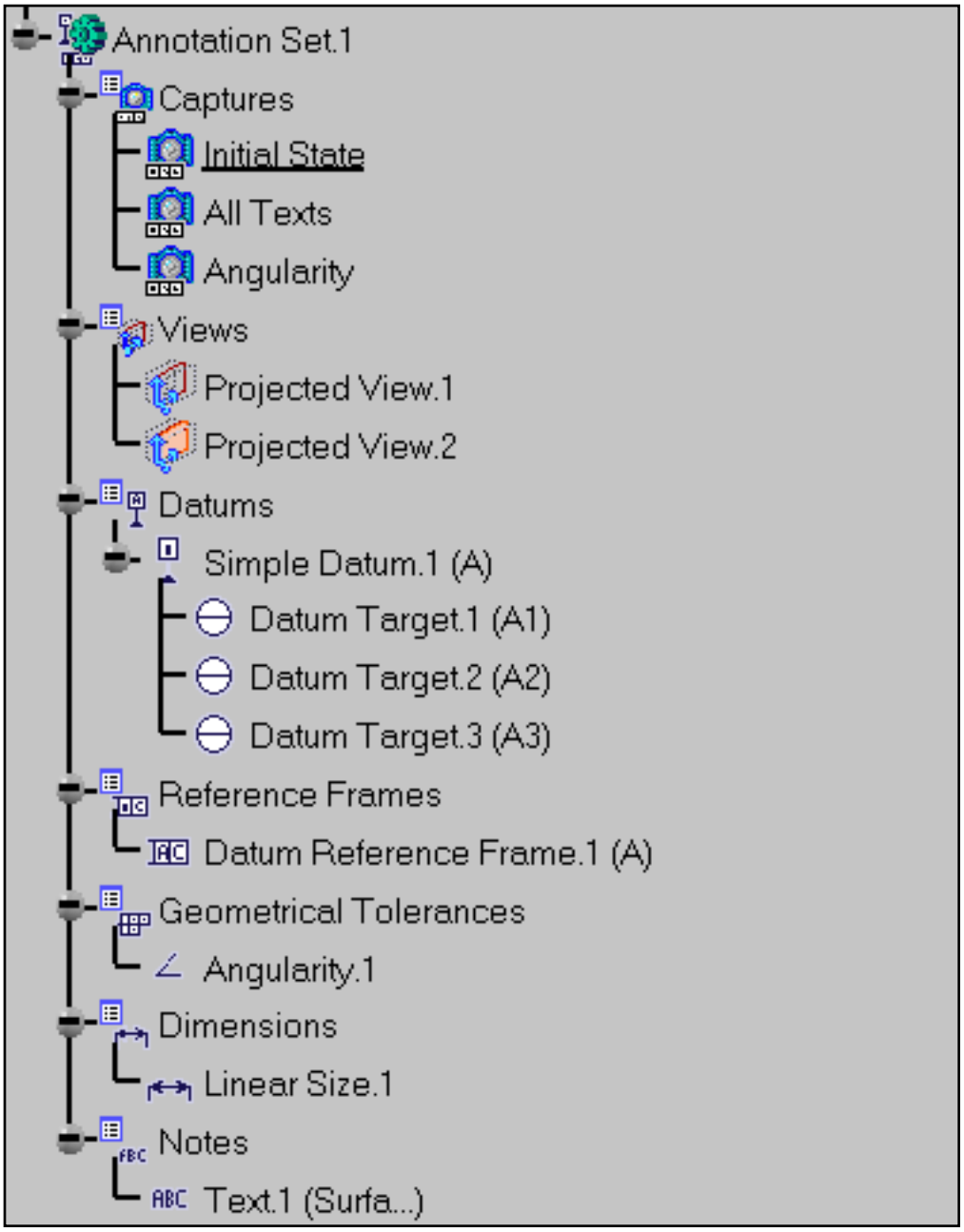
Open the Tolerancing_Annotations_05 CATPart document.

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

   You are now in the Tolerancing & Annotation Captures workshop.

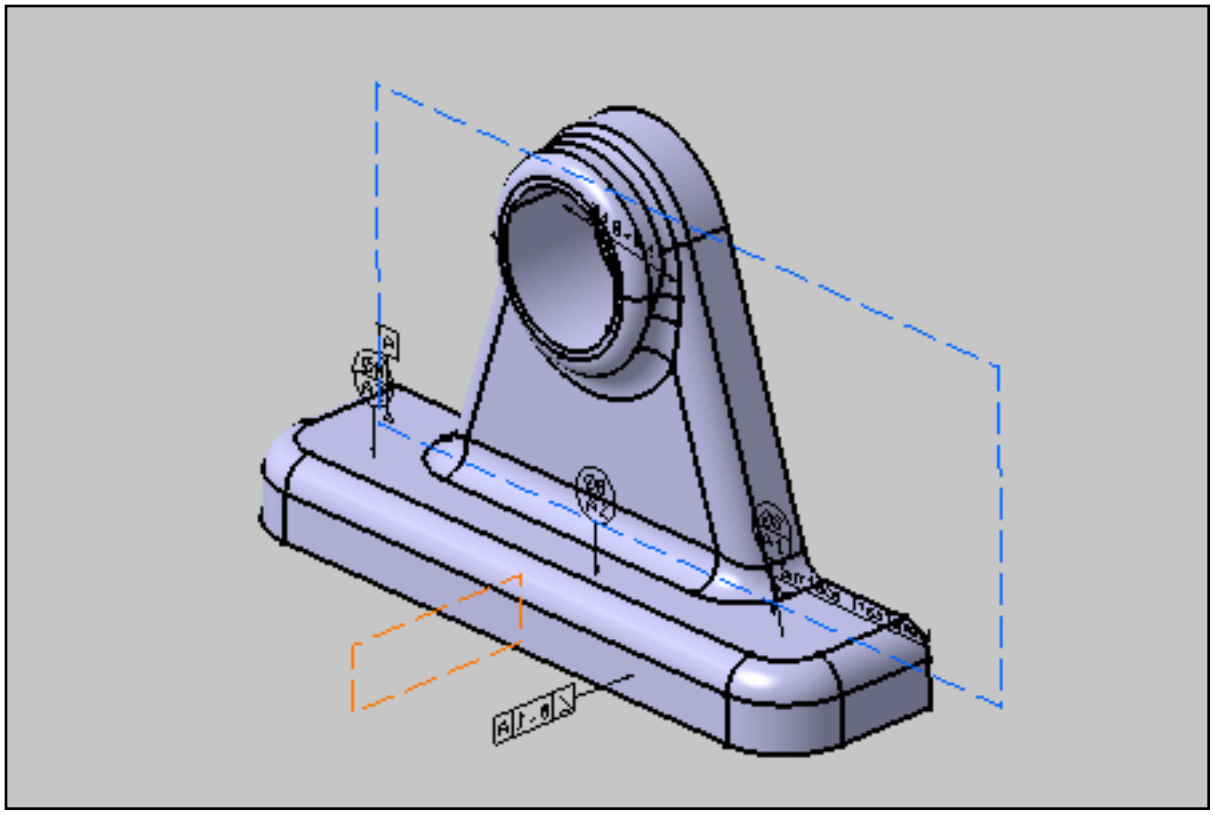


   The geometry and specification tree are displayed according to the capture options.



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

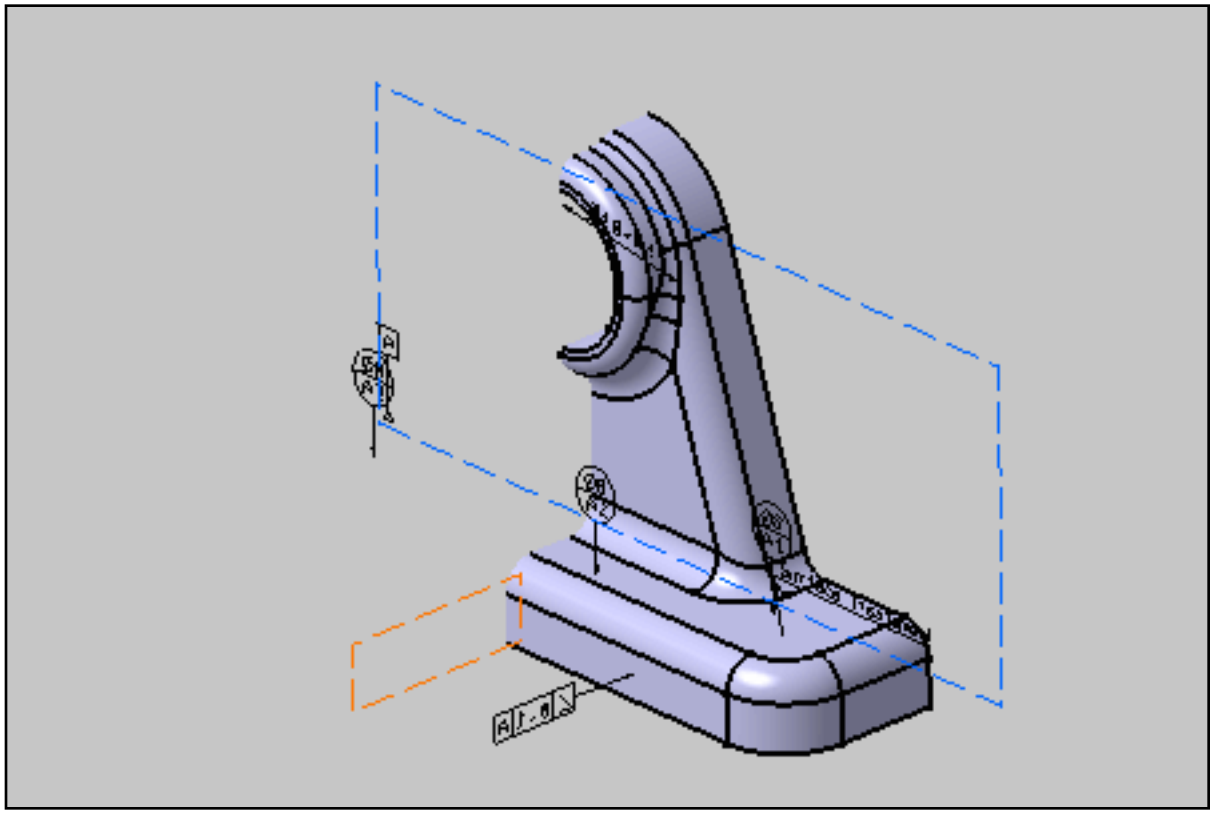
   A camera is associated with the capture. See Creating a Camera.



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.



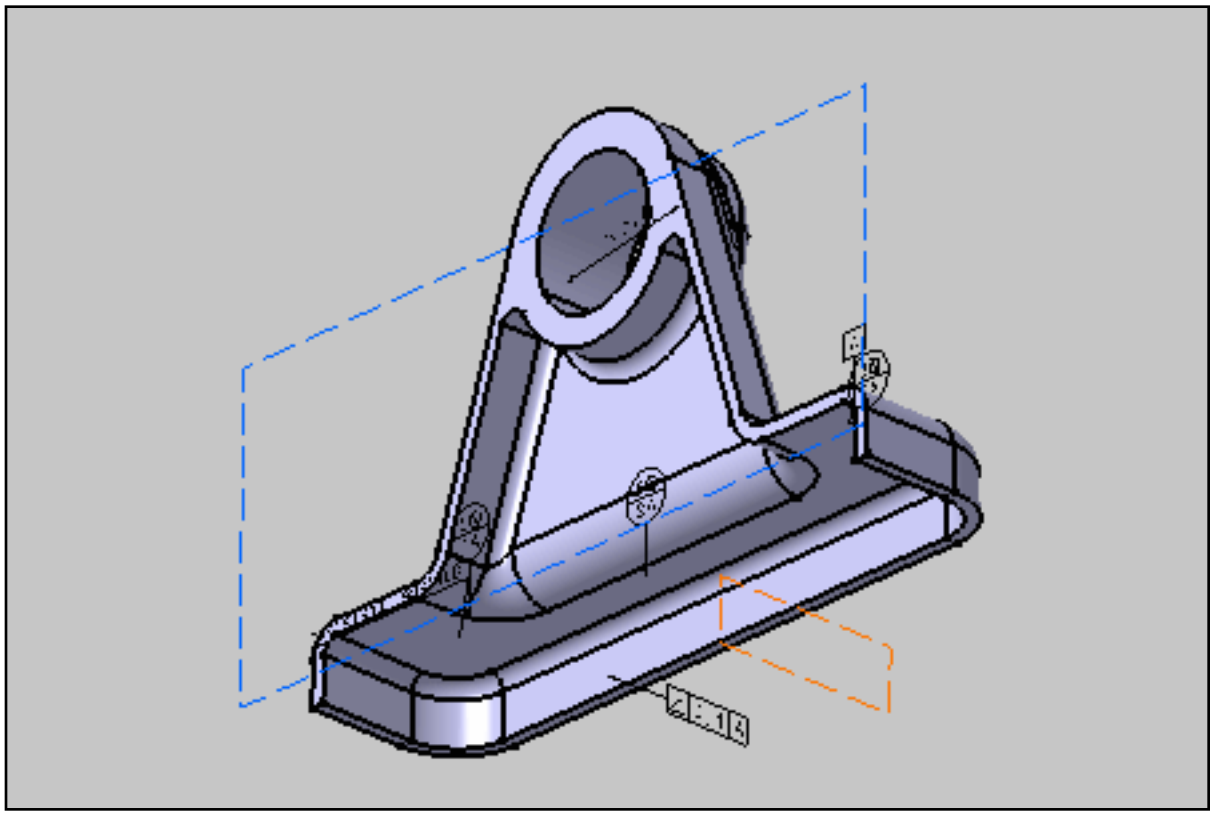
Back Up Next

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.

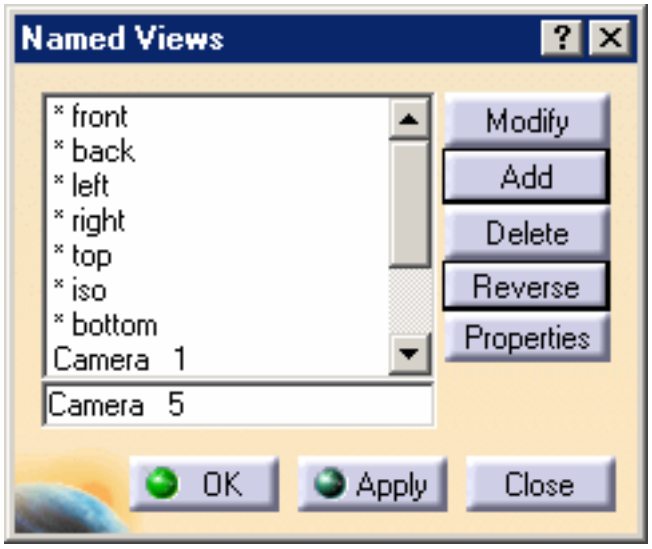
1. Move and rotate the part like this.



2. Select the **Named views** icon 

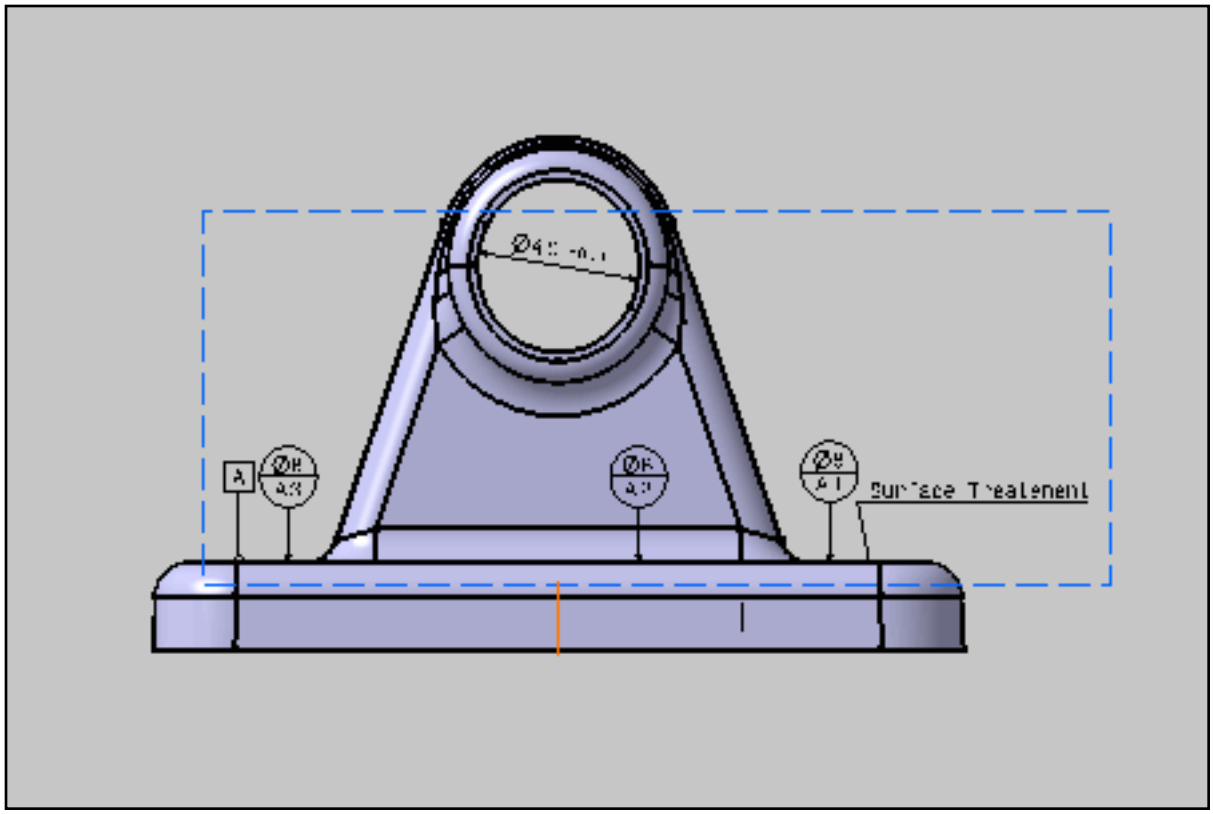
3. Click **Add** in the **Named Views** dialog box which appears.

   The **Camera 5** is created.



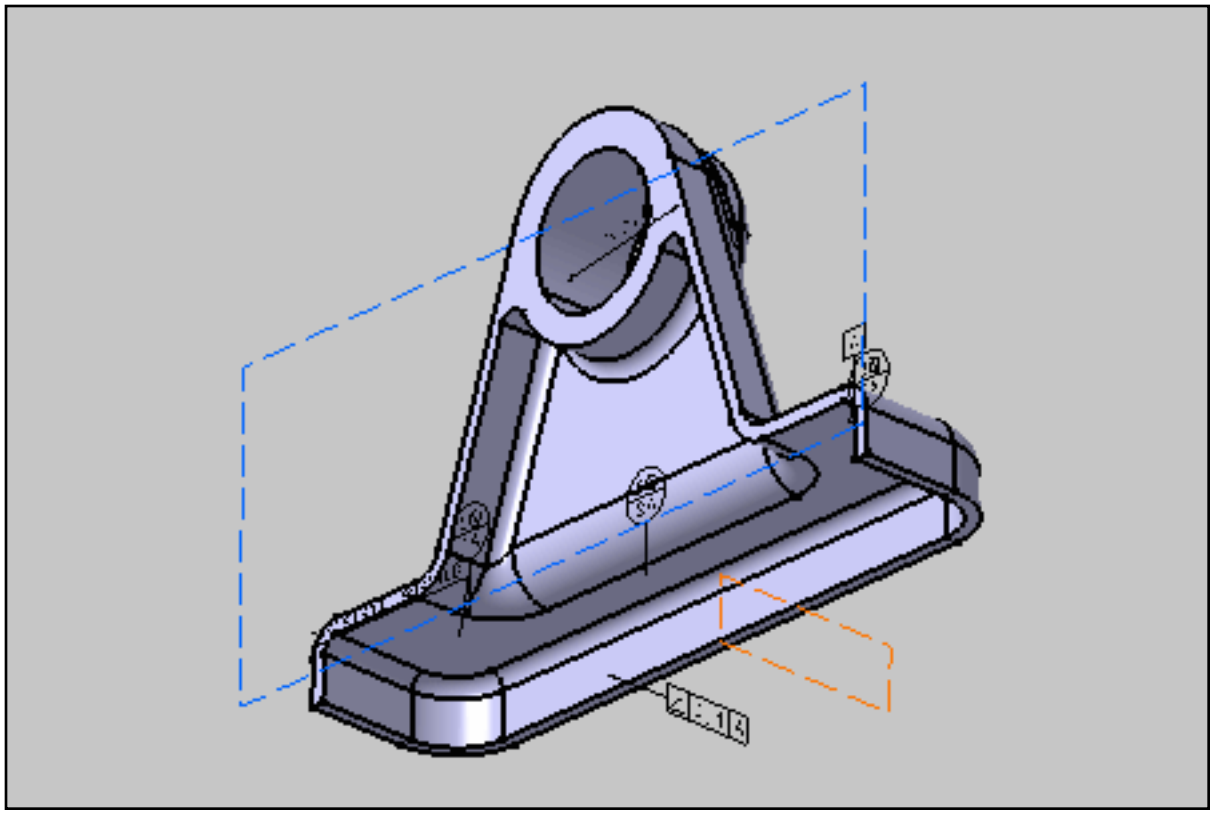
4. Select the *** right** camera the **Named Views** dialog box and click **Apply**.

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**.

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Using Capture Management

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

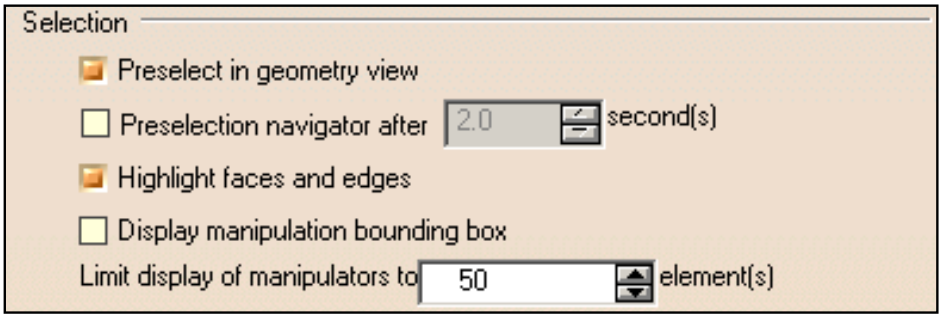
This command allows you to:

- 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.

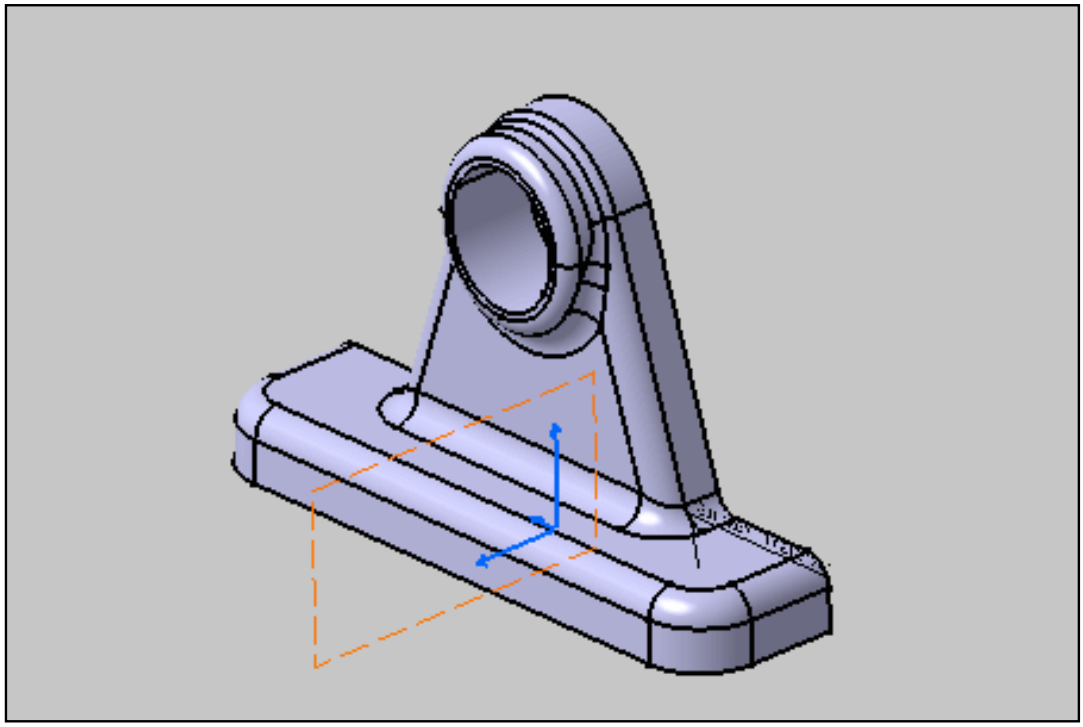
Open the Tolerancing_Annotations_05 CATPart document.

To improve the highlight of the geometry selection, select the **Tools->Options...** command. In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

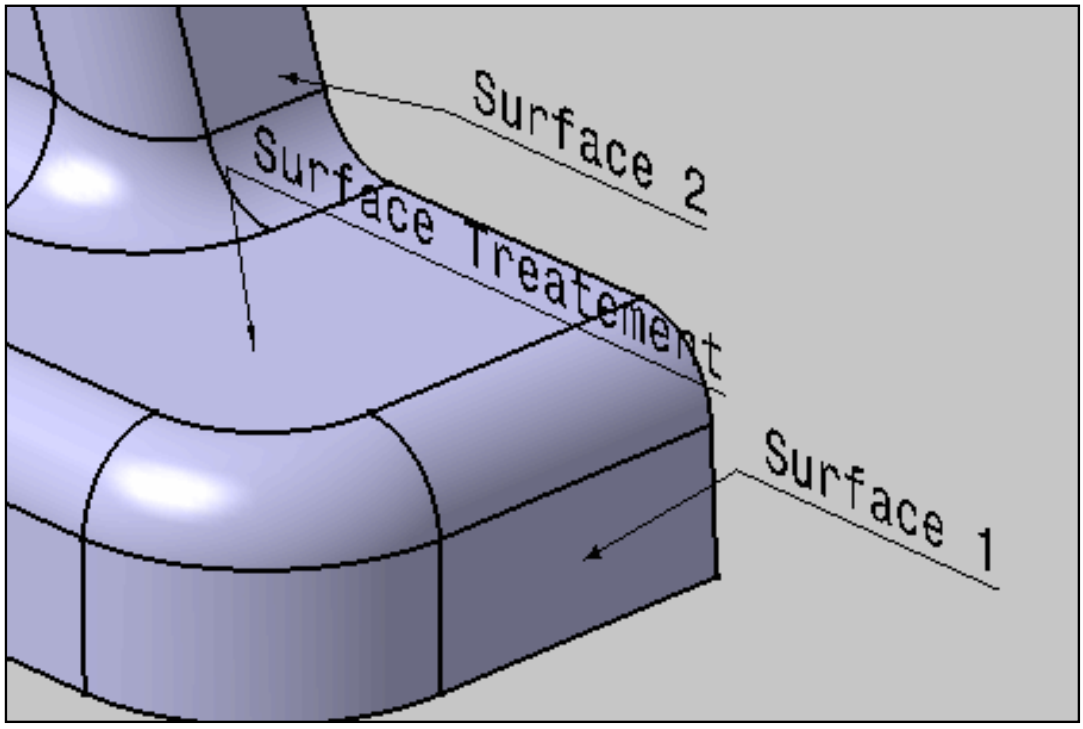


1. Right-click the **All Texts** capture and select **Display Capture** command from the contextual menu.

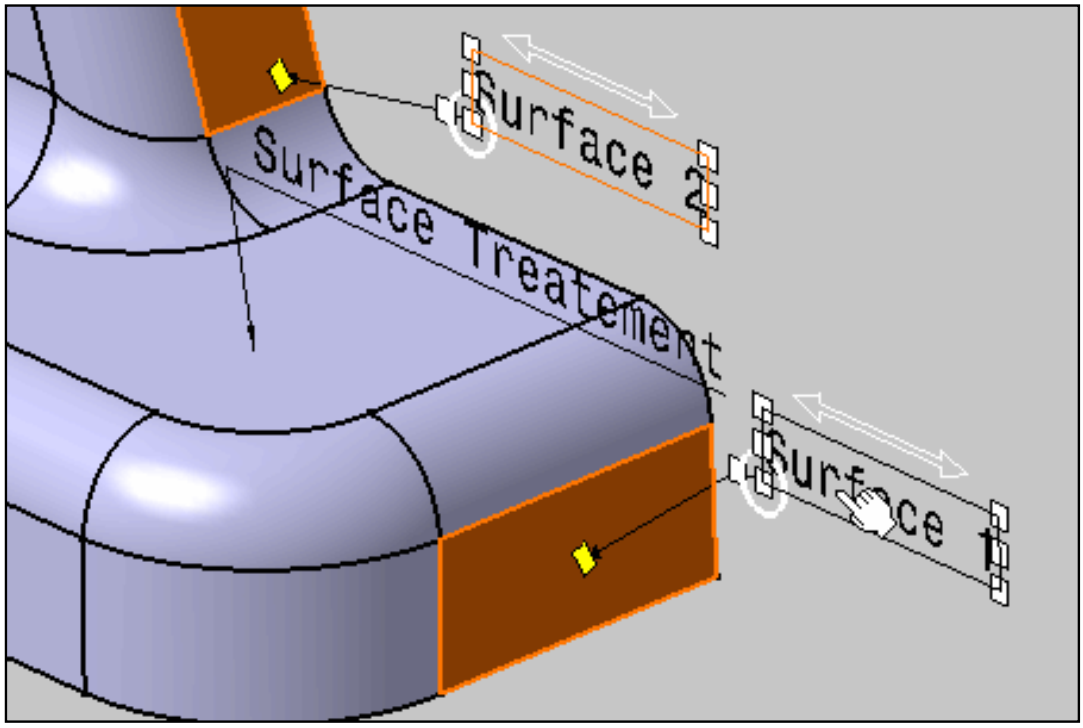
   The geometry is displayed according to the capture options.



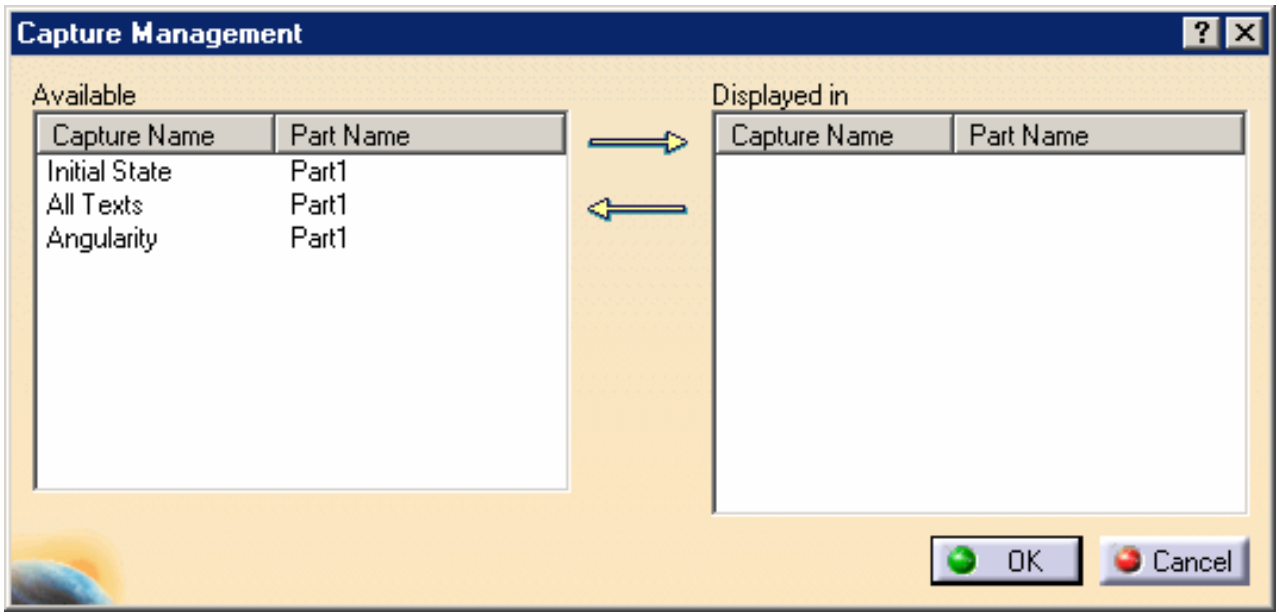
2. Create two new texts (See Creating an Annotation Text).



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.



Back Up

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

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

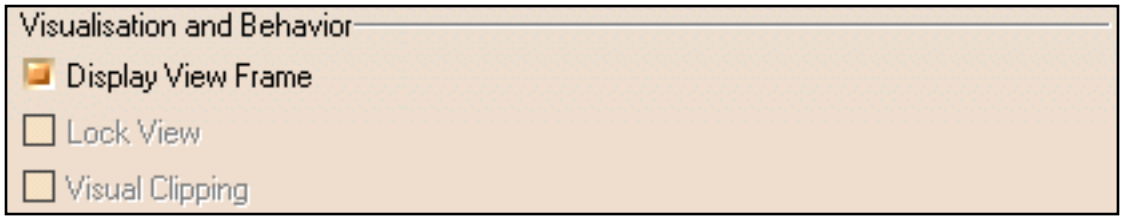
The views/annotation planes are by default displayed in the 3D with a dashed frame that is resized to always frame all the annotations that are linked to it.

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.



Up Next

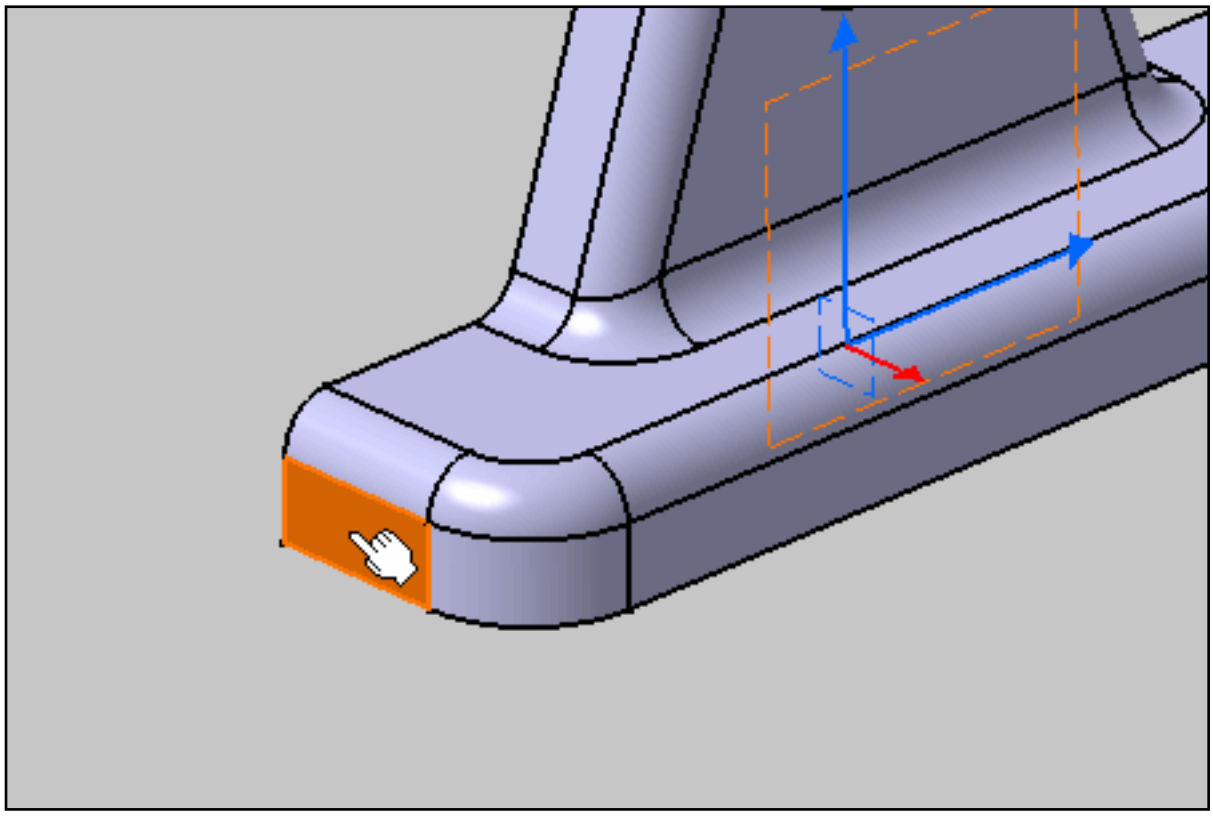
Creating a Projection View/Annotation Plane

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.

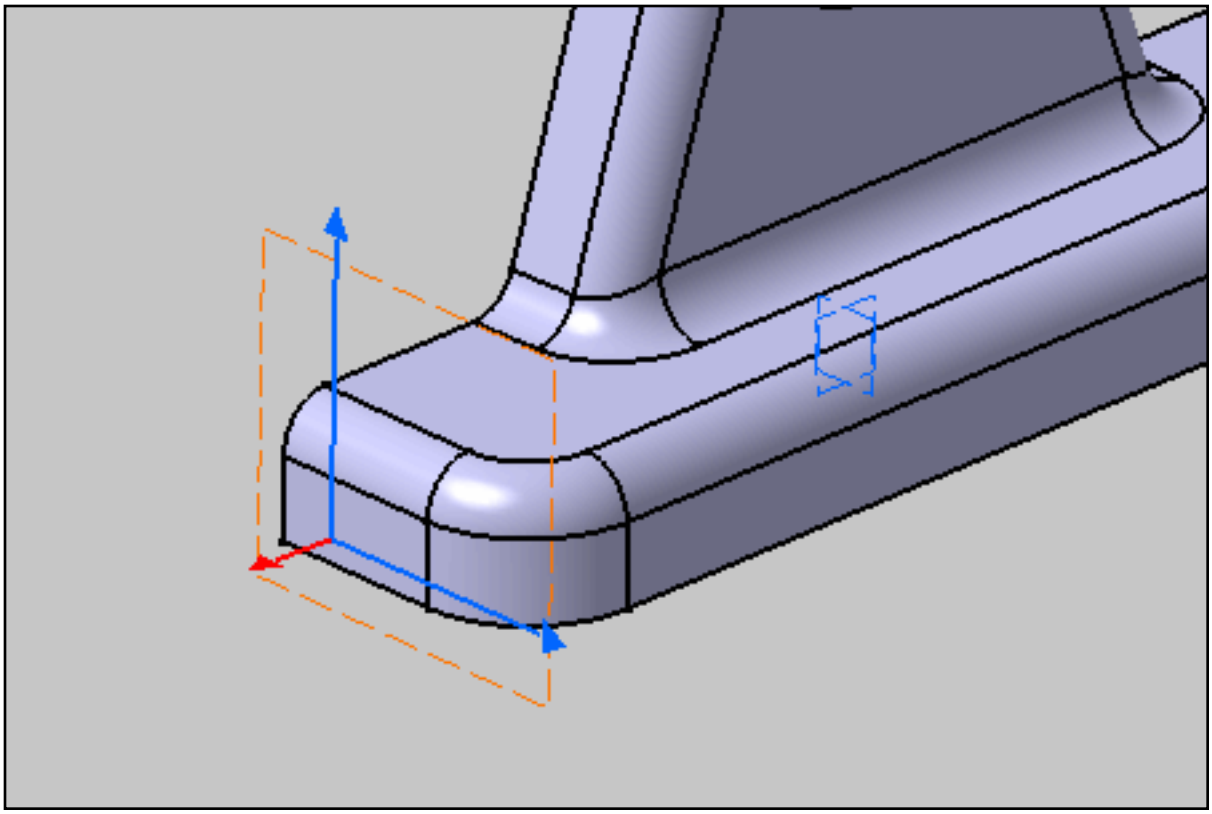
1. Click the **Projection View** icon:

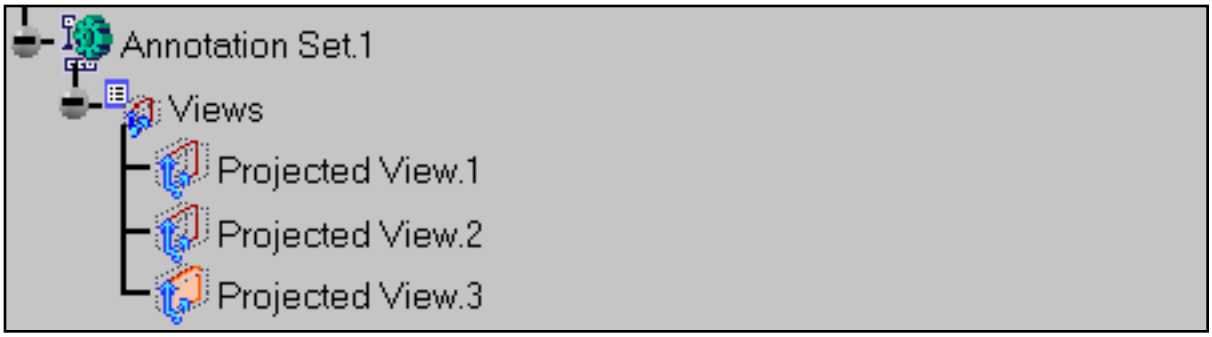
2. Select the face as shown.



You have to select a planar element only to perform this command.

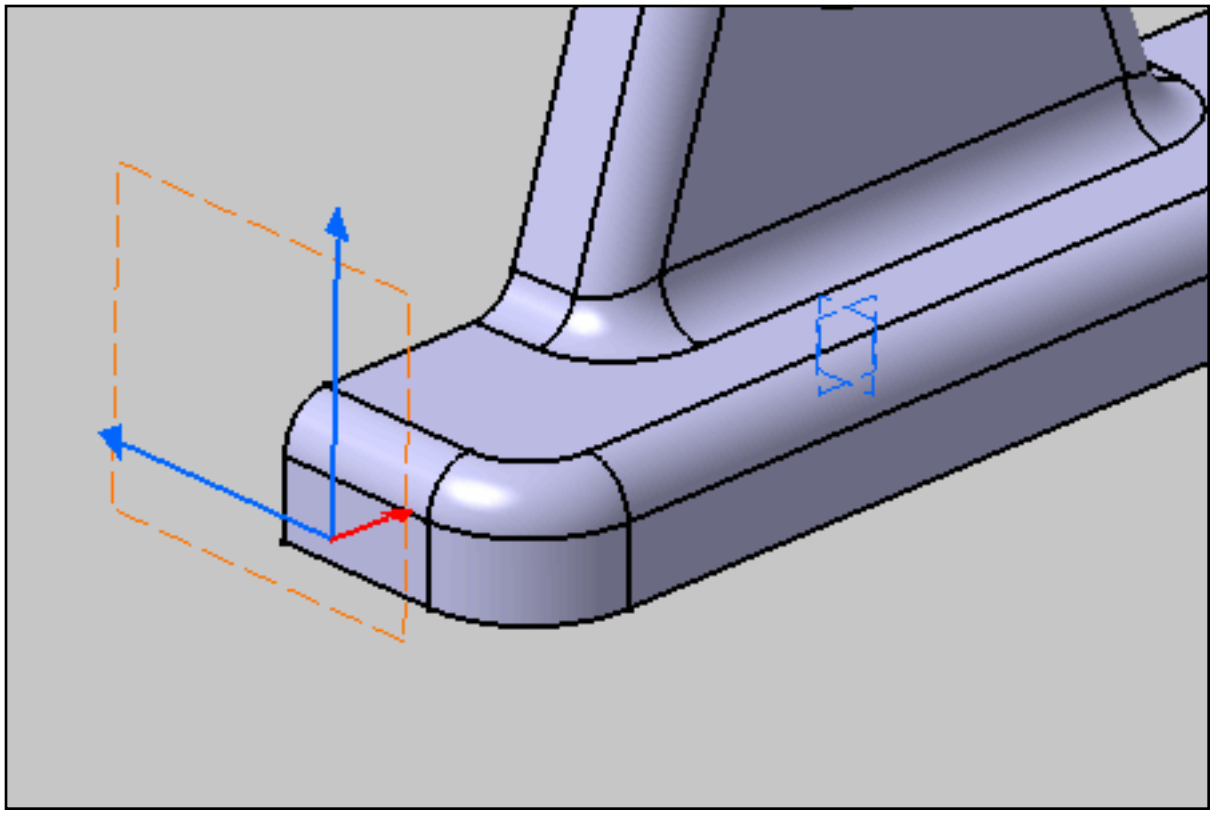
The projection view is created.
Projection views are represented by a blue reference axis, its normal axis is red until you create an annotation, and are identified as **Projection View.3** in the specification tree.





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.



Back Up Next

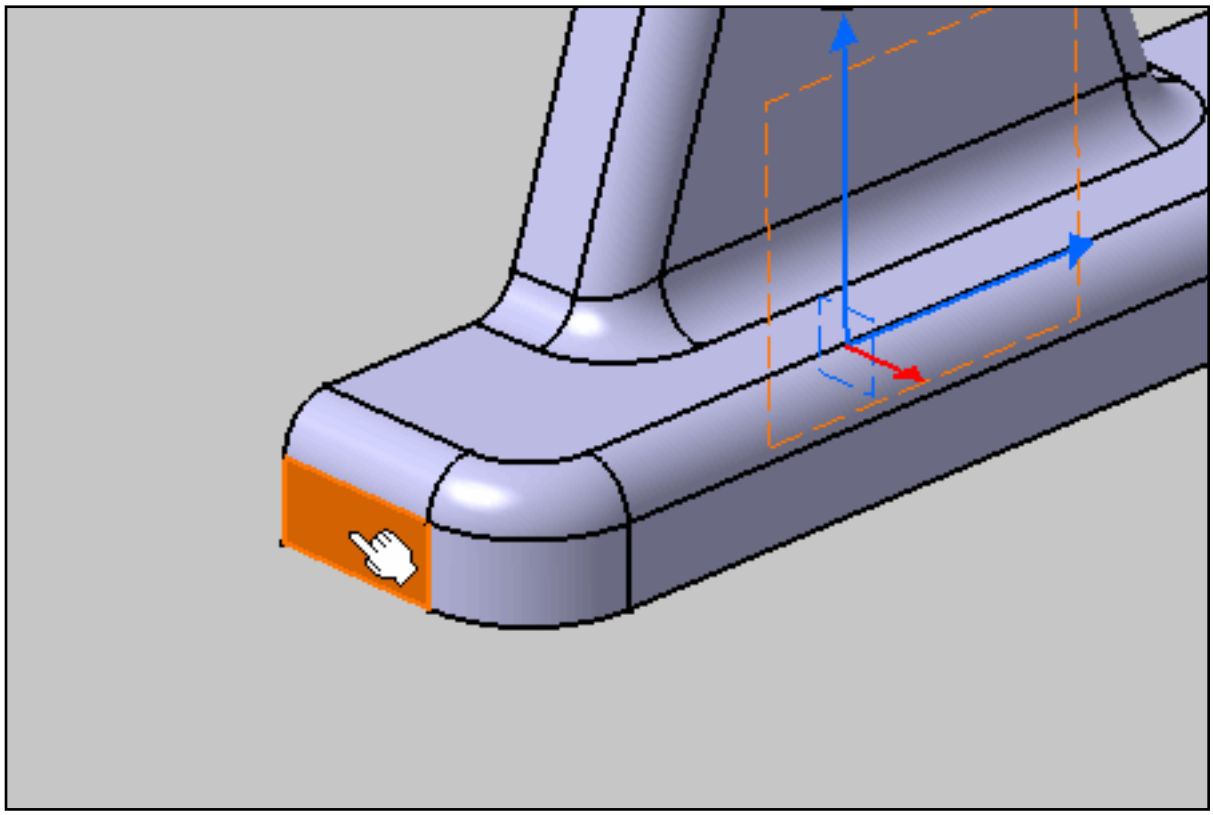
Creating a Section View/Annotation Plane

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.

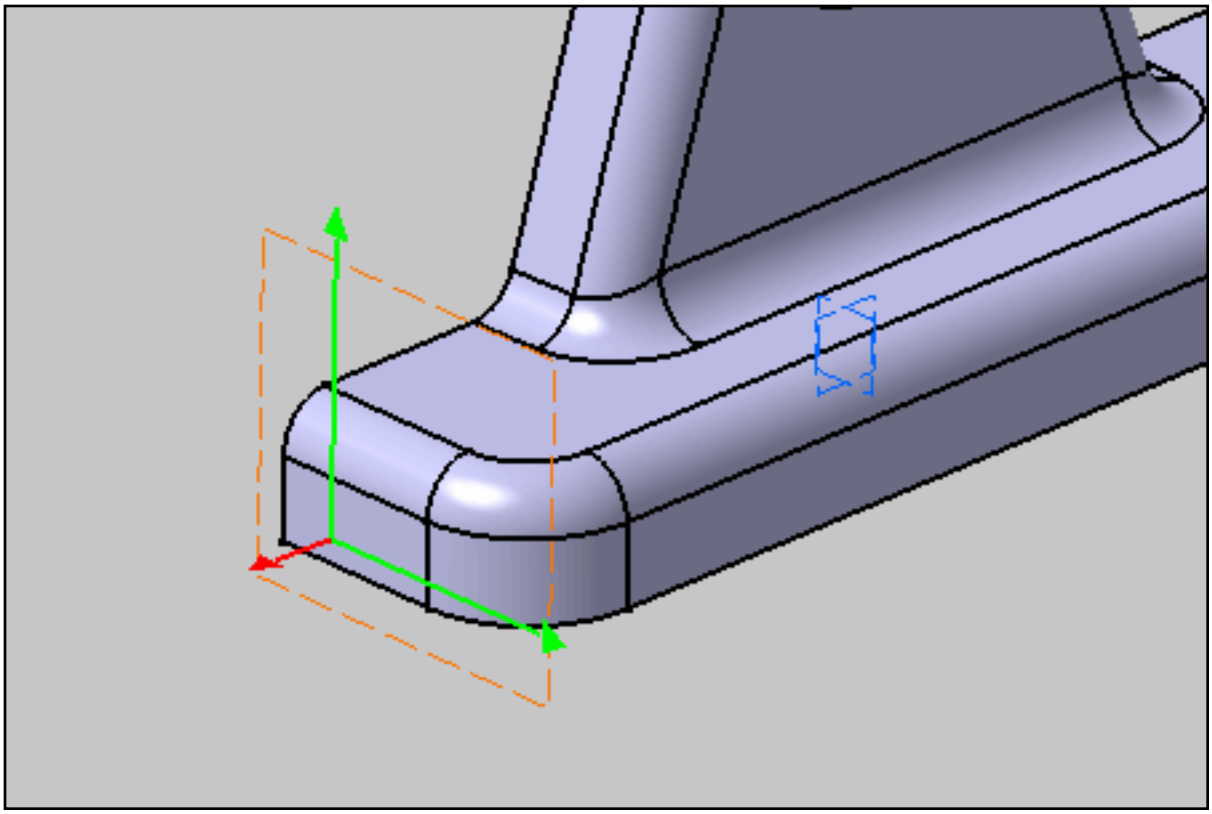
1. Click the **Section View** icon:

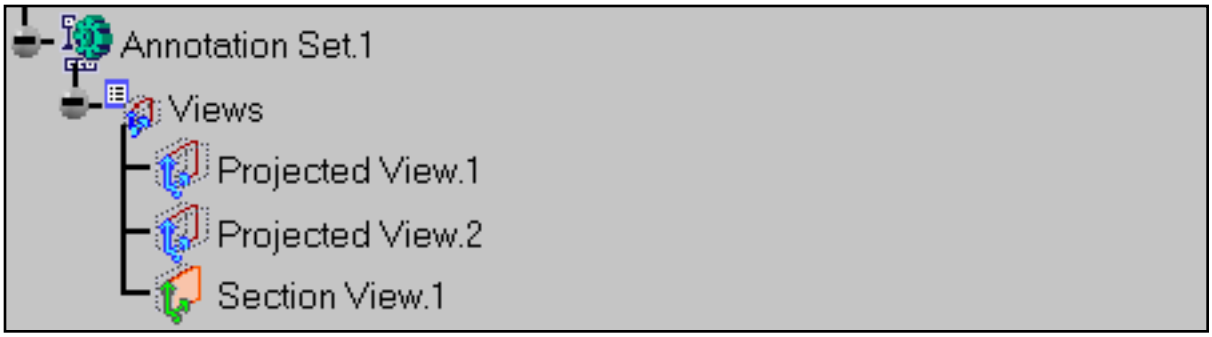
2. Select the face as shown.



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

Back Up Next

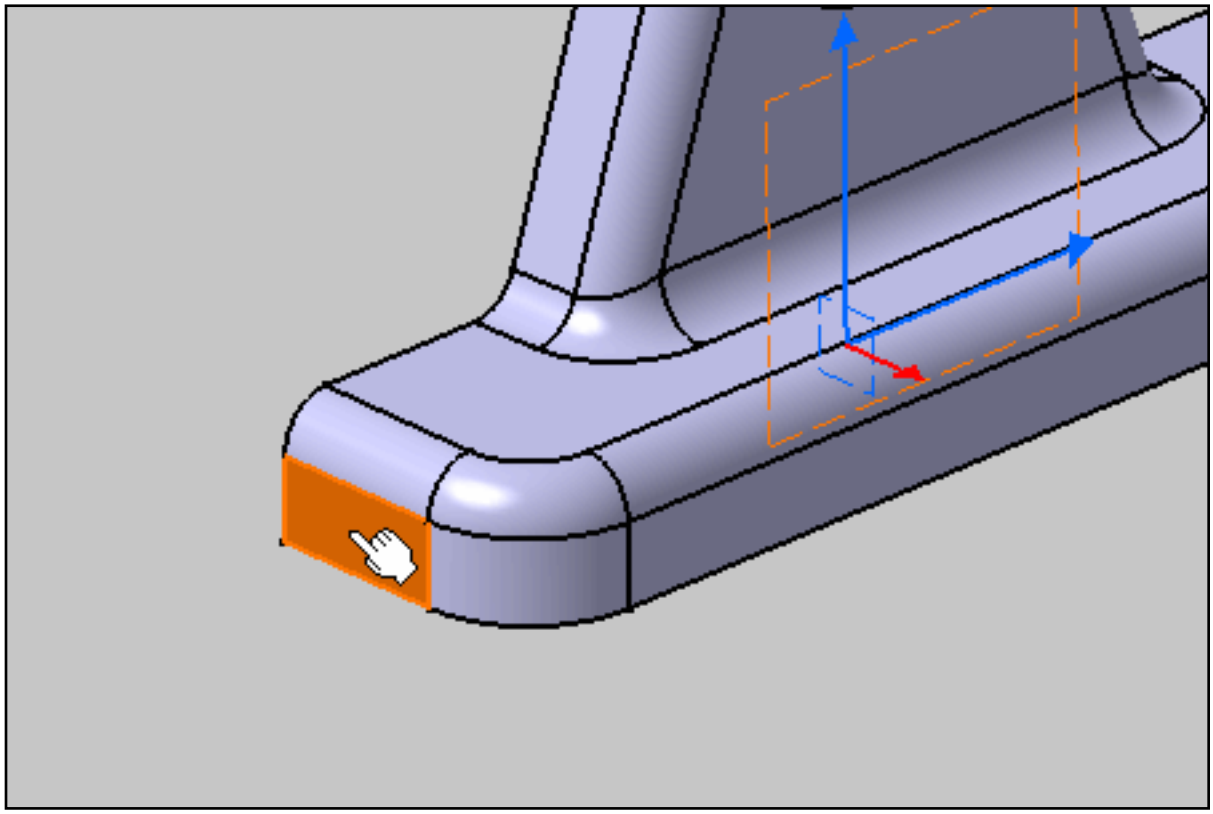
Creating a Section Cut View/Annotation Plane

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.

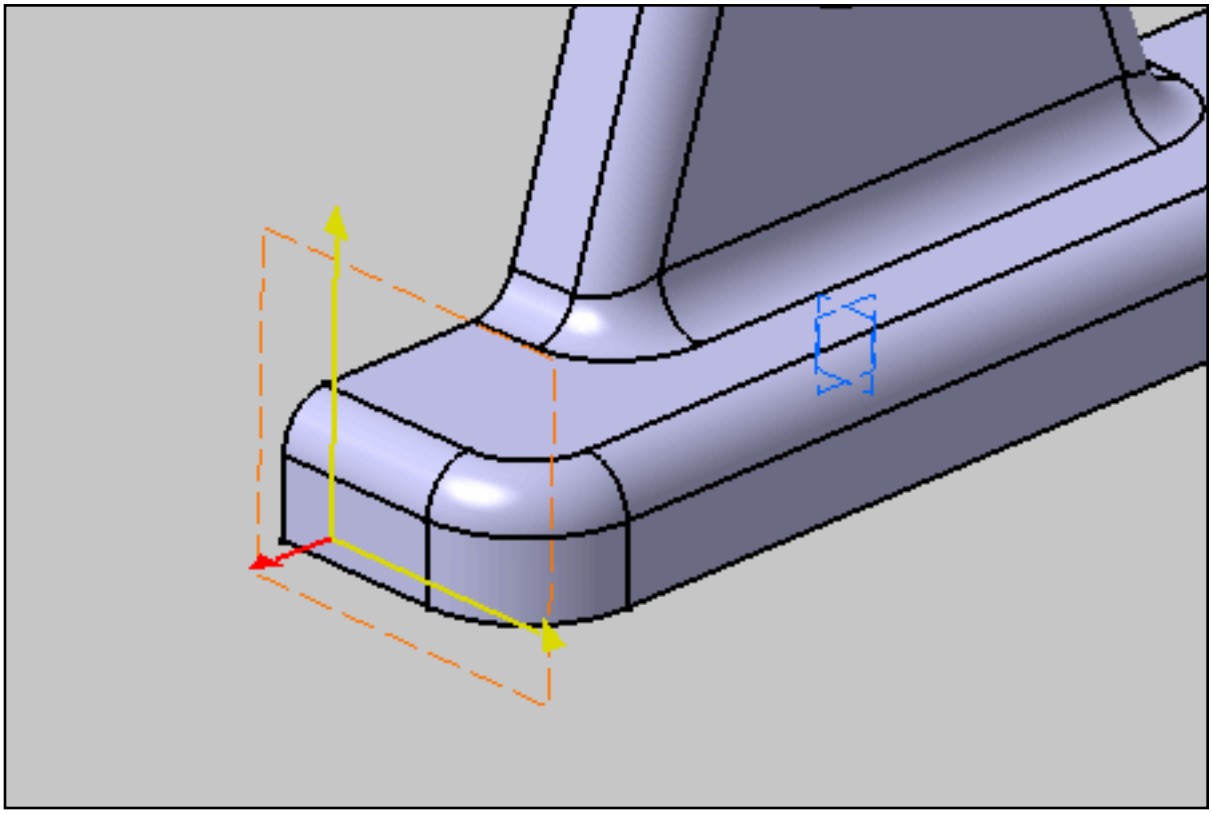
1. Click the **Section Cut View** icon: 

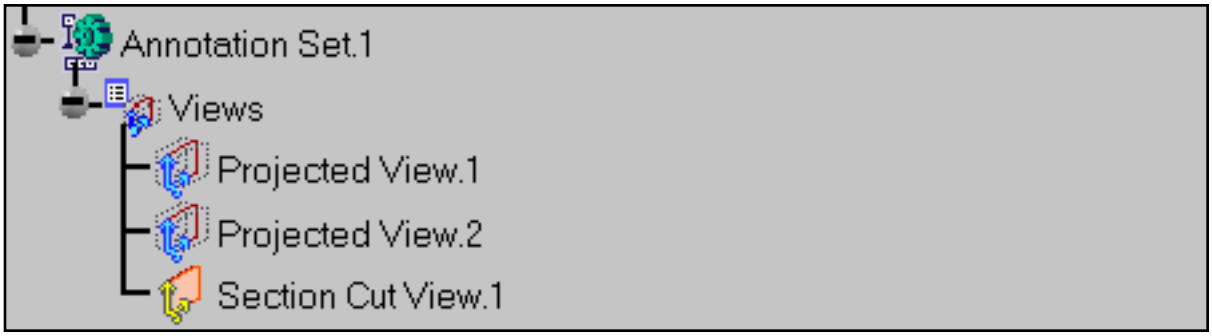
2. Select the face as shown.



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.







Back Up Next

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.

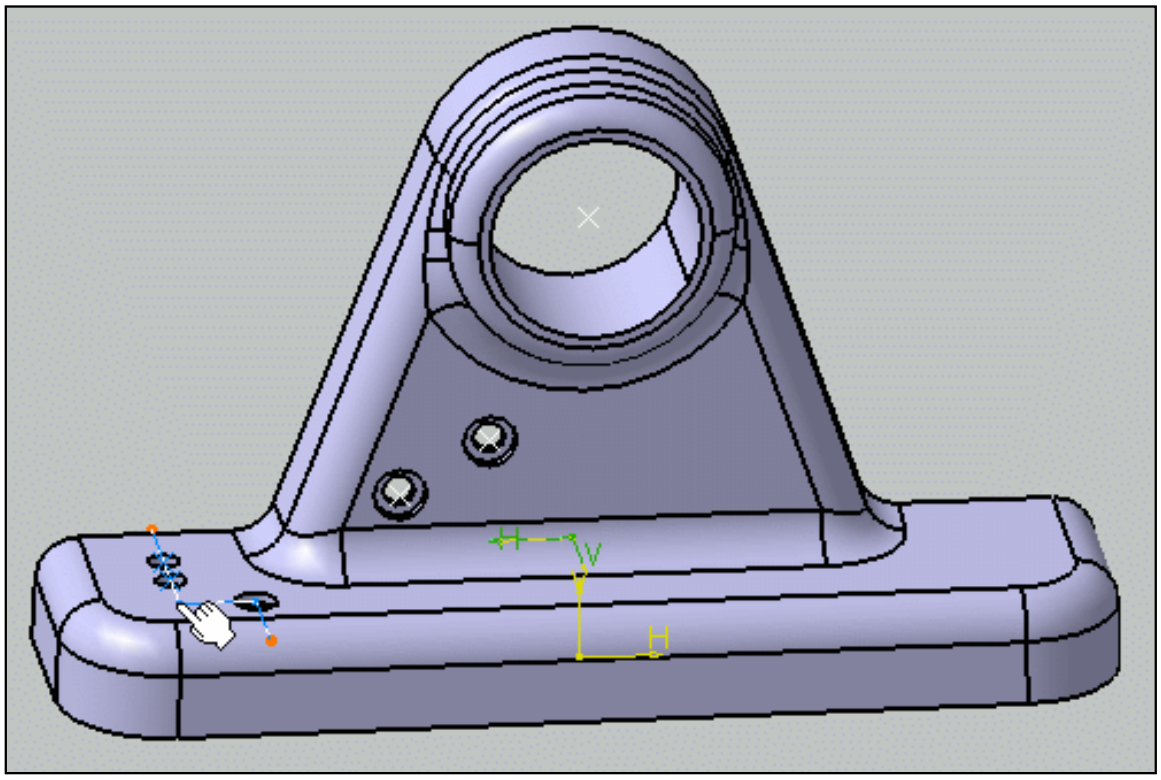
See Using a View for more information.

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.

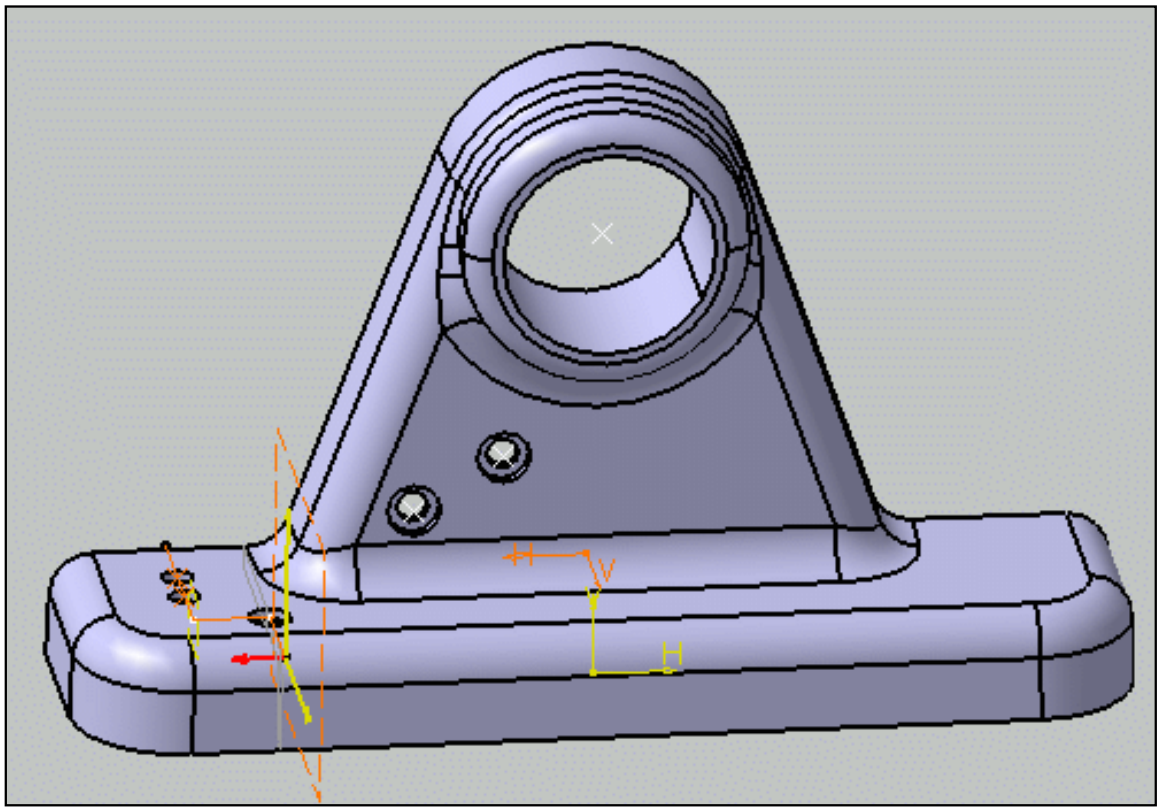
1. Click the **Offset Section View/Section Cut** icon:  The Section View Creation dialog box is displayed.
2. Specify the type of view that you want to create: **Section View** or **Section Cut**. For the purpose of this scenario, select **Section Cut**.
3. Select the profile that will be used as cutting 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.

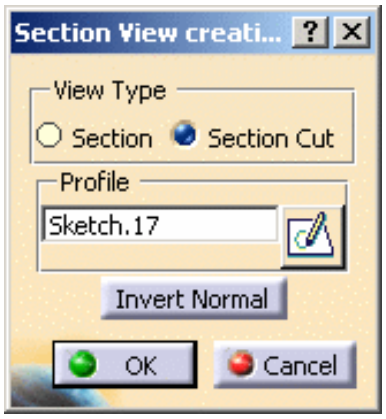


The offset section cut is previewed: it is made up of two distinct section cut views/ annotation planes.

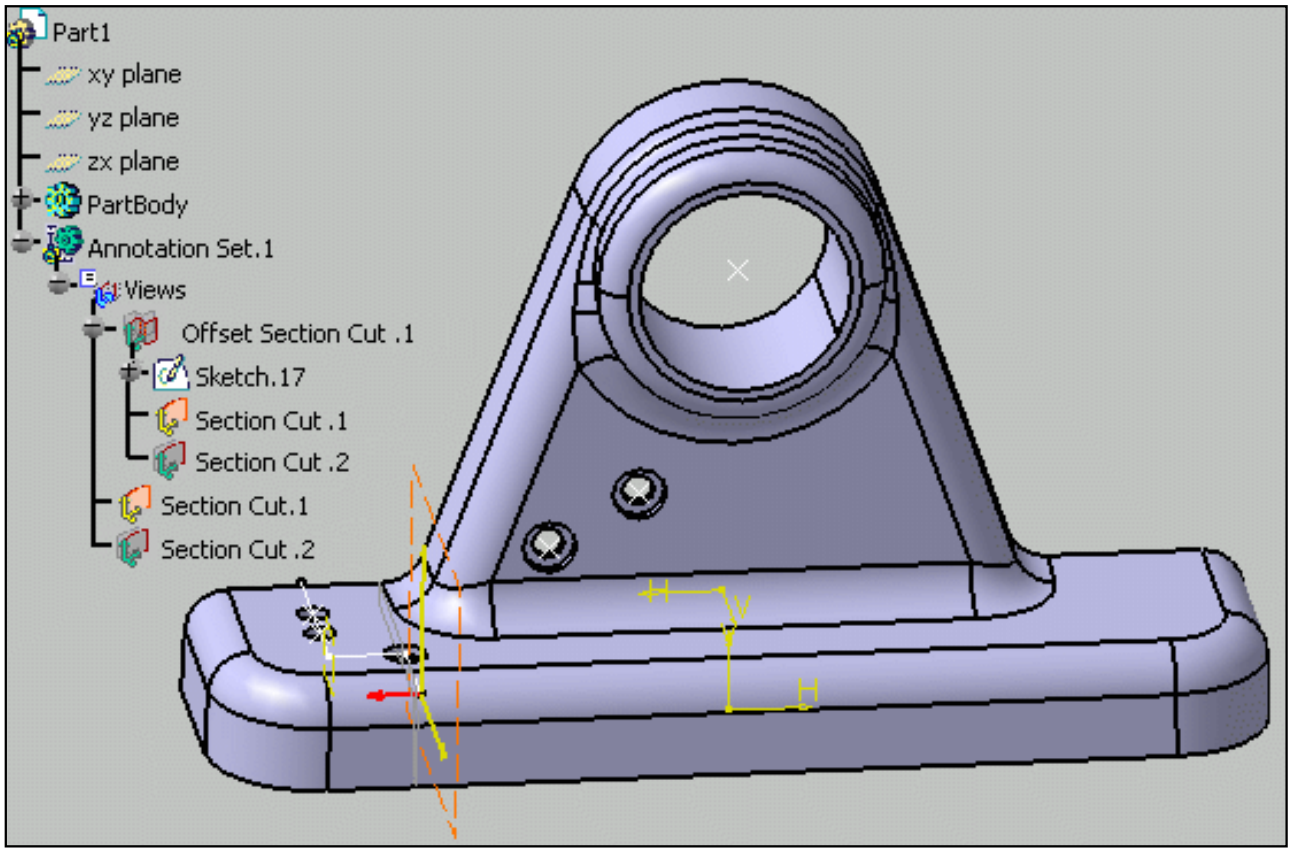
4. Click the **Invert Normal** button to invert the normal of the offset section cut: this actually inverts the normal of the two section cut views/annotation planes that make up the offset section cut.



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.

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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.

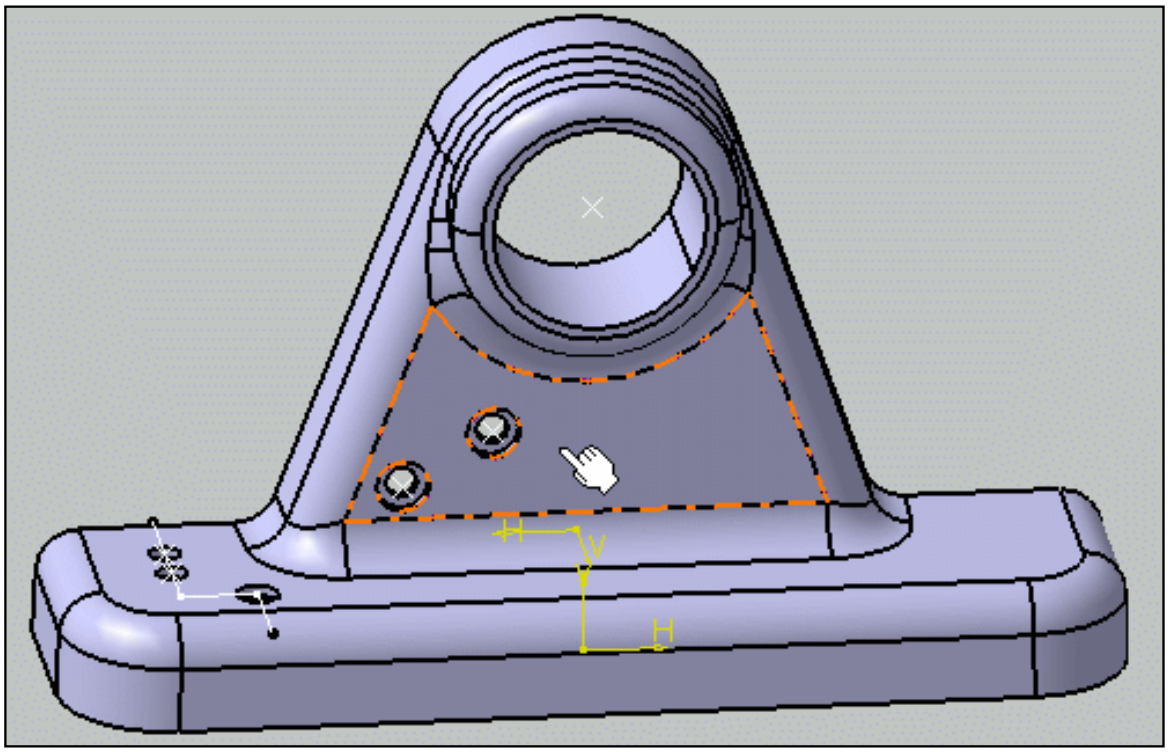
An aligned section view/aligned section cut is created from a cutting profile defined from non parallel planes.

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.

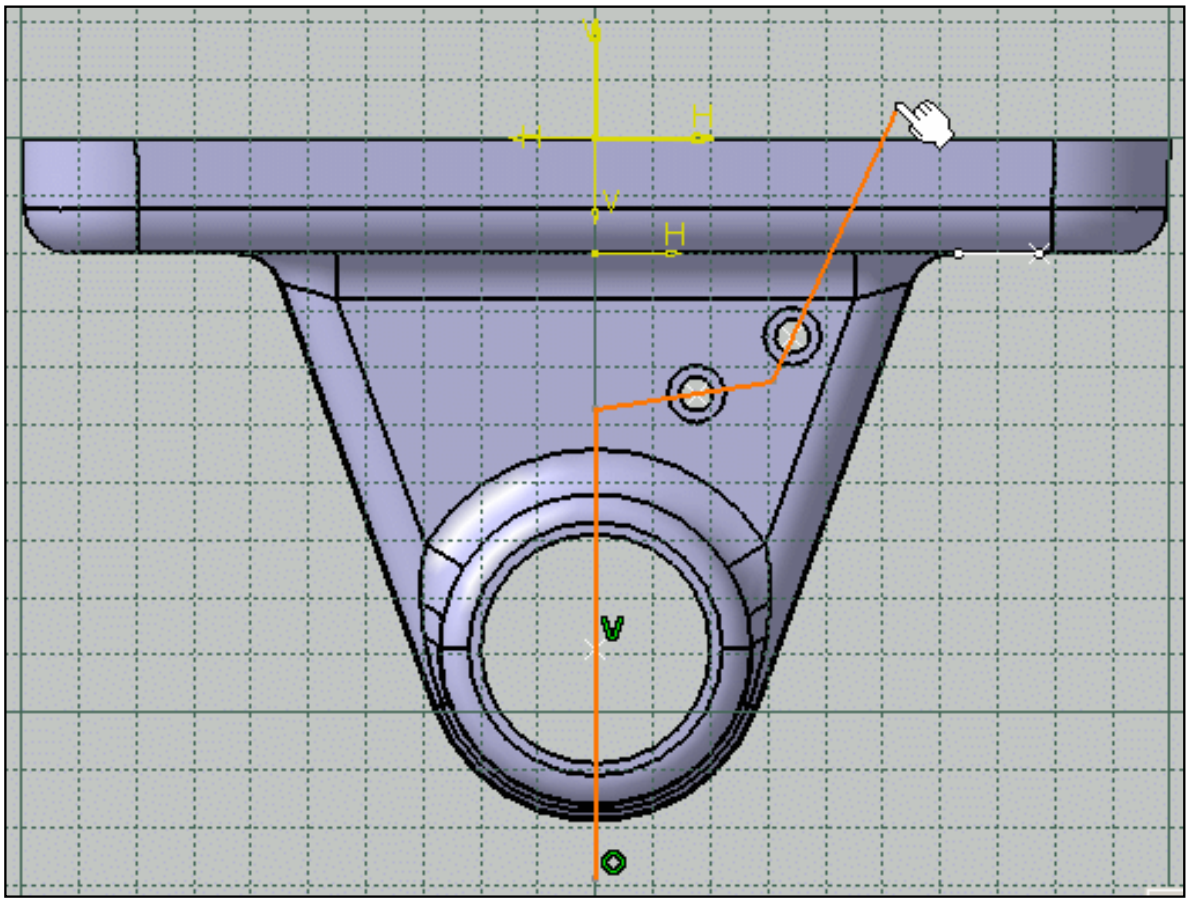
Note that, at this stage, you can also select an existing sketch, if one is available; in this case, you can only select a sketch which is valid for the type of view to be created. For more information on using this method, you can refer to Creating an Offset Section View/Section Cut: the procedure is similar when creating aligned section views/section cuts.

4. Select a planar surface.



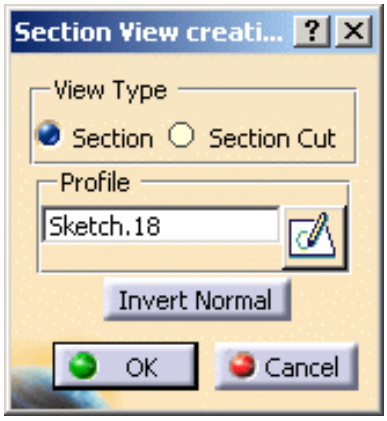
The application switches to the Sketcher workbench to let you sketch the cutting profile.

5. Click the **Profile** icon: 
6. Sketch your cutting profile as shown here.



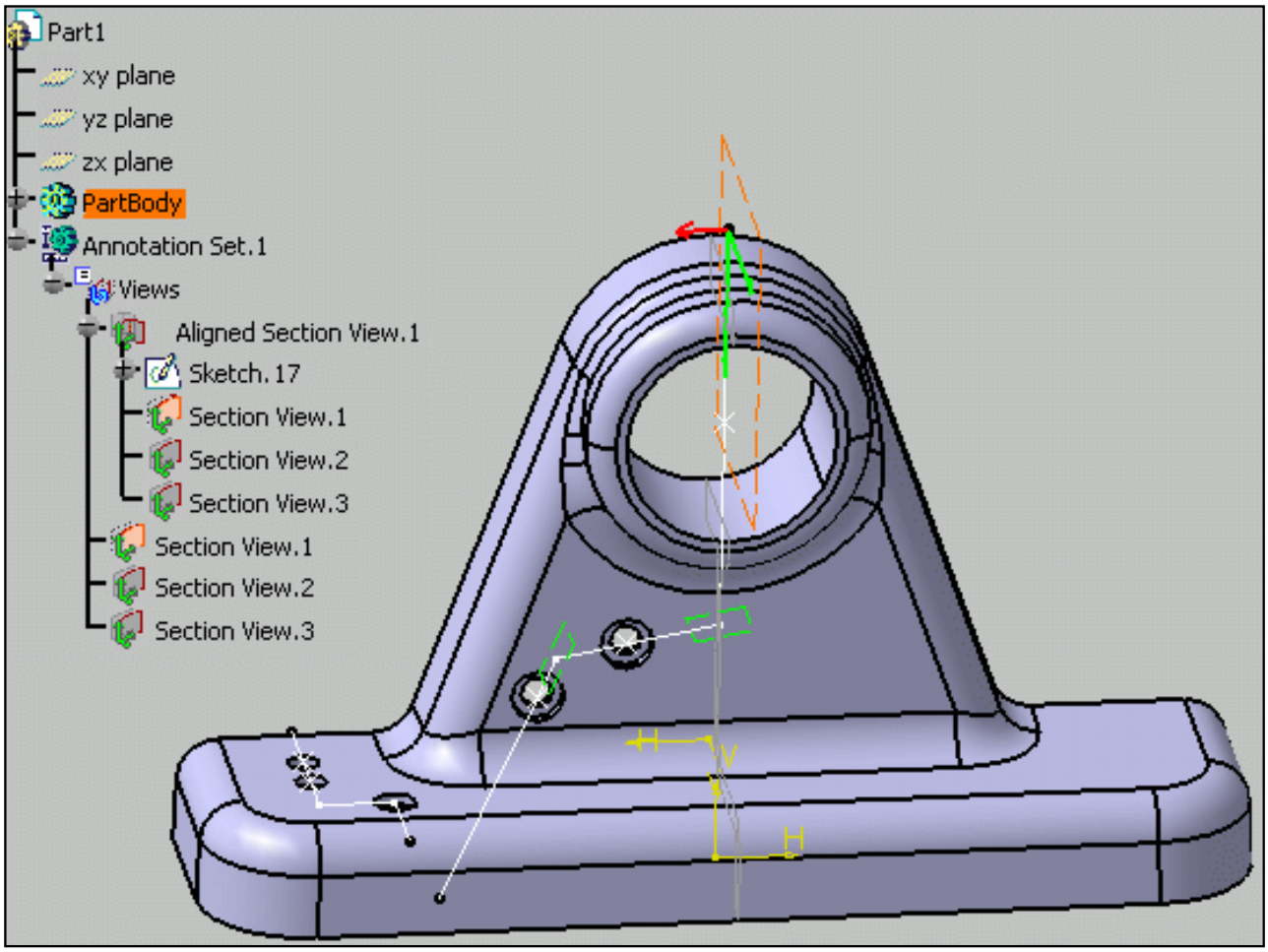
You can constrain the sketch lines in order to ensure their position according to some features of the part.

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.

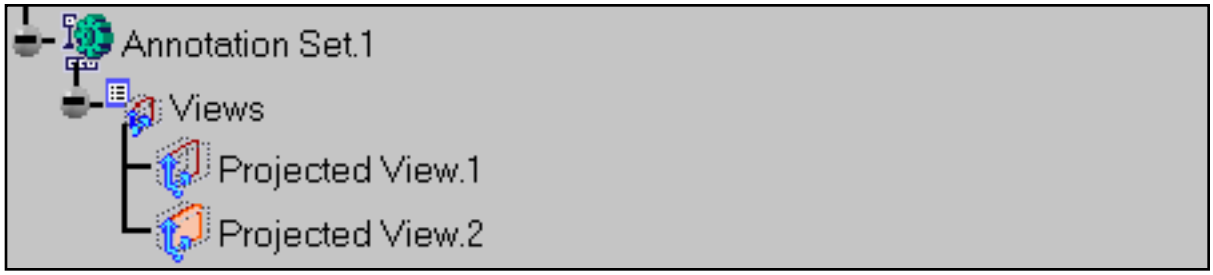


Activating a View/Annotation Plane

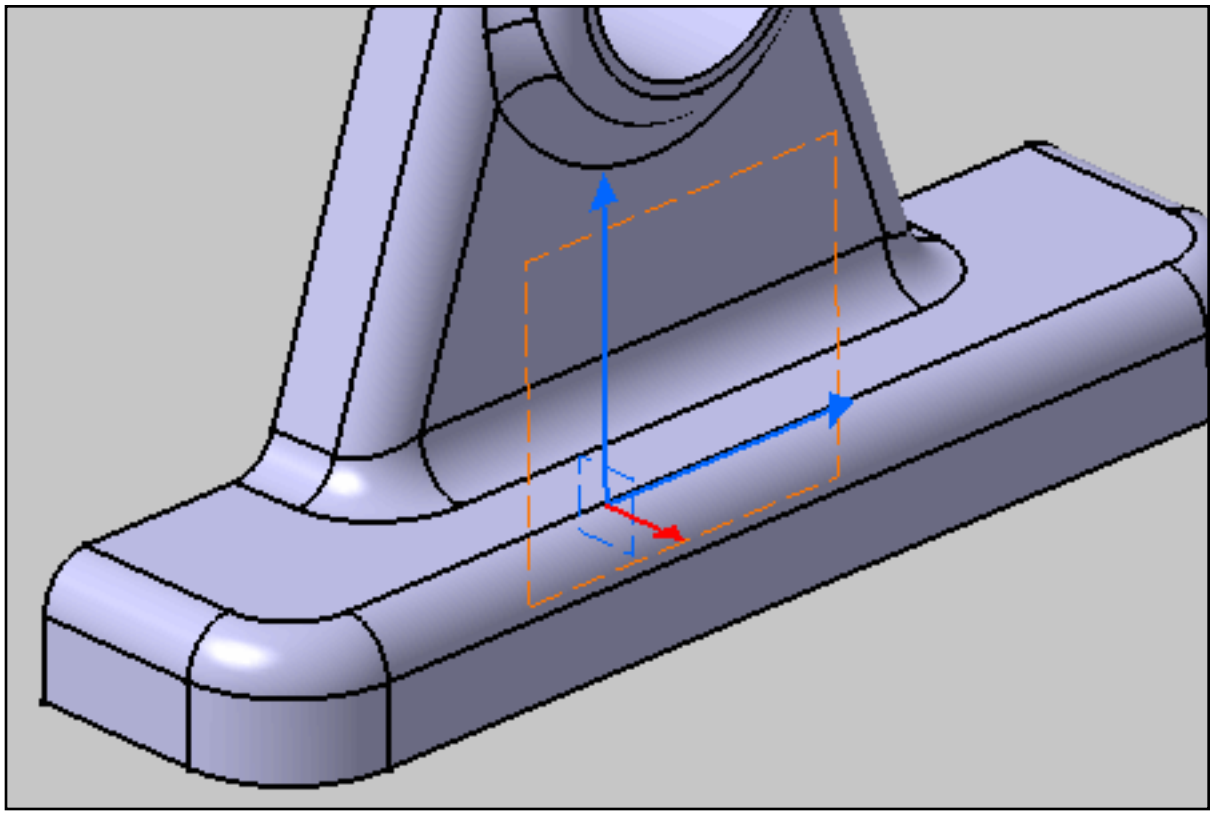
When a tolerancing set includes several annotation planes, to activate a plane, you either double-click it or use the **Activate View** contextual command.

Open the Common_Tolerancing_Annotations_01 CATPart document.

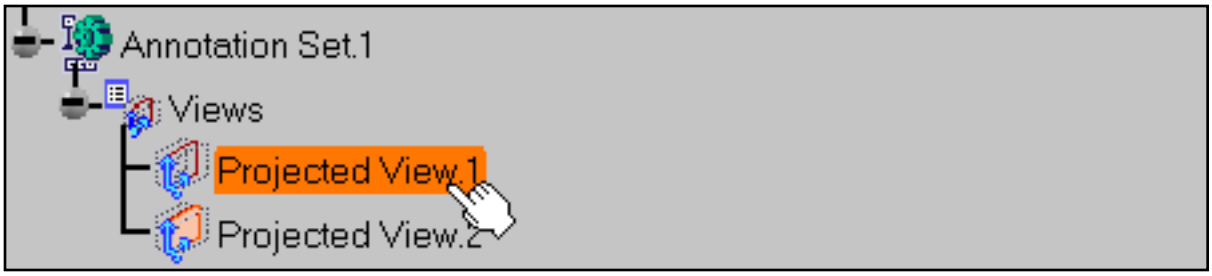
The active annotation plane is orange-colored in the specification tree.



The active annotation plane frame is orange-colored in the geometry and its reference axis appears.



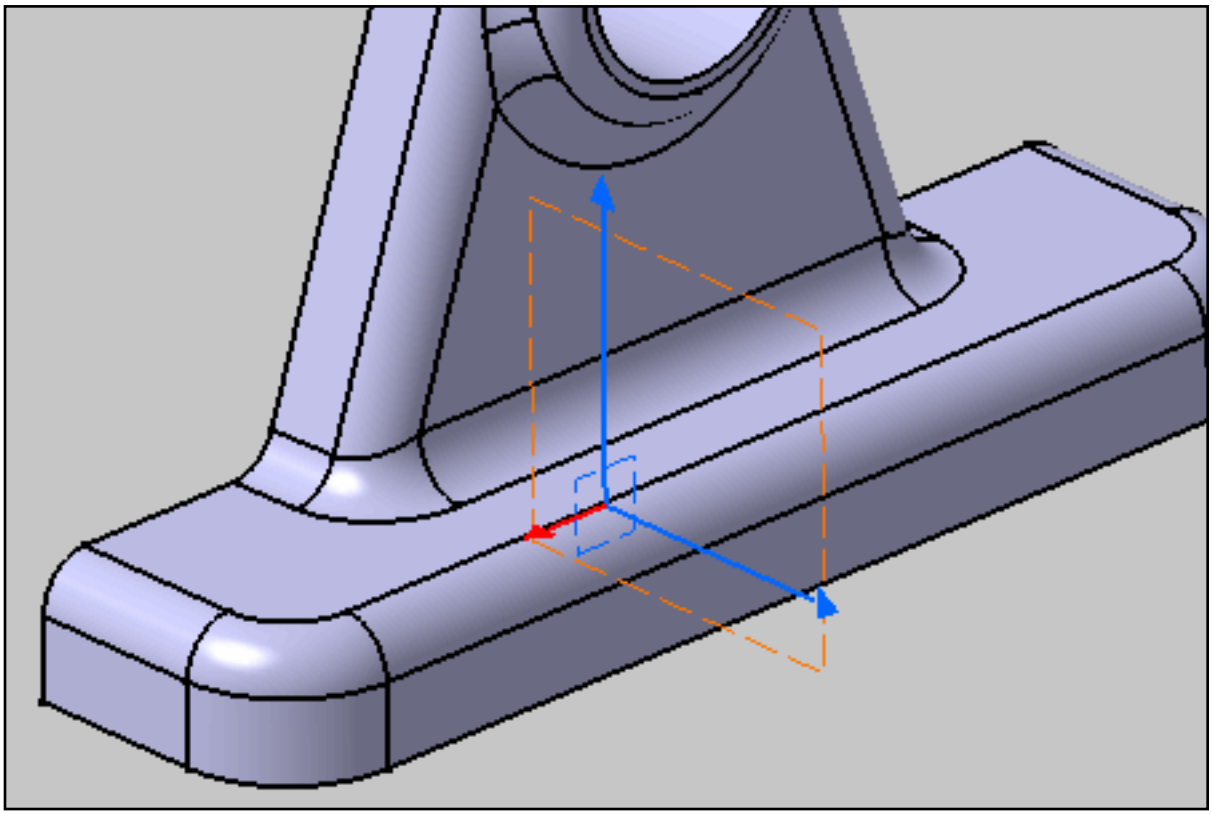
1. Double-click the **Projection View.1** annotation plane.



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.



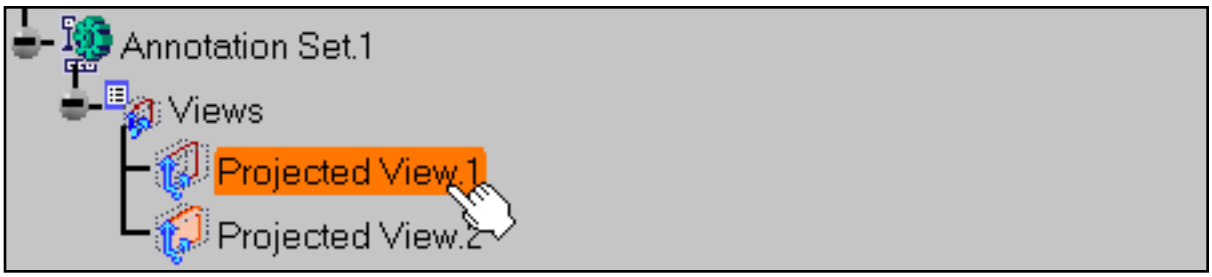
Back Up Next

Editing View/Annotation Plane Properties

This task shows you how to rename an annotation plane and hide its frame.

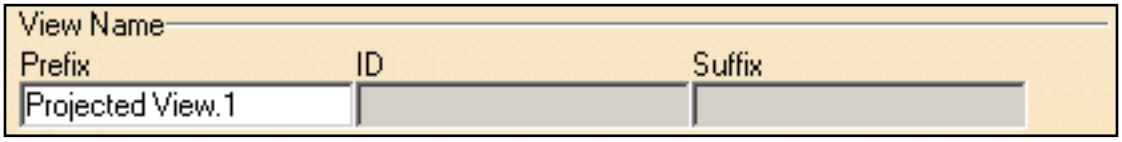
Open the Common_Tolerancing_Annotations_01 CATPart document.

1. Right-click the **Projection View.1** annotation plane and select the **Properties** contextual command.

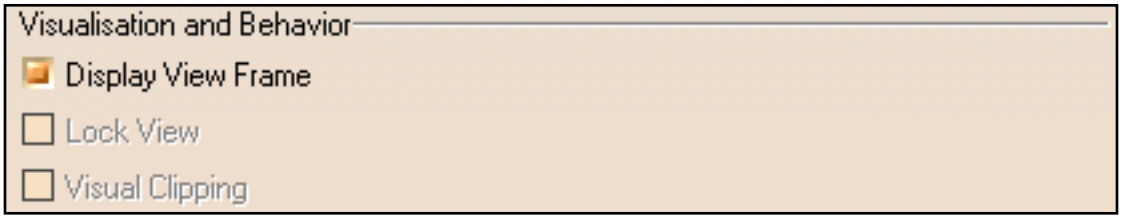


2. Select the **View** tab in the **Properties** dialog box which is displayed.

   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.

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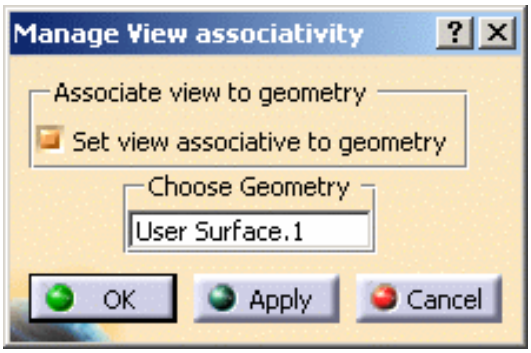
Managing View/Annotation Plane Associativity

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.

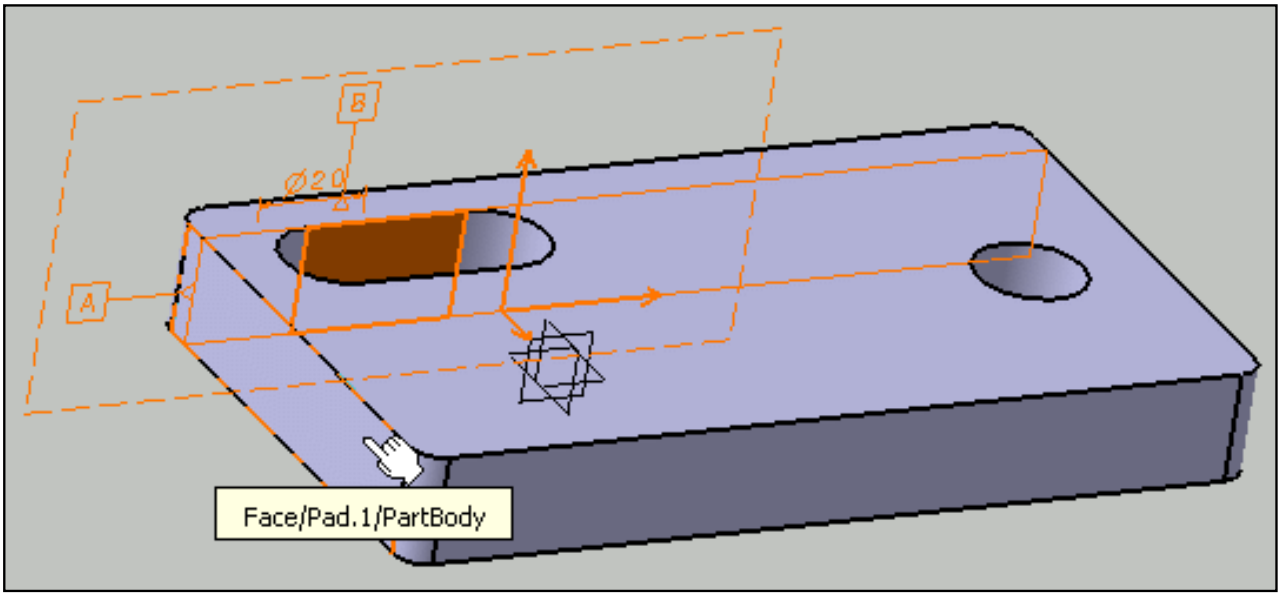
Open the Common_Tolerancing_Annotations_02 CATPart document.

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.

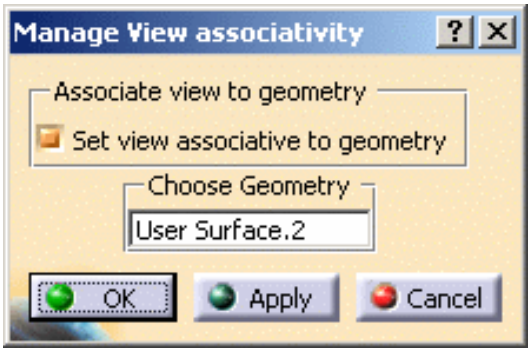


2. You have two possibilities:
   - If you want to disassociate the view from the geometry, uncheck the **Set view associative to geometry** field. In this case, you will then be able to modify the geometry or the axis system without changing the view definition.
   - If you want to associate the view to another geometry, select a planar face or an axis system.

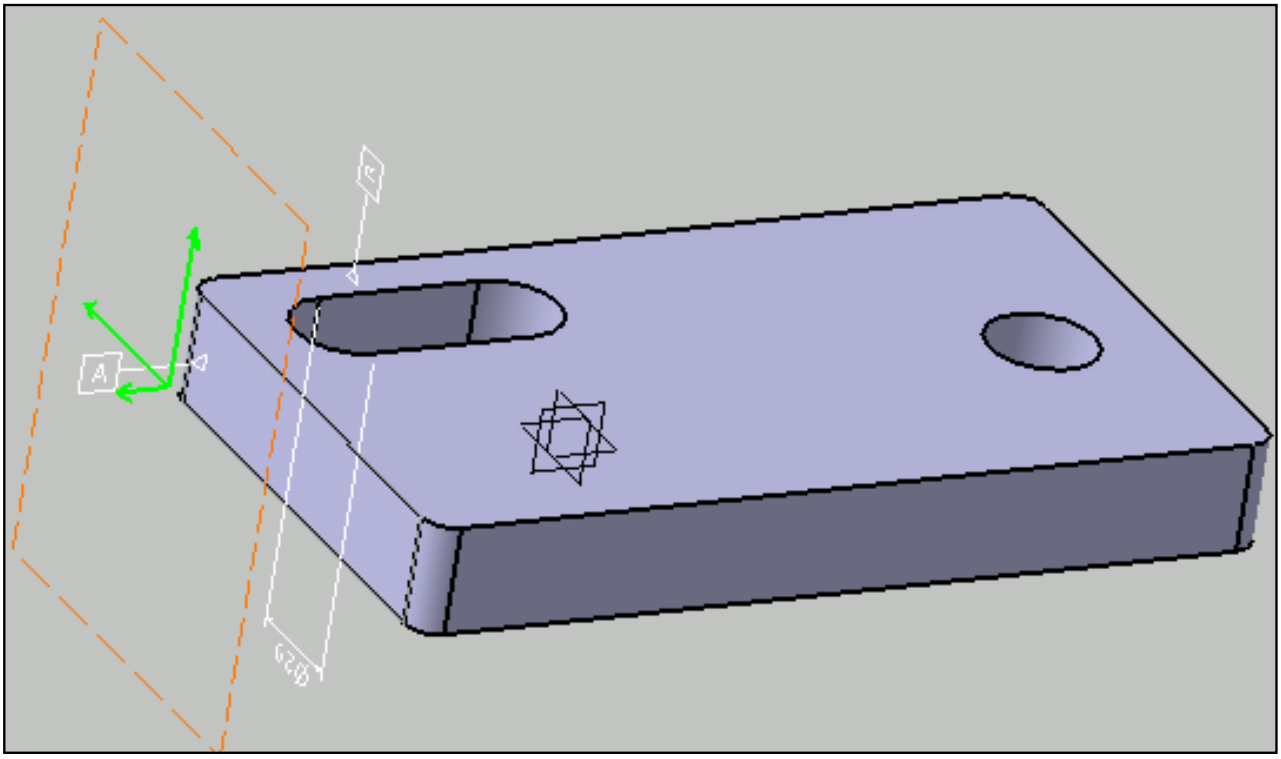
   For the purpose of this scenario, select the face as shown below to associate the view to another geometry.



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.

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Advanced Tasks

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

Creating Note Object Attribute
Managing Annotation Connection
Re-specifying Geometry Canonicity
Reporting Annotations
Annotation Associativity
Managing Power Copies
Providing Constructed Geometry for 3D Annotations

Creating Note Object Attribute

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.

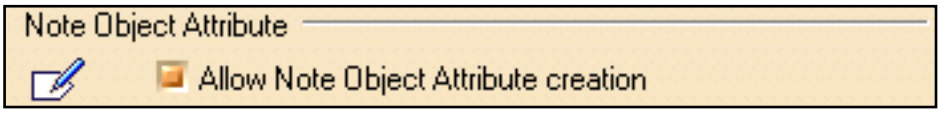
Up Next

Note Object Attribute From a Text

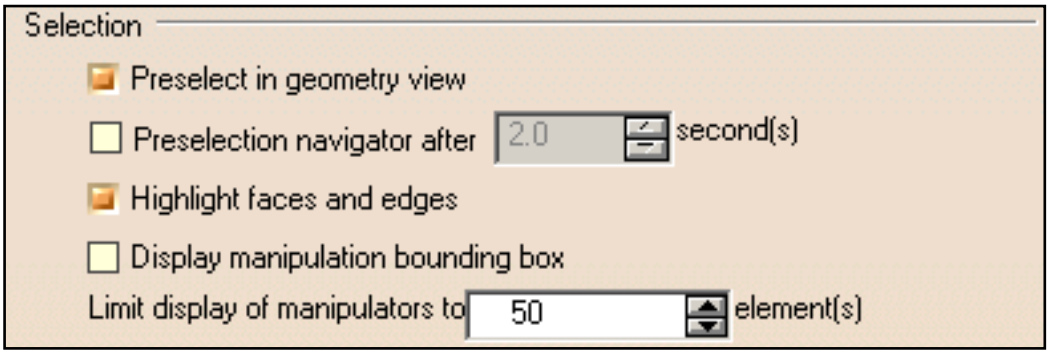
This task shows you how to create a Note Object Attribute from a text.
See Instantiating a Note Object Attribute task and Note Object Attribute concept.

Open the Tolerancing_Annotations_01 CATPart document.

Check the option allowing you to create a Note Object Attribute.
See Tolerancing setting.



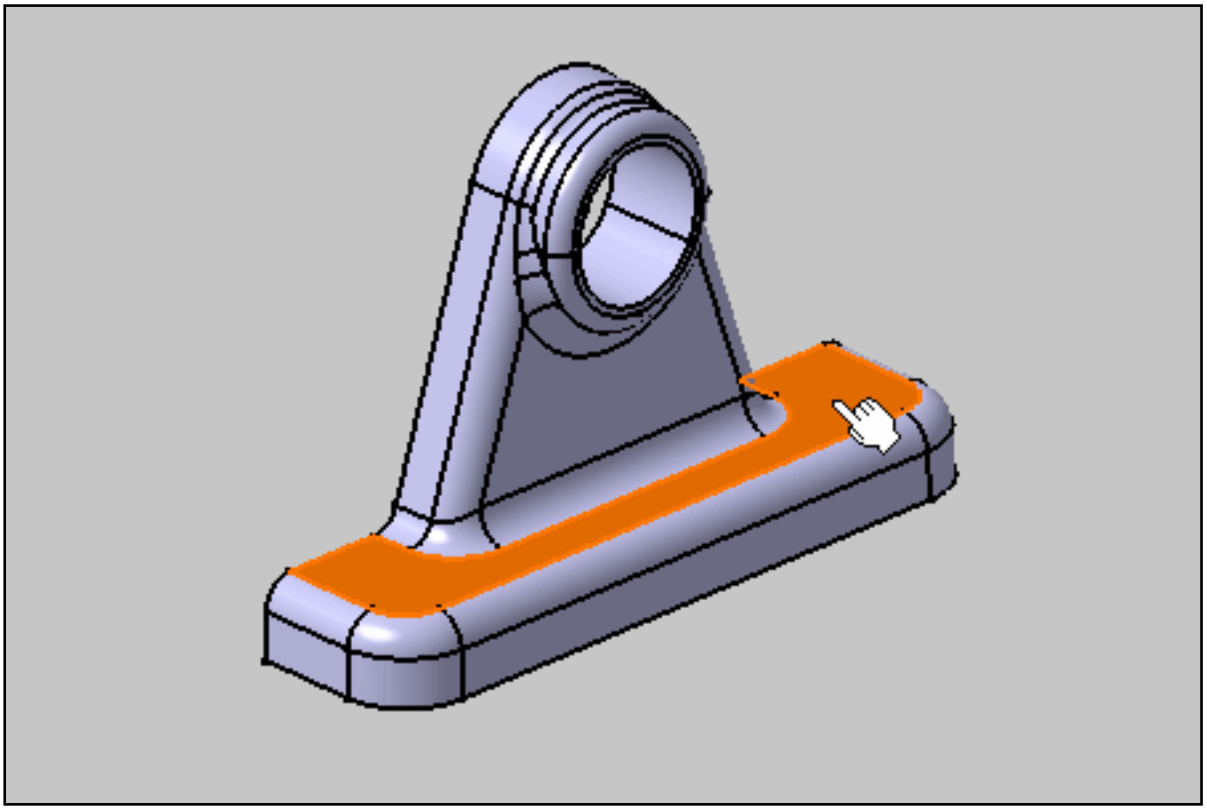
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



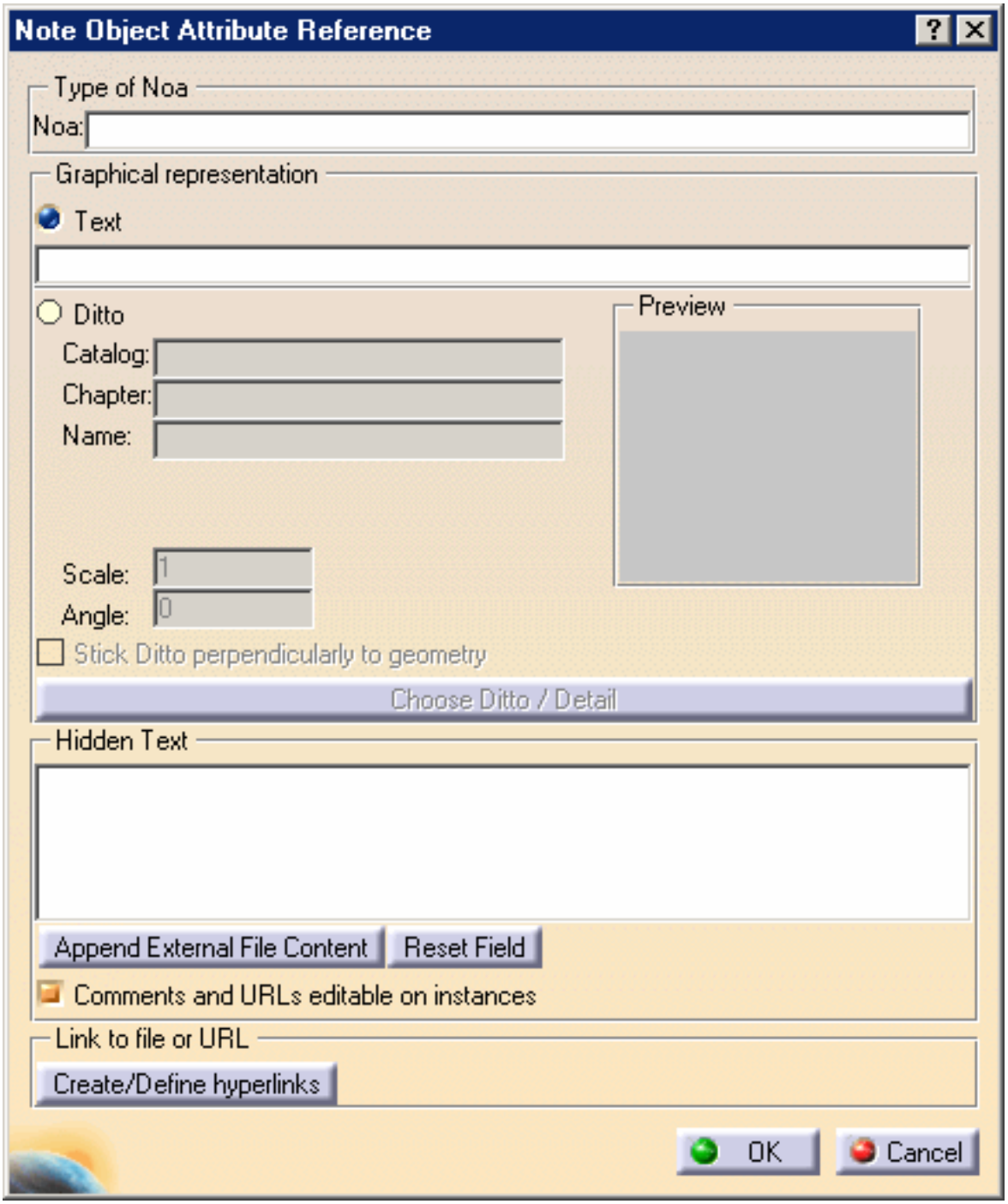
1. Click the **Note Object Attribute** icon: 

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.



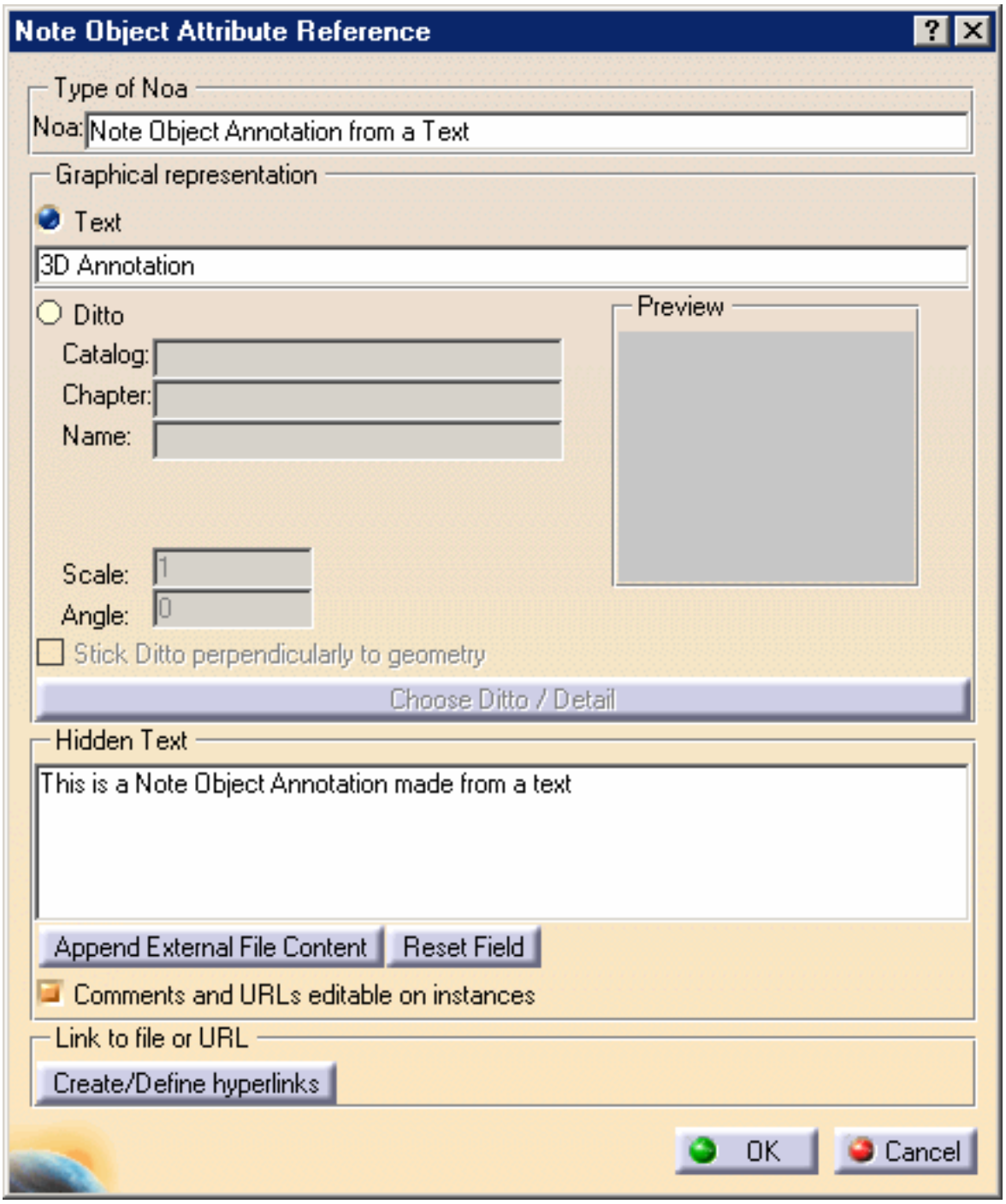
The **Note Object Attribute Reference** dialog box appears.



3. Select **Text** in **Graphical representation** and enter the following texts:

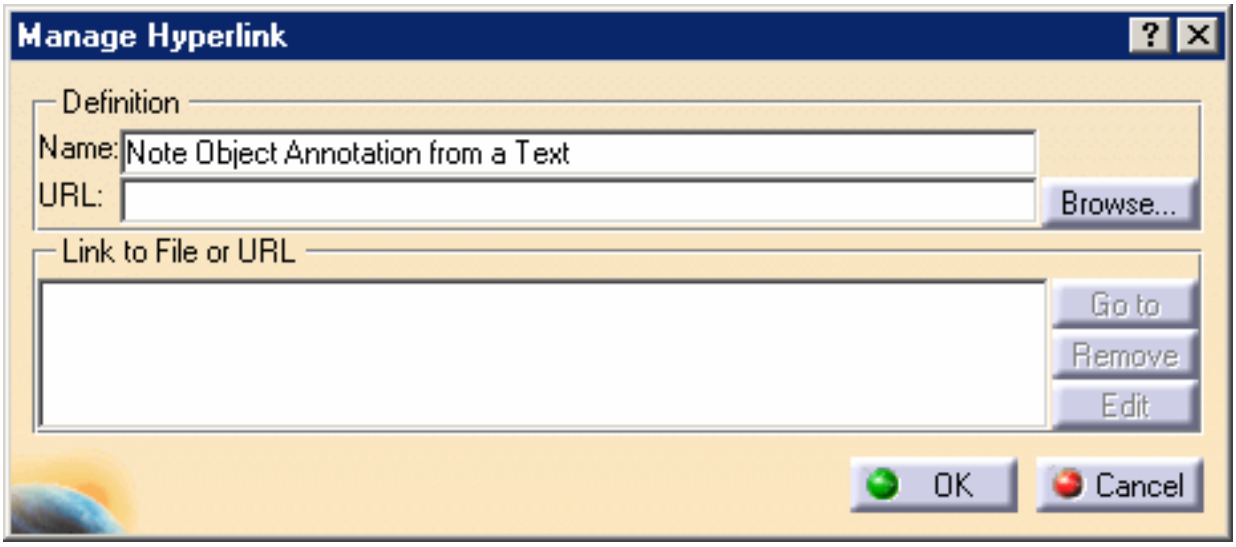
   - The type of the Note Object Attribute: **Note Object Annotation from a Text**
   - The text: **3D Annotation**
   - A hidden text: **This is a Note Object Annotation made from a text**

The **Comments and URLs editable on instances** check button allows user to modify the **Hidden Text** data during Note Object Attribute instantiation or modification.



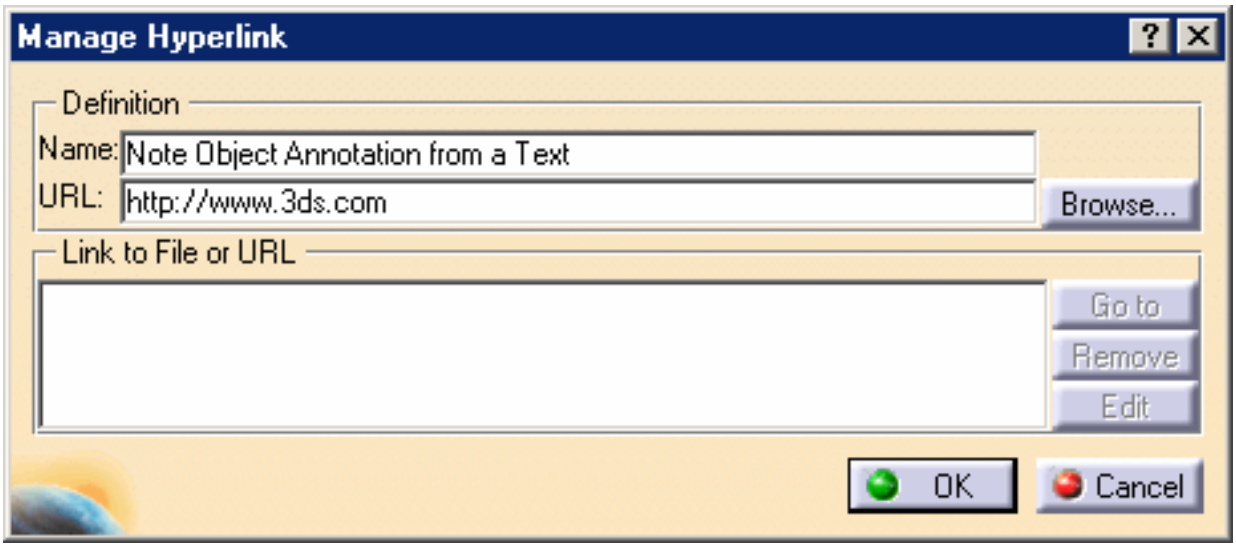
4. Click **Create / Define hyperlinks**.

The **Manage Hyperlink** dialog box appears.



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

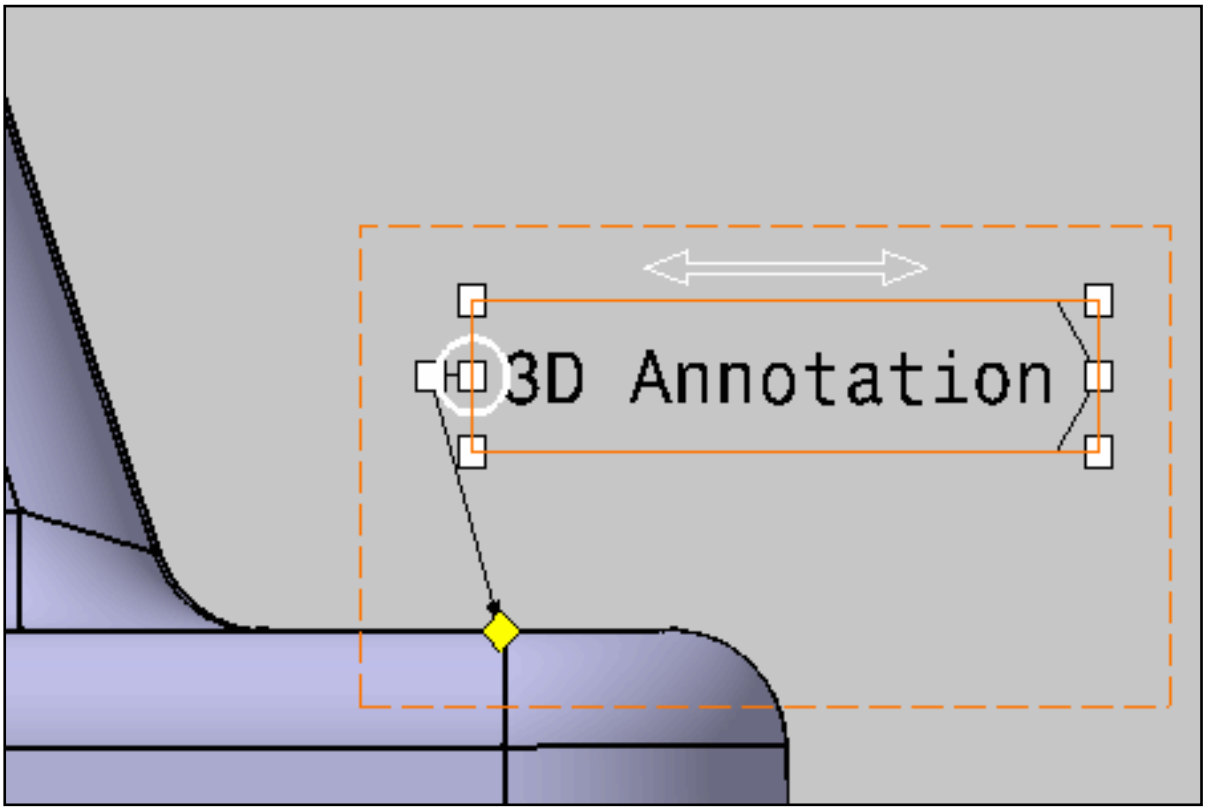


6. Click **OK** in the **Manage Hyperlink** dialog box.

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.

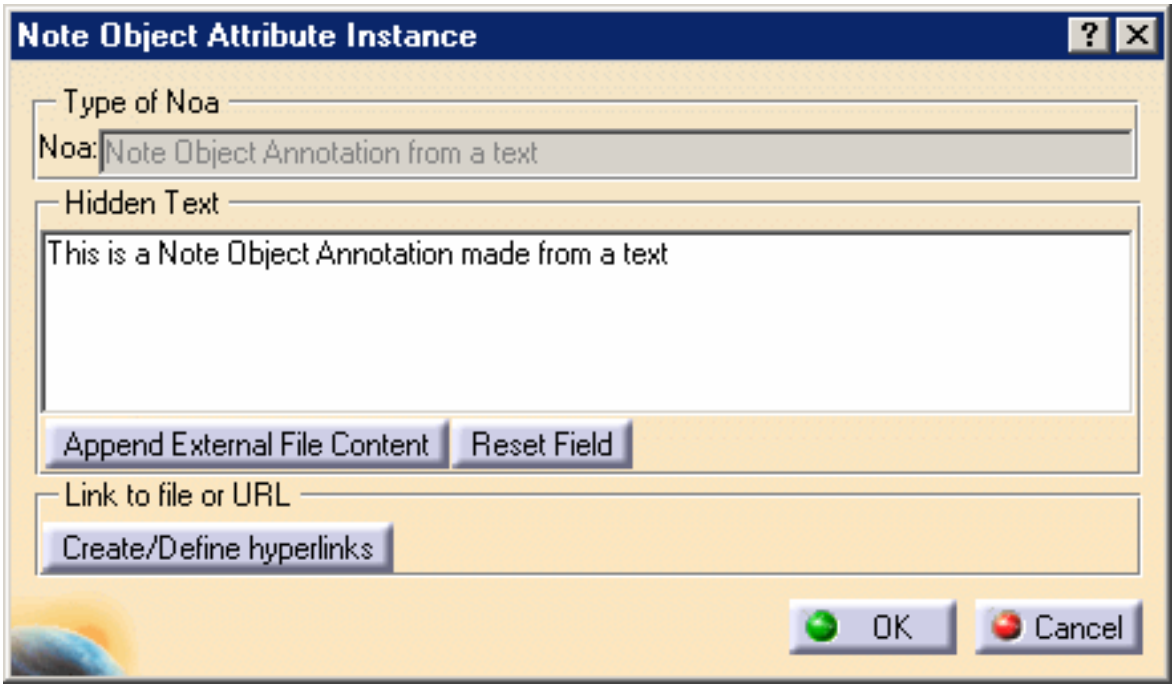
7. Click **OK** in the **Note Object Attribute Reference** dialog box.

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**.



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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.

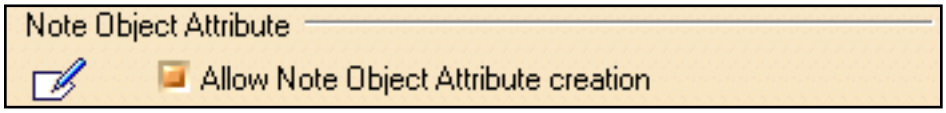
When the Note Object Attribute orientation is modified, text contained in the 2D component follows or not according to the text orientation reference.

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**

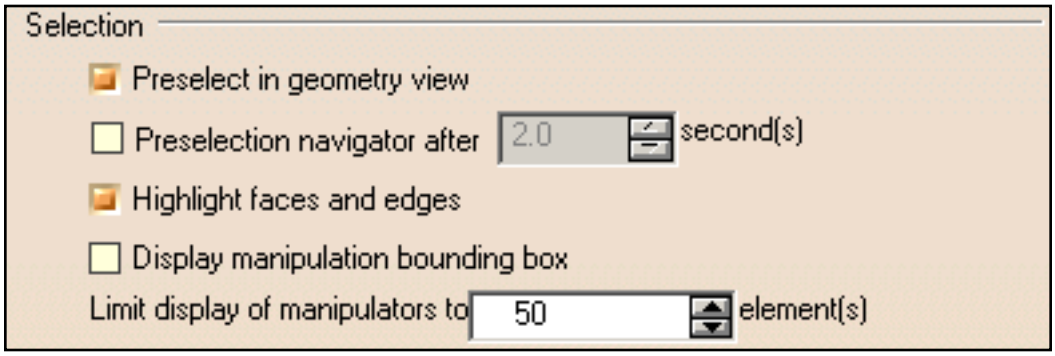
- With the **Sheet** option, the text does not follow the Note Object Attribute orientation.
- With the **View / 2D Component** option, the text follows the Note Object Attribute orientation.

Open the Tolerancing_Annotations_01 CATPart document.

Check the option allowing you to create a Note Object Attribute.
See Tolerancing setting.

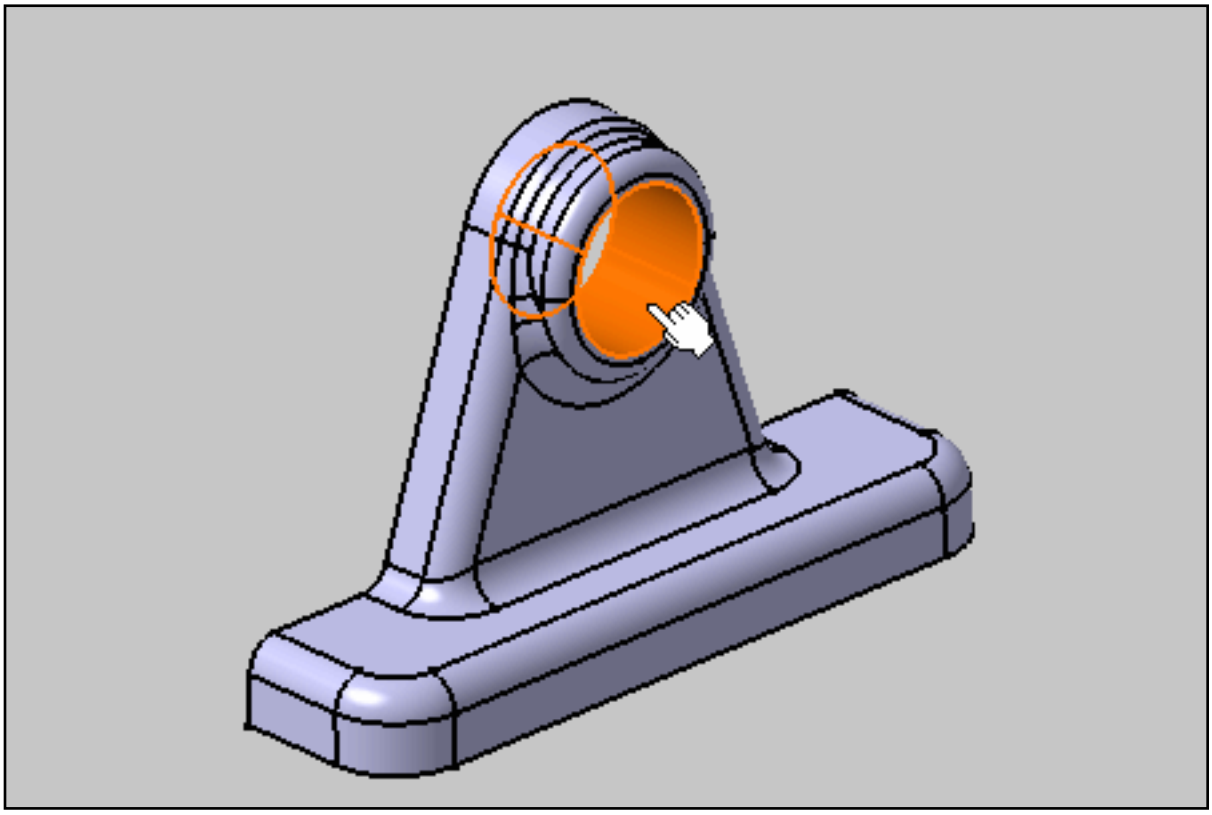


To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

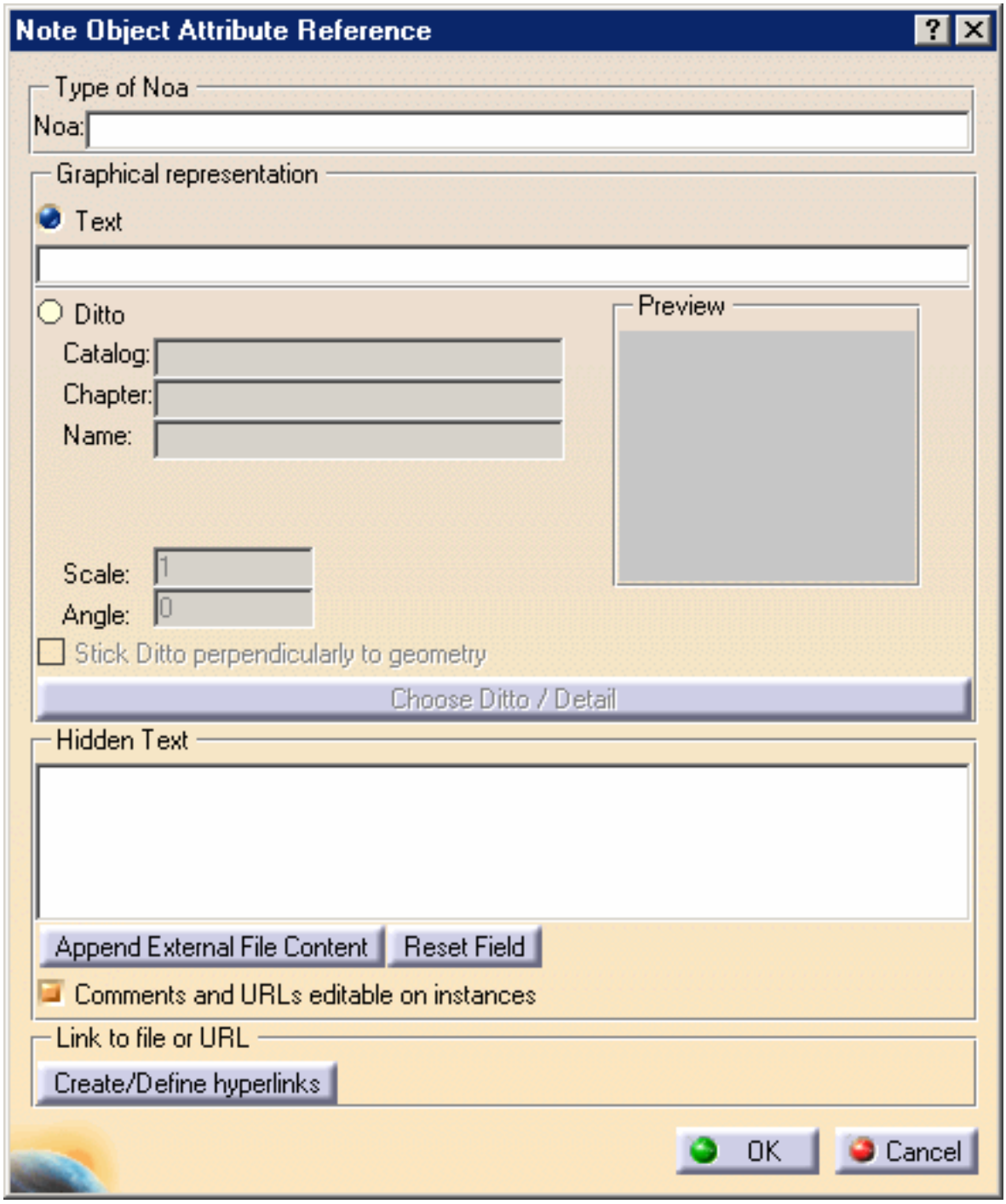


1. Click the **Note Object Attribute** icon: 

2. Select the surface as shown on the part.



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

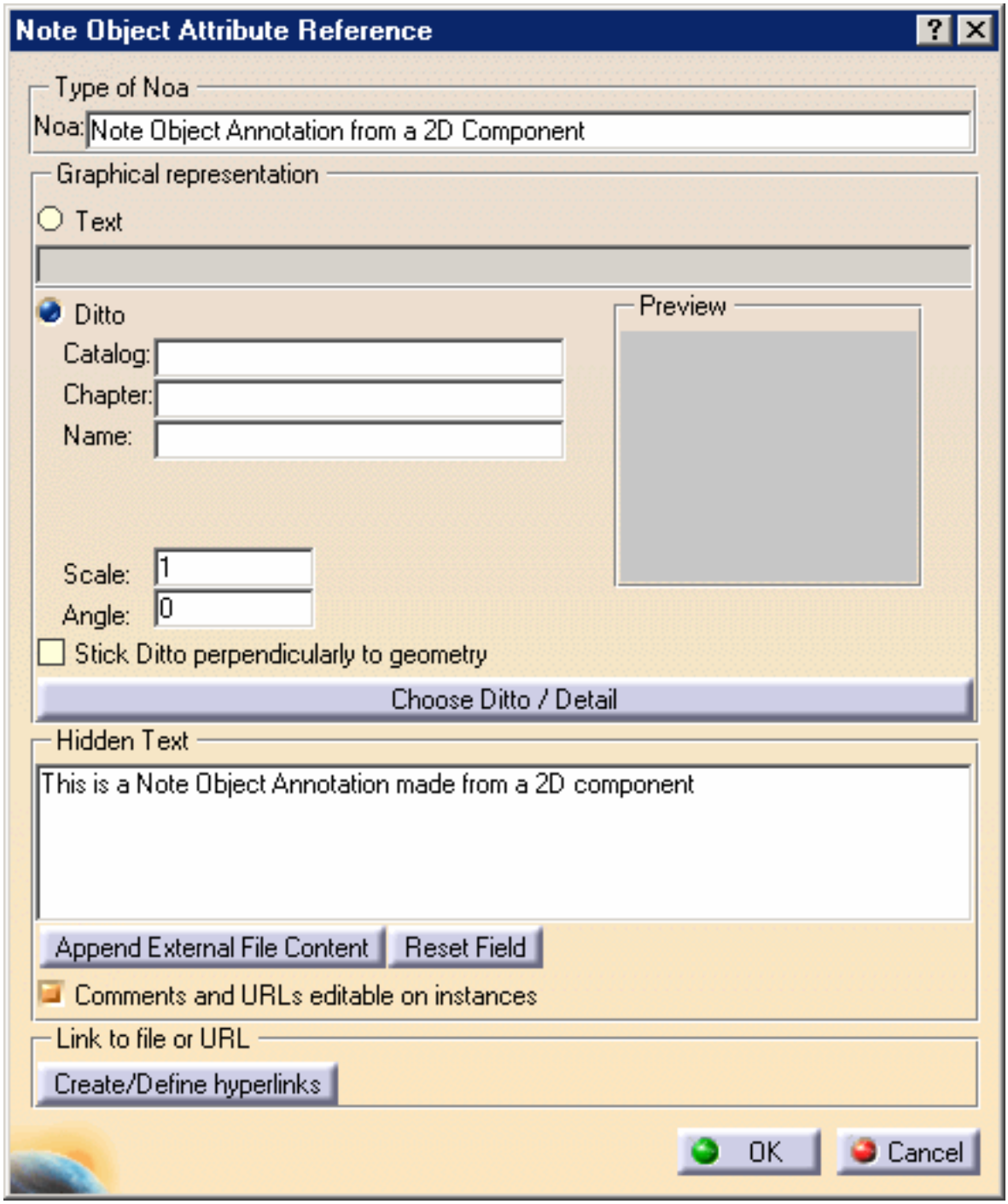


3. Select **Ditto** in **Graphical representation** and enter the following texts:

   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**

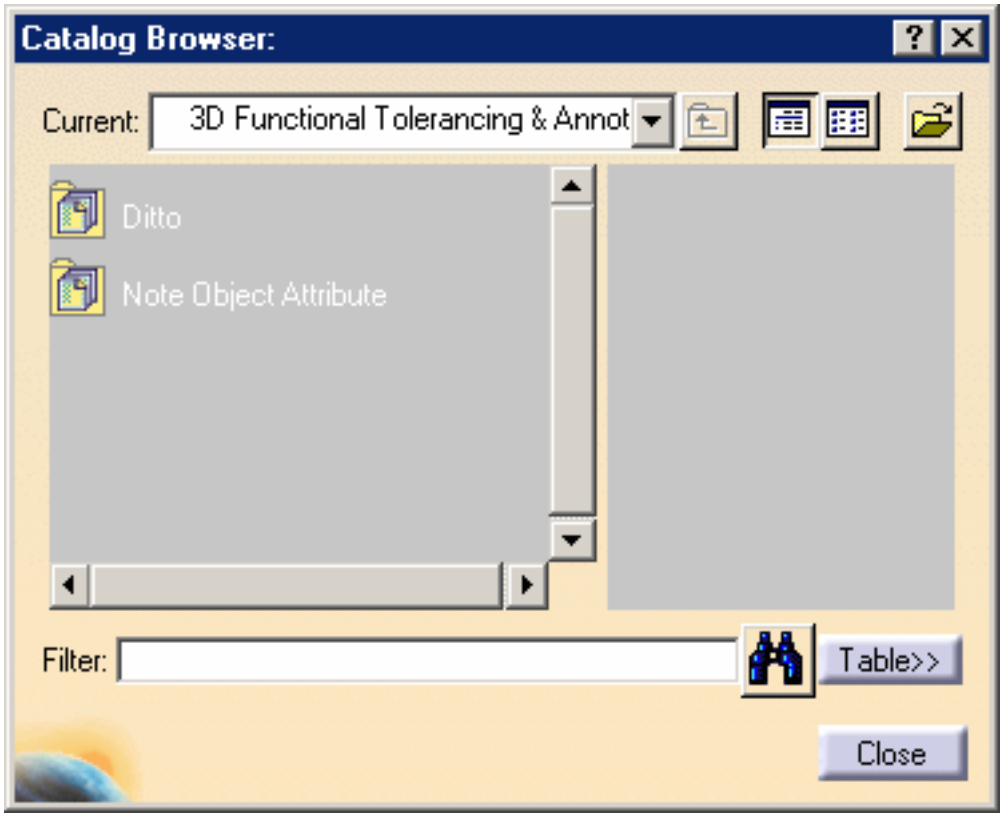
The **Comments and URLs editable on instances** check button allows user to modify the **Hidden Text** data during Note Object Attribute instantiation or modification.



4. Click **Choose Ditto / Detail**.

   The **Catalog Browser** dialog box appears.

5. Browse and select for the Component document.



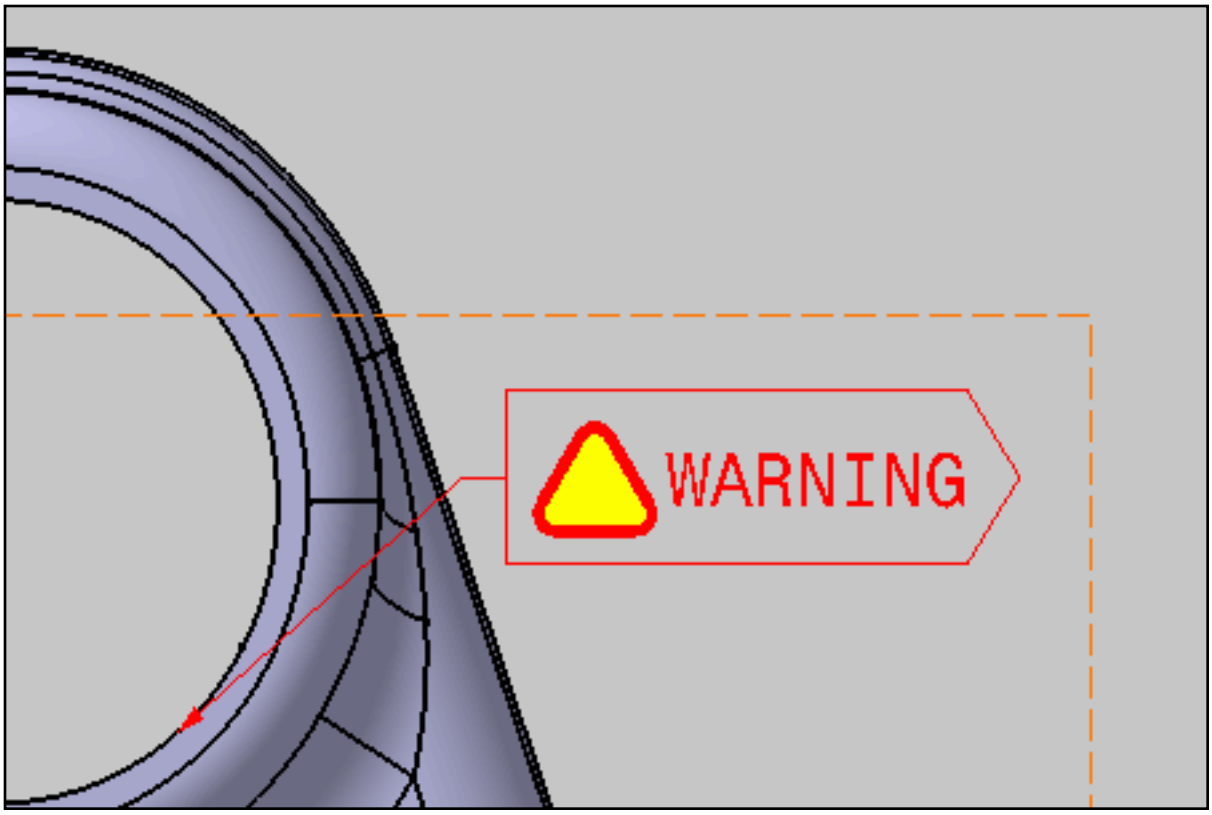
6. Double-click the **Ditto** component family item.

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 .

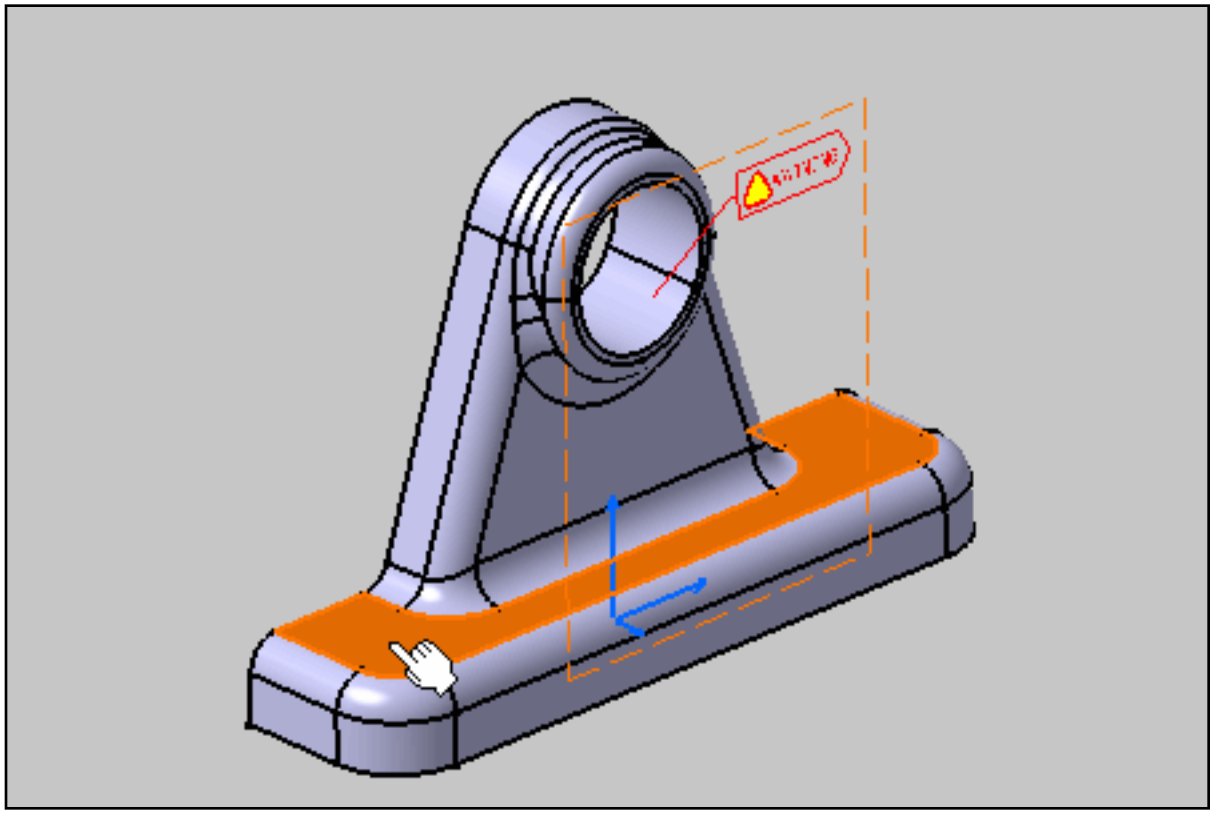
   The Note Object Attribute is created.



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.

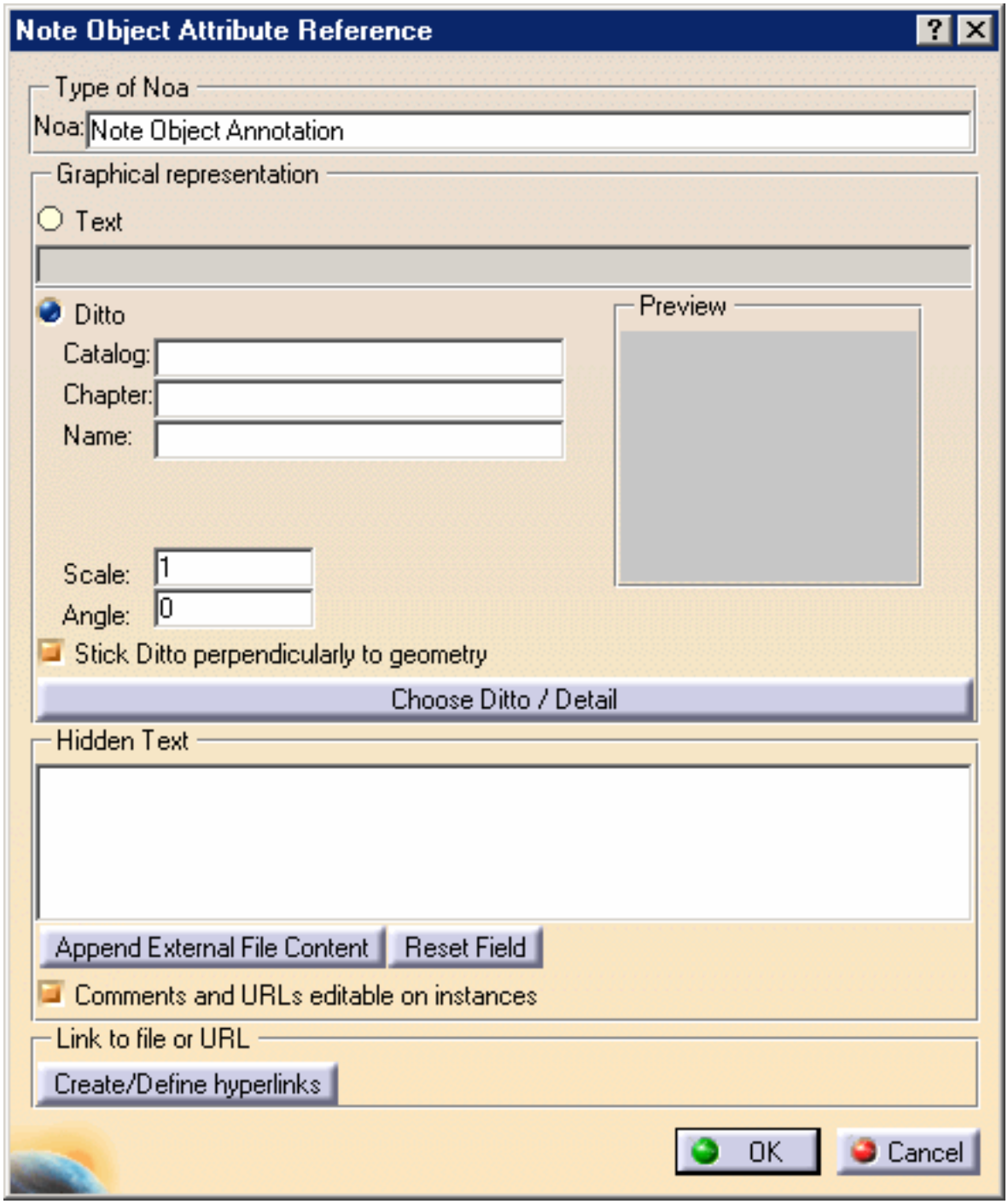


The **Note Object Attribute Reference** dialog box appears.

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**.

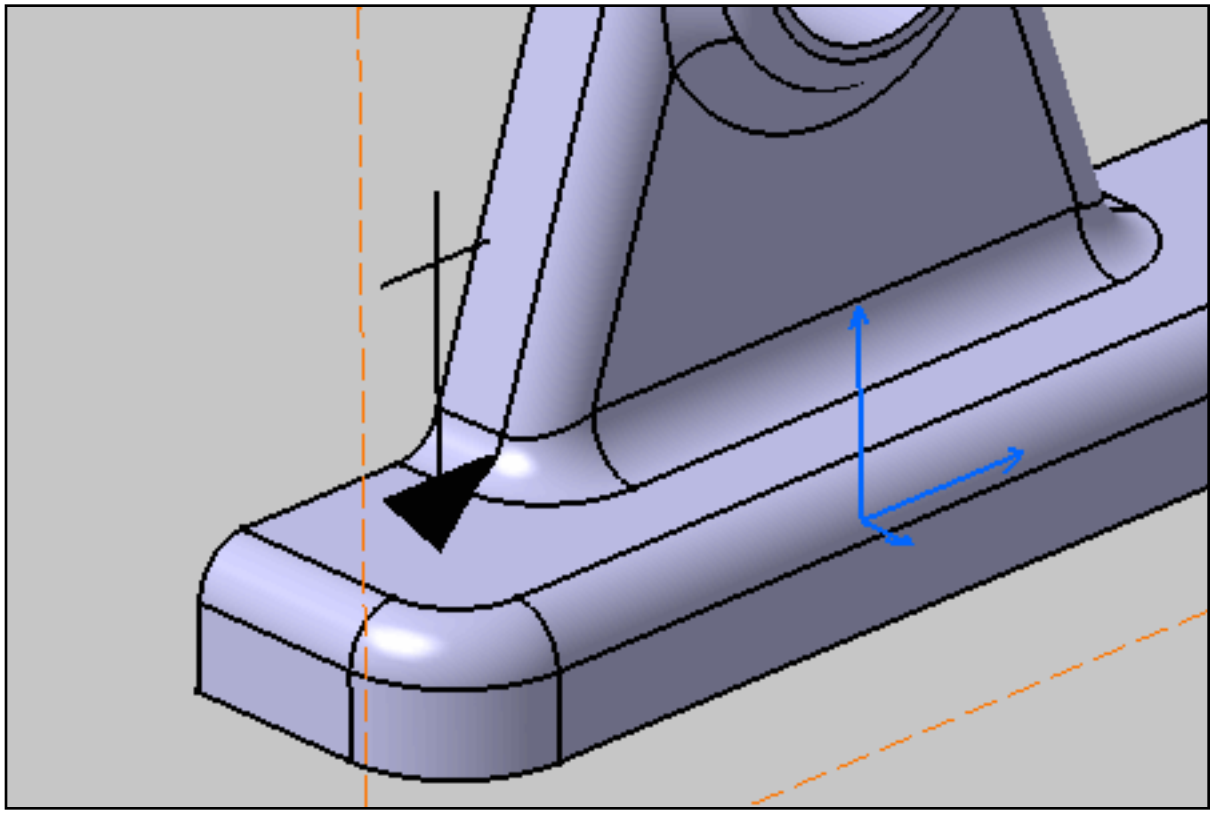
    The **Catalog Browser** dialog box appears.

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.



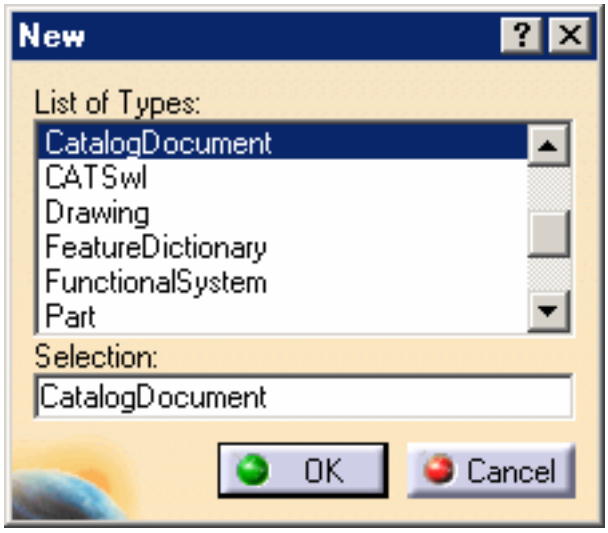
Back Up Next

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**

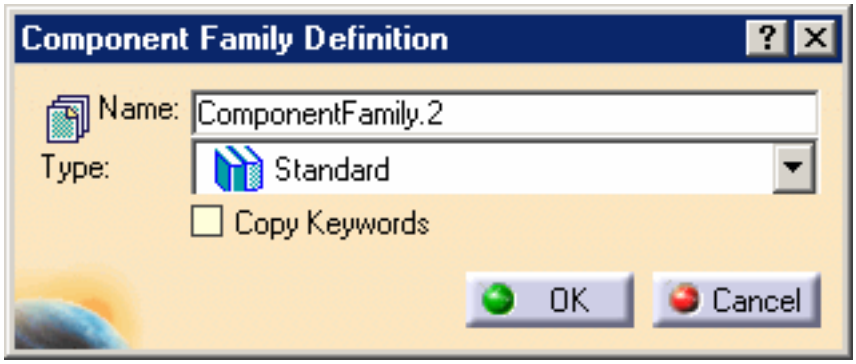


2. Check that Chapter.1 is activated in the catalog tree.



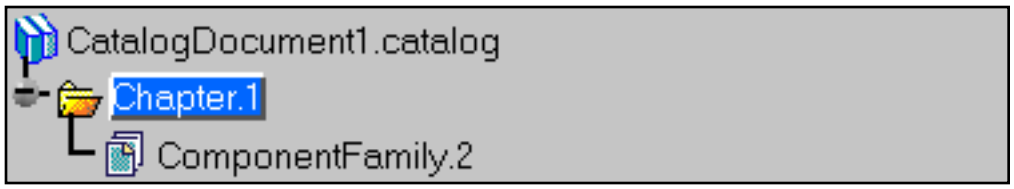
3. Click the **Add Family** icon: 

   The **Component Family Definition** dialog box appears.

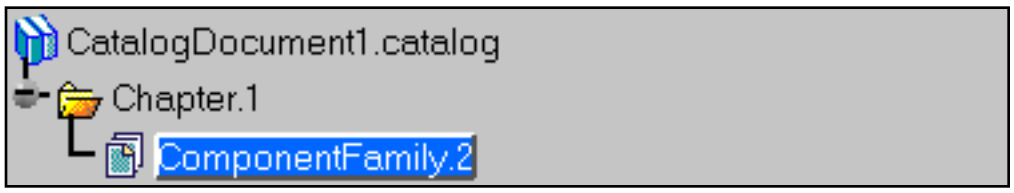


4. Click **OK**.

   The **Component Family Definition** is added to the chapter.

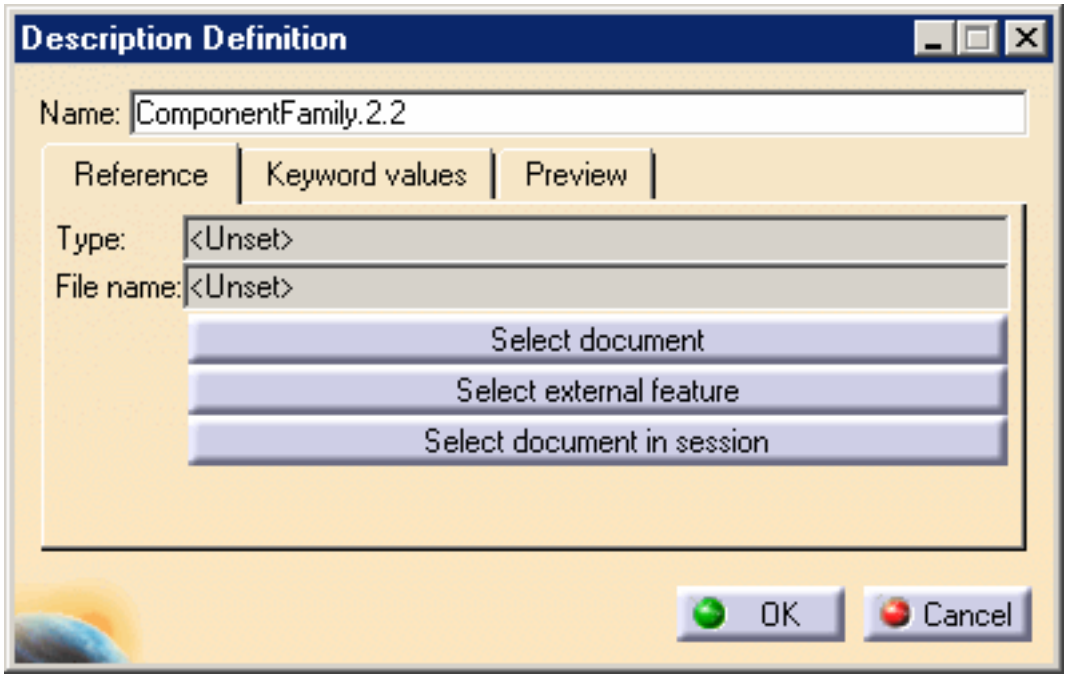


5. Double-click the Component Family to activate it.



6. Click the **Add Component** icon: 

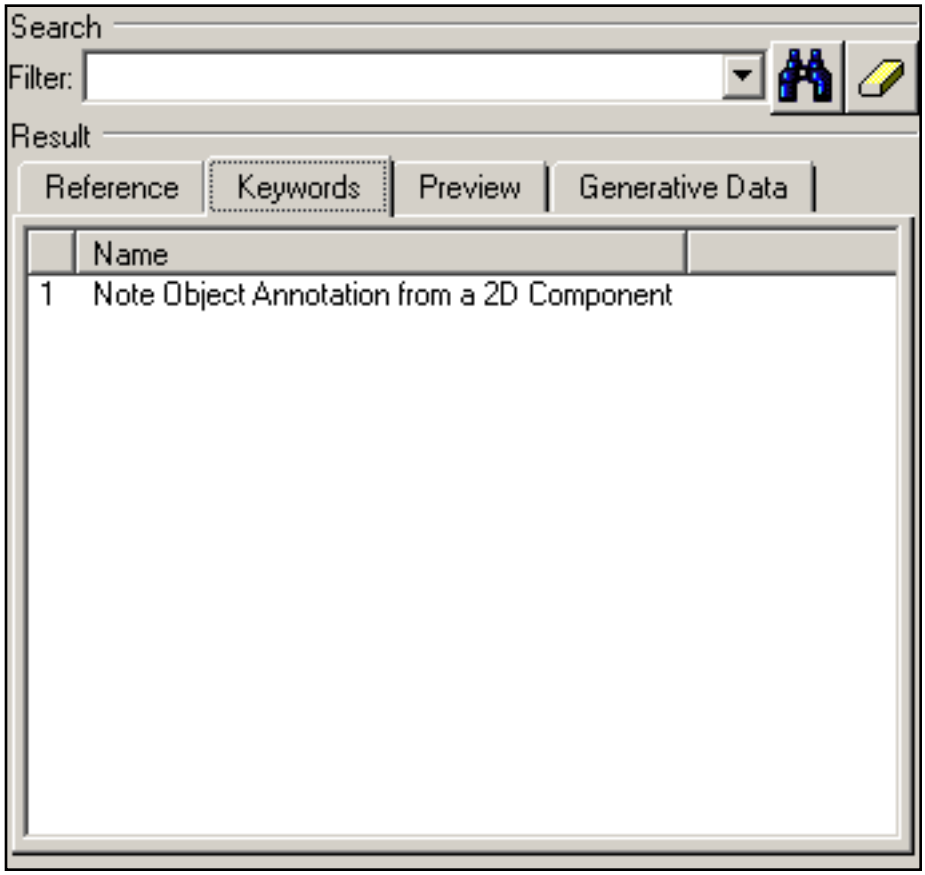
   The **Description Definition** dialog box appears.



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

Back Up

Managing Annotation Connection

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.

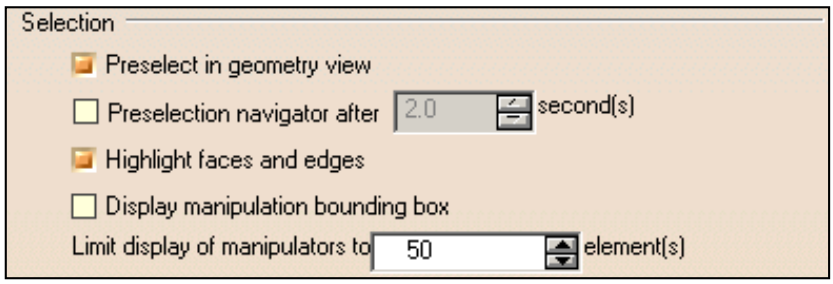
Back Up Next

Using the Scope Range

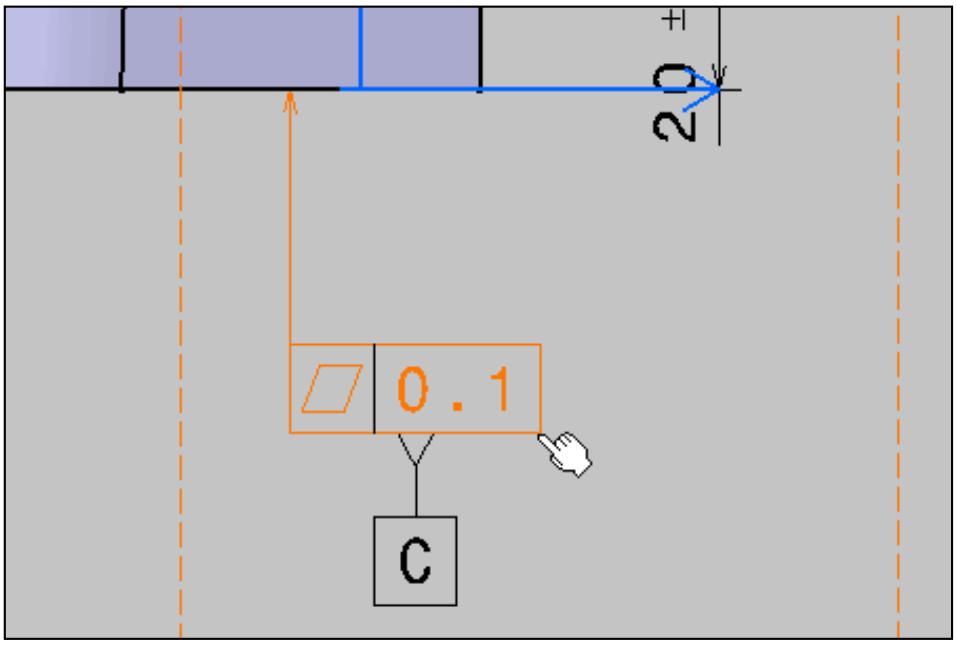
This task shows you how to use the scope range when using the connection management and check the validity of the reconnected annotations according to the selected scope option.

Open the Tolerancing_Annotations_03 CATPart document.

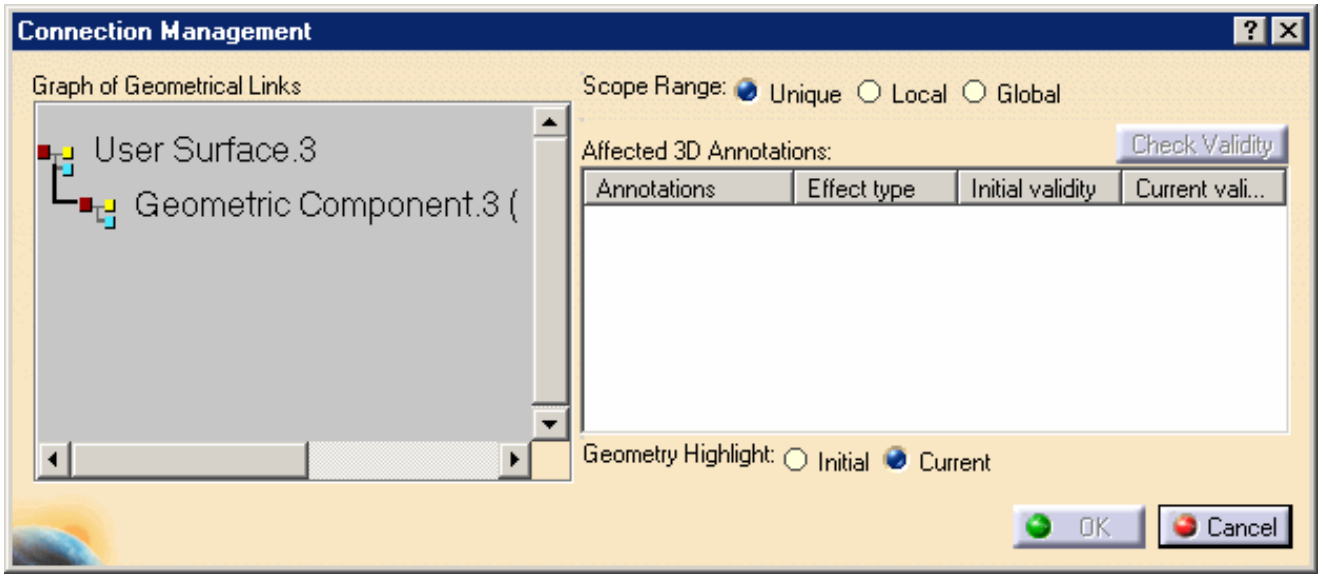
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



1. Right-click the annotation as shown on the part and select the **Associated Geometry -> Geometry Connection Management** from the contextual menu.

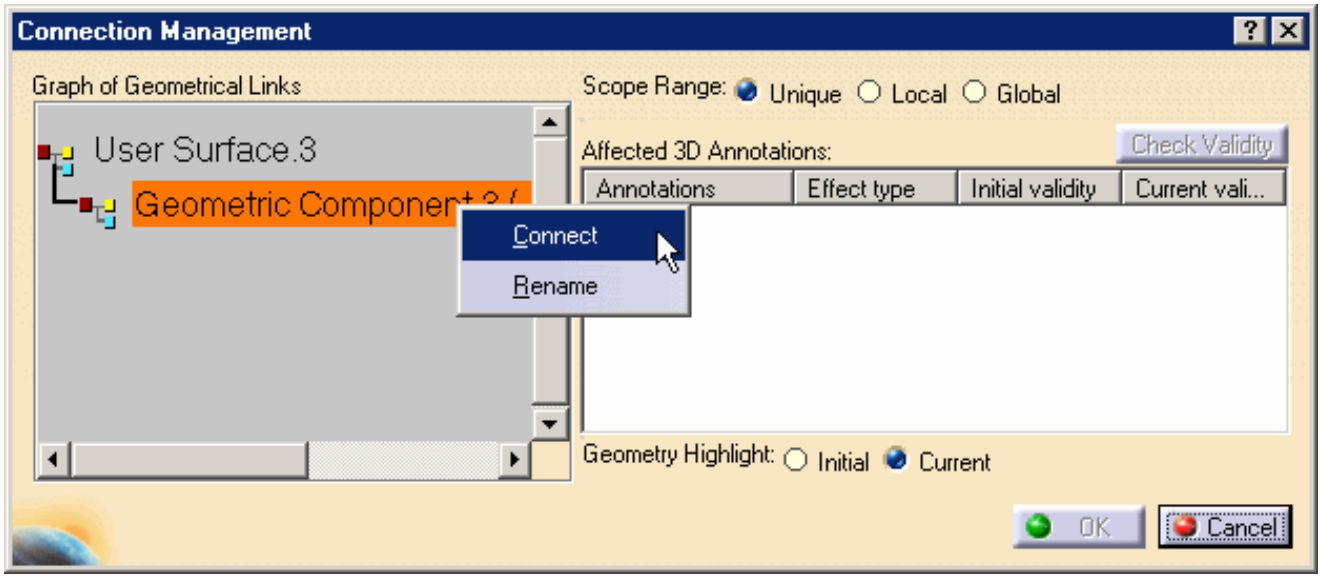


The **Connection Management** dialog box appears.

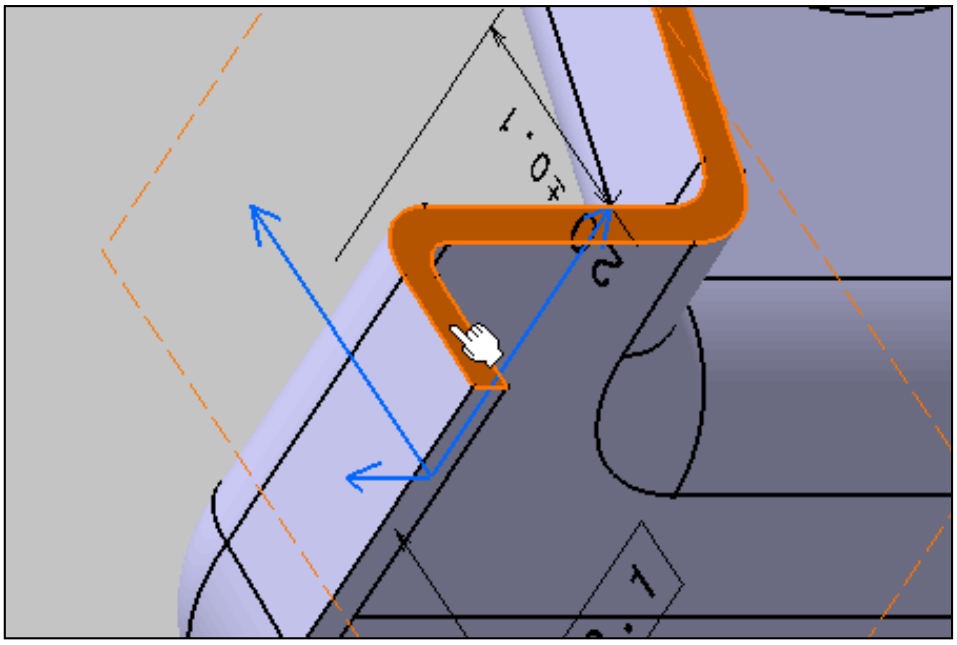


2. Check that **Unique** option is activated in **Scope Range**.
The geometry connection modification will only affect the selected annotation.

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.



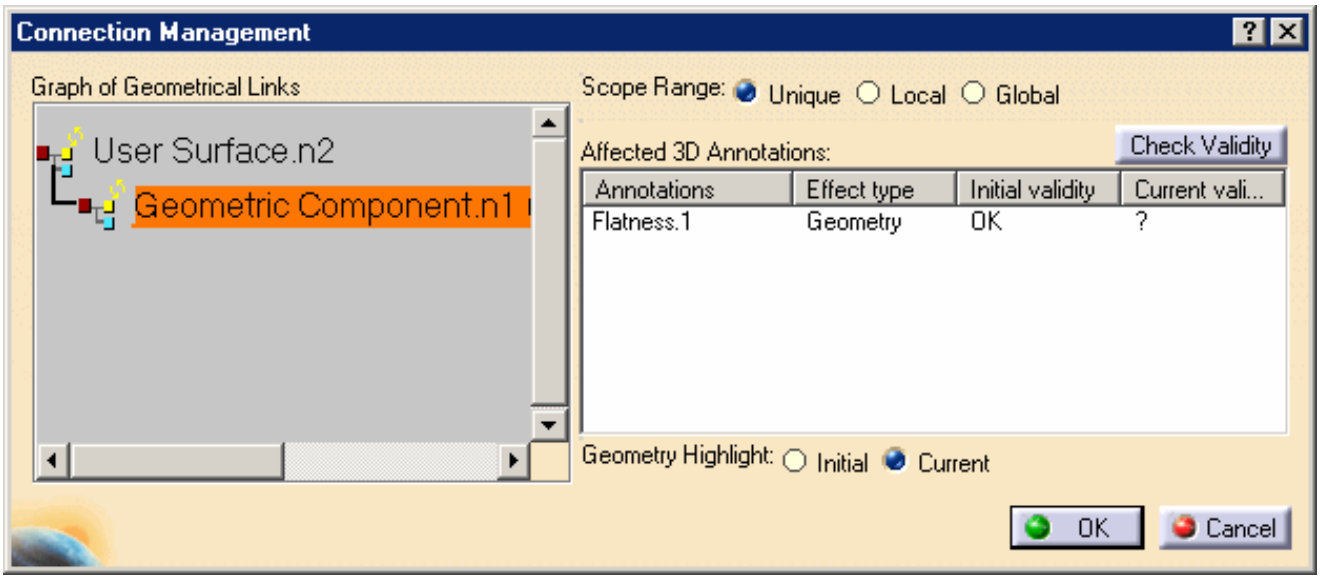
The **Connection Management** dialog box displays:

The selected annotation

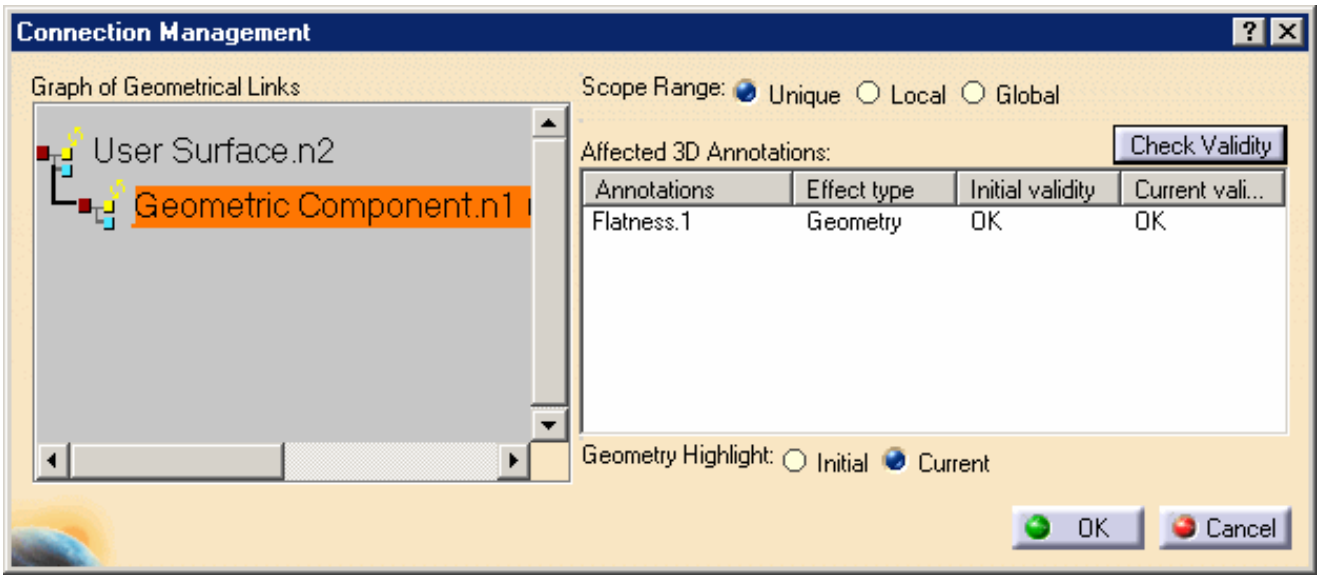
The effect type

The initial validity of the selected annotation

The current validity of the selected annotation after the reconnection.

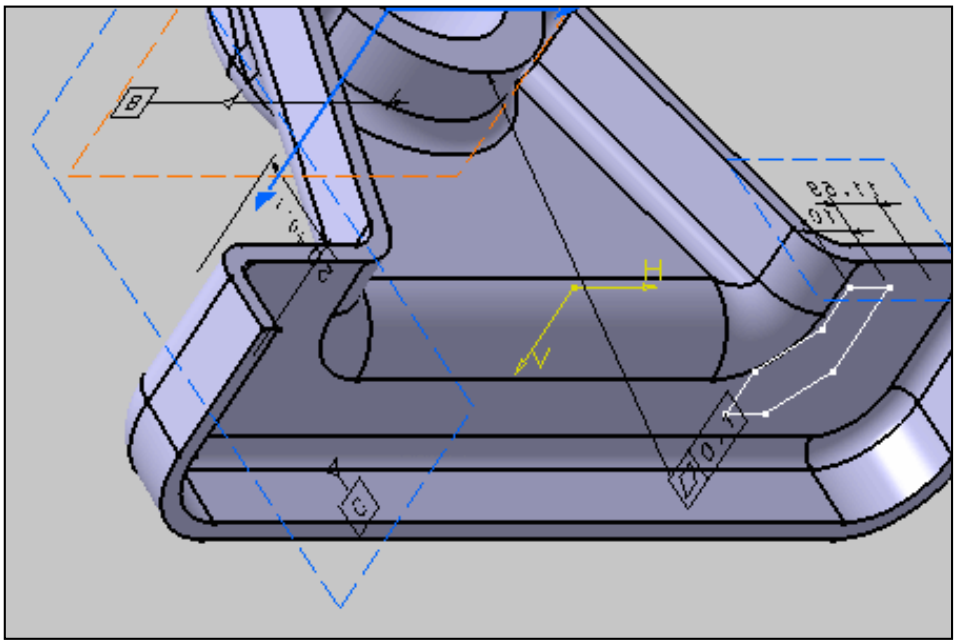


5. Click the **Check Validity** command to check the new geometry component validity relative to the selected annotation.



6. Click **OK**.

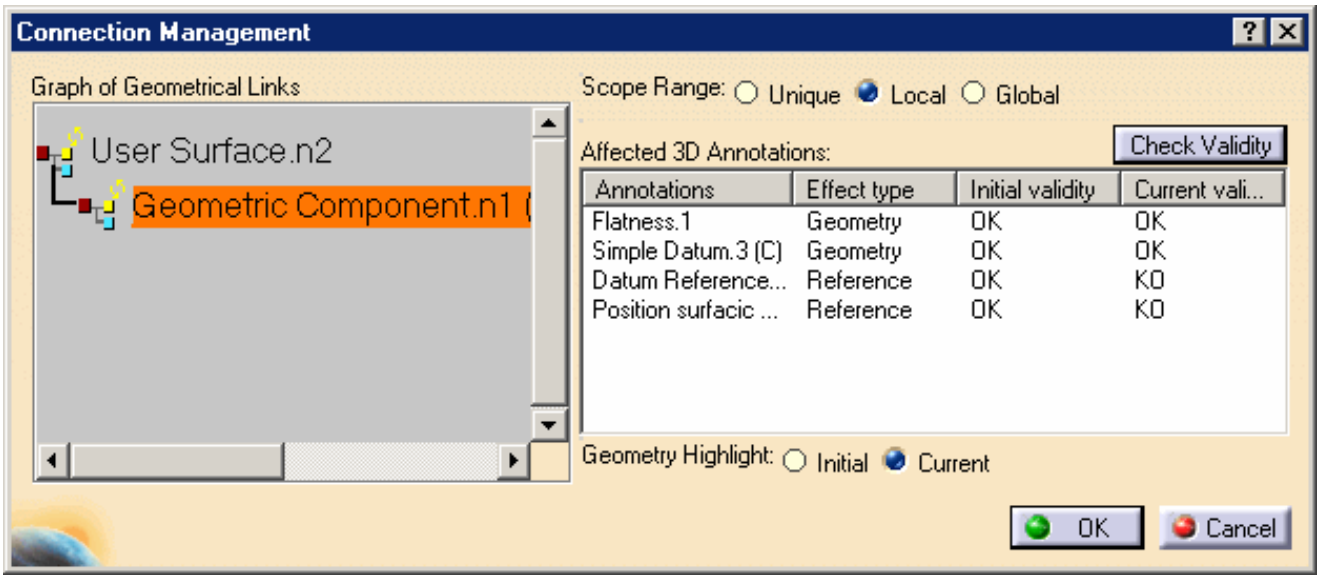
The annotation is now connected to the new surface



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.

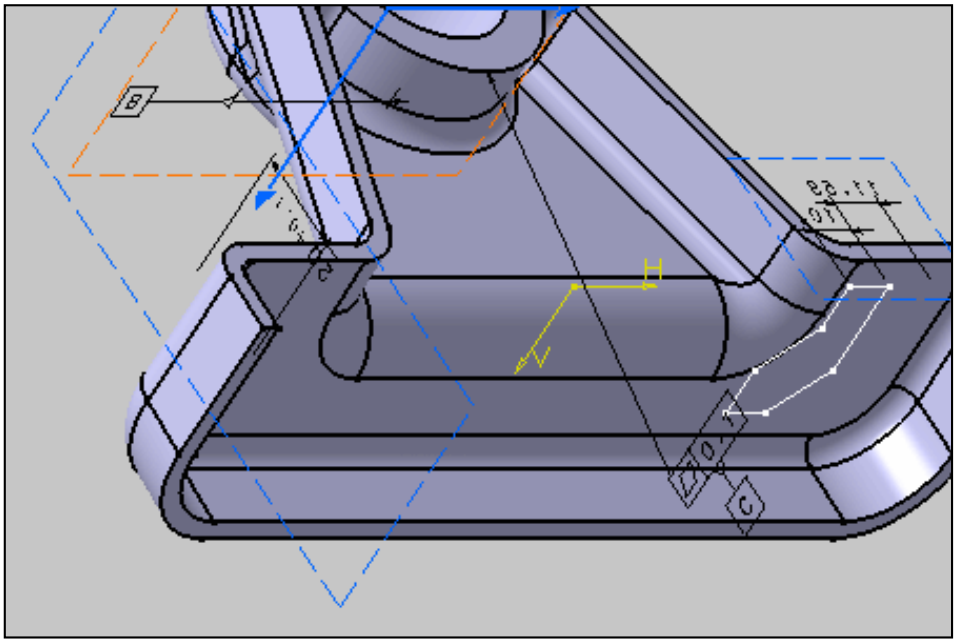
The **Connection Management** dialog box displays now the selected annotation and all annotations related with it.



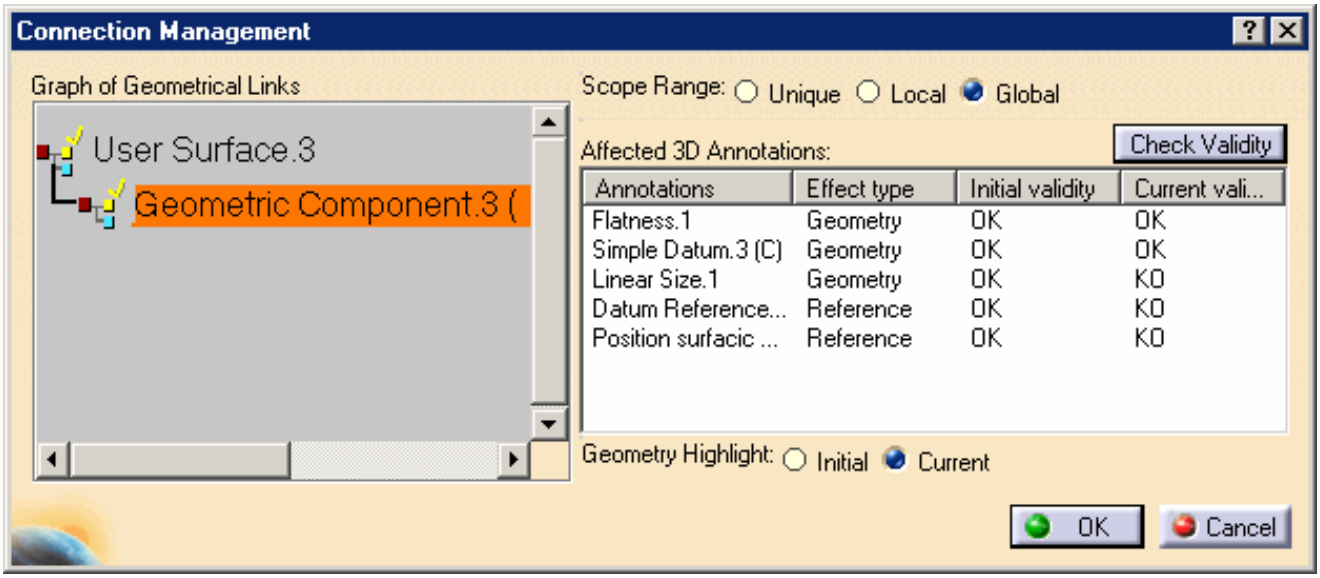
The validity status is KO for the **Datum Reference Frame.1** and **Position surfacic profile.1** annotations because they will not have any meaning if the modification is applied.

8. Click **OK**.

The annotations are now connected on the new surface.



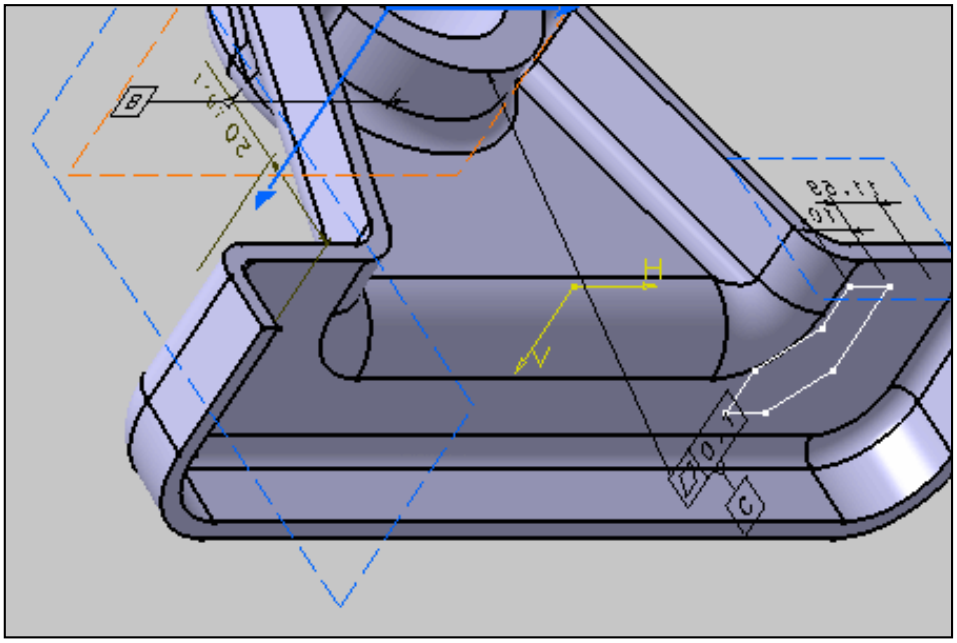
9. Close the part document, re-open it and redo step 1 to 5 with the **Global** option in **Scope Range.** The geometry connection modification will only affect all the annotation that are directly or indirectly applied to the User Surface.1 feature.



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.



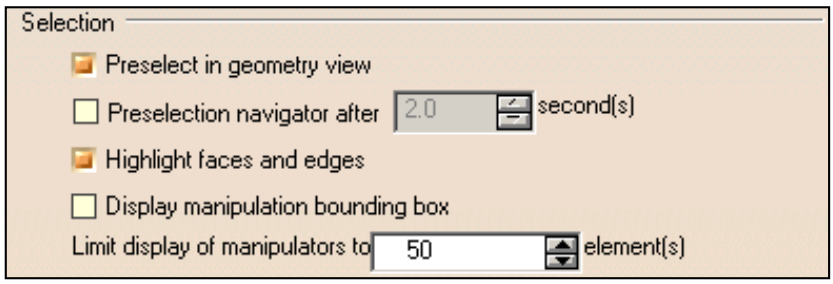
Up Next

Adding Geometry

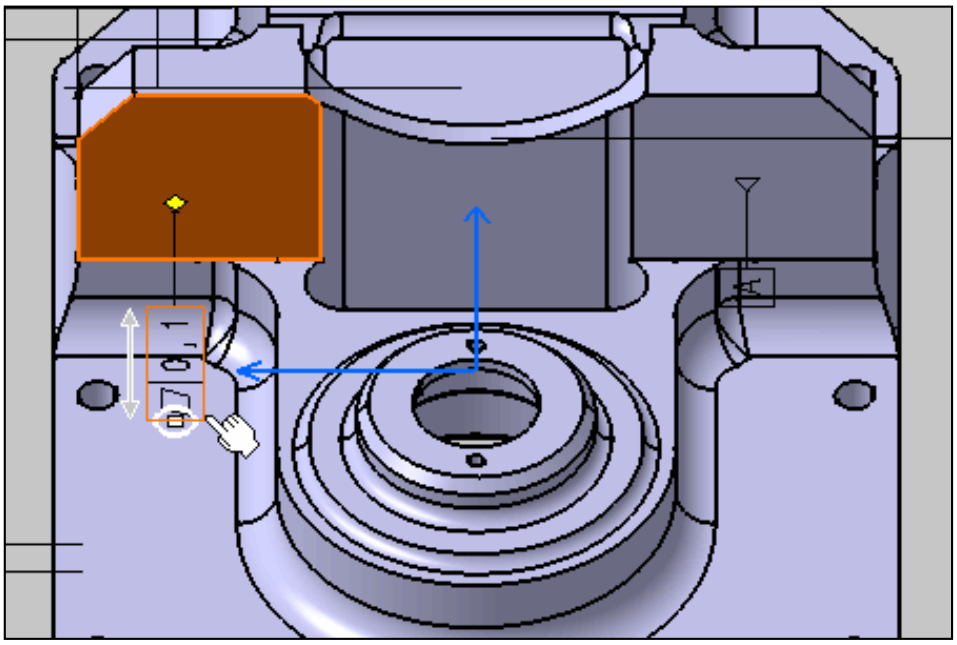
This task shows you how to add geometry to the user surface of an annotation.
See 3D Annotations and Annotation Planes concept.

Open the Tolerancing_Annotations_02 CATPart document.

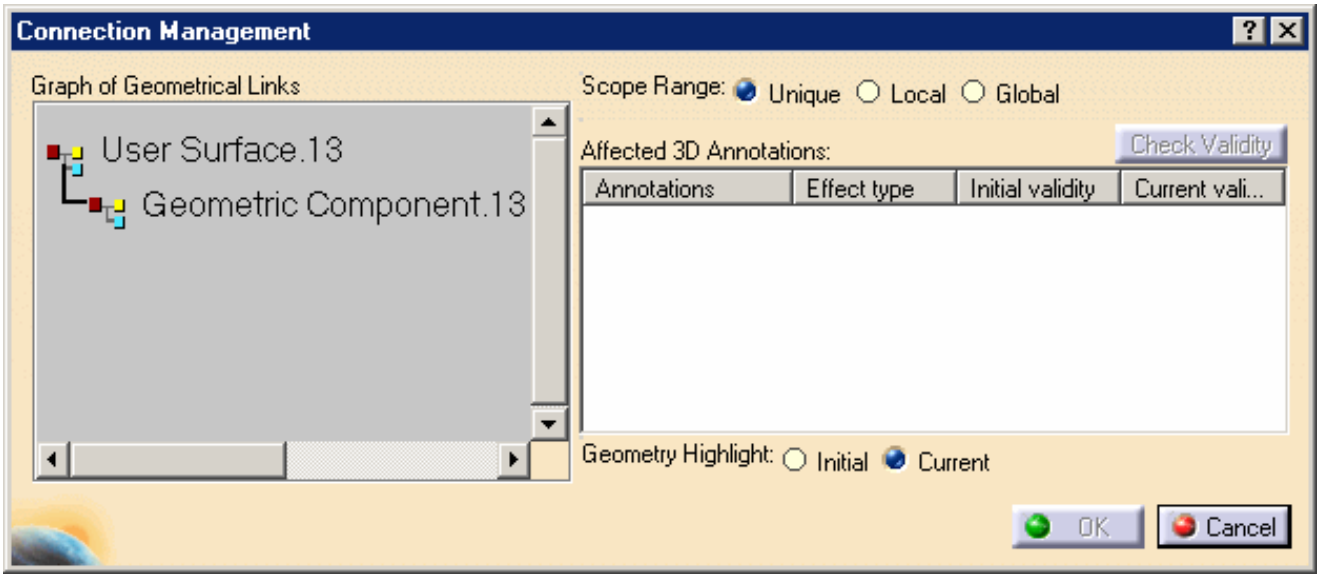
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



1. Right-click the annotation as shown on the part and select the **Associated Geometry ->Geometry Connection Management** from the contextual menu.



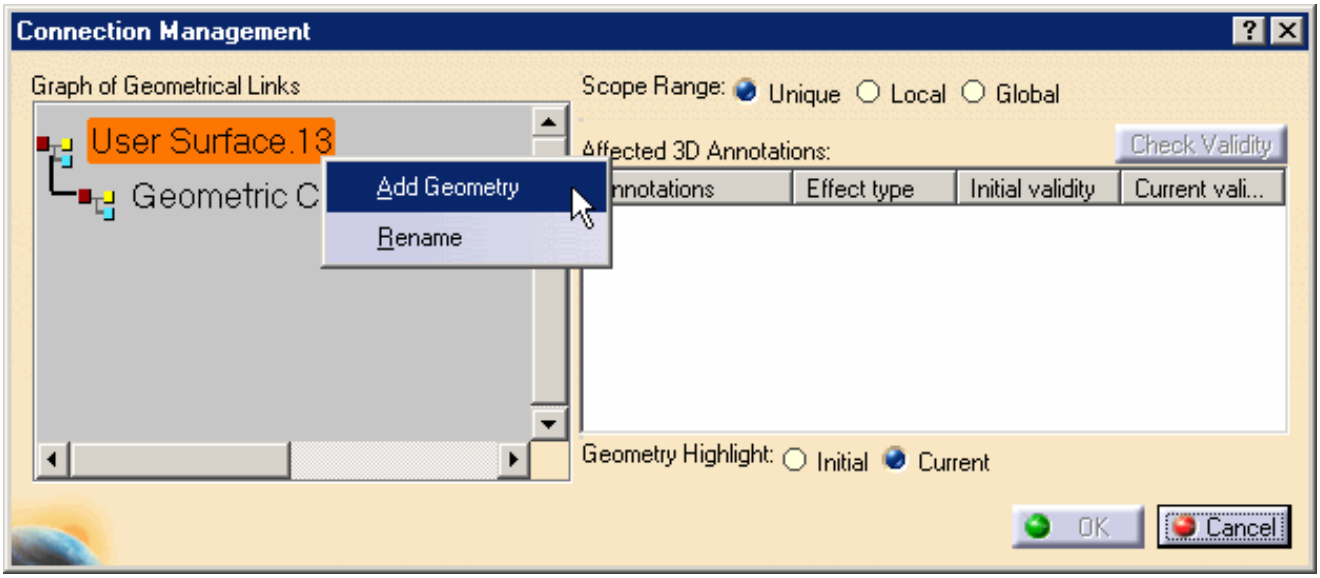
The **Connection Management** dialog box appears.



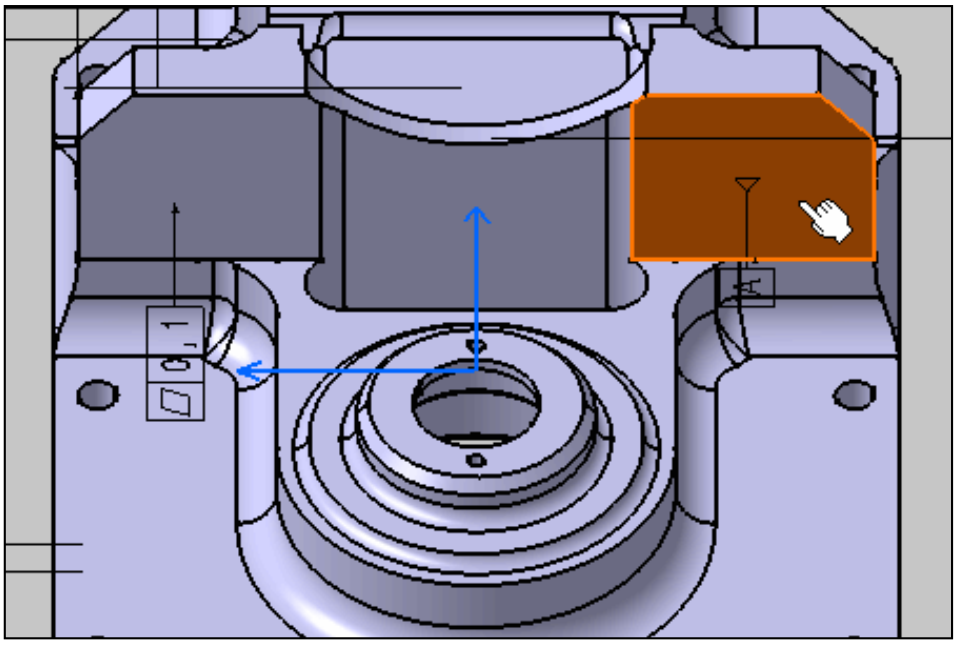
2. Check that **Unique** option is activated in **Scope Range**.

3. Right-click **User Surface.13** in the **Graph of Geometrical Links** as shown and select the **Add Geometry** from the contextual menu.

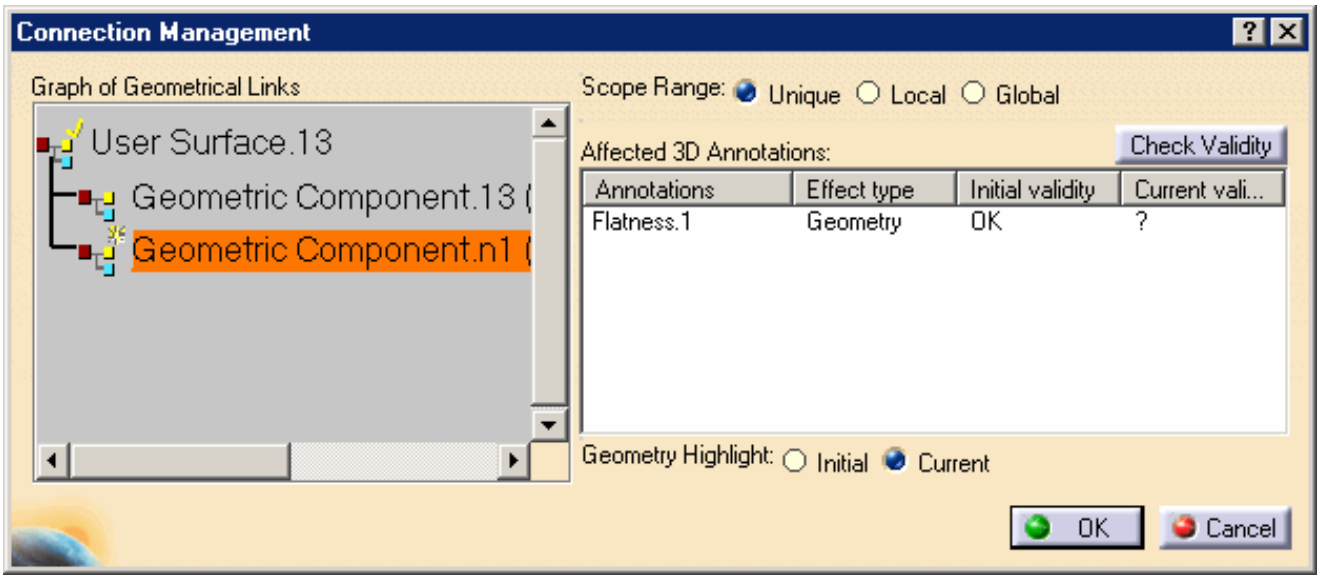
It will add a new geometric component to the user surface **User Surface.13** feature and prompt you to select the new geometrical element to be linked to the annotations.



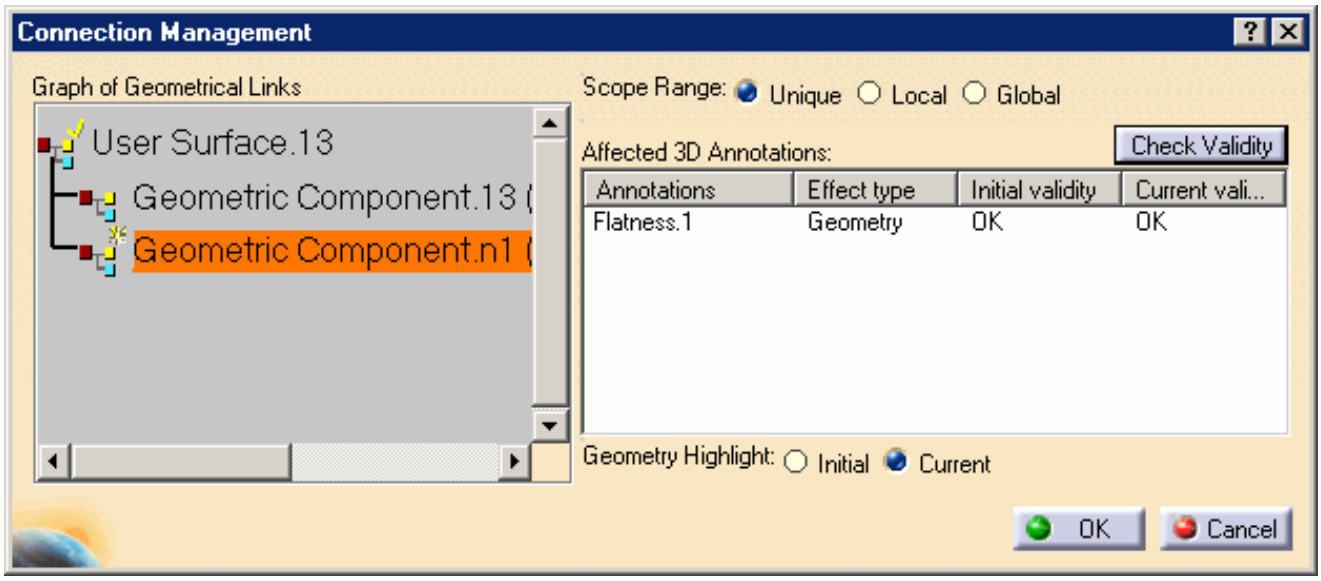
4. Select the surface as shown on the part.



The **Connection Management** dialog box displays the new added geometry to the structure: Geometric Component.1

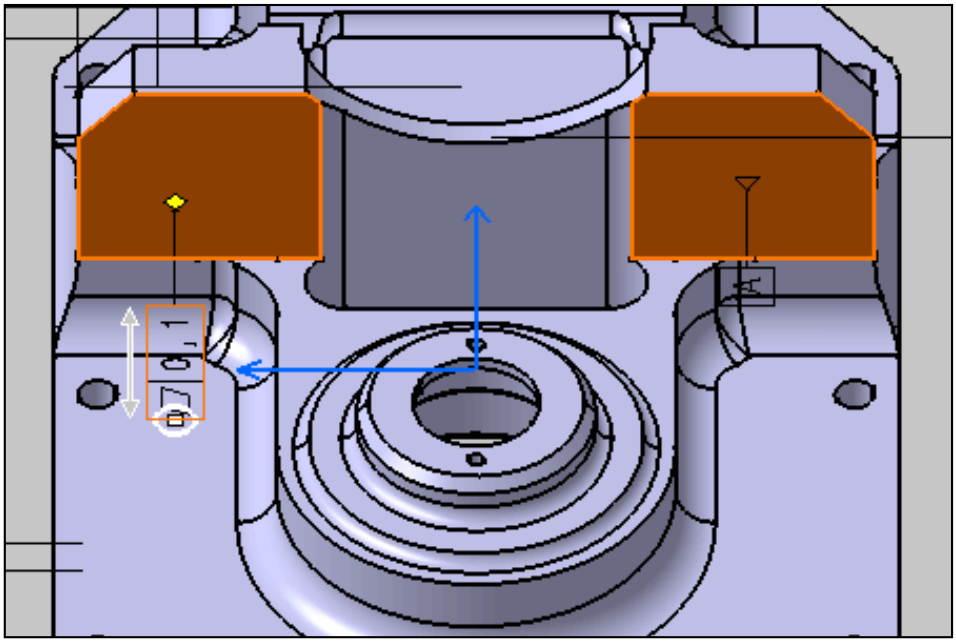


5. Click the **Check Validity** command to check the new geometry component validity relative to the selected annotation.



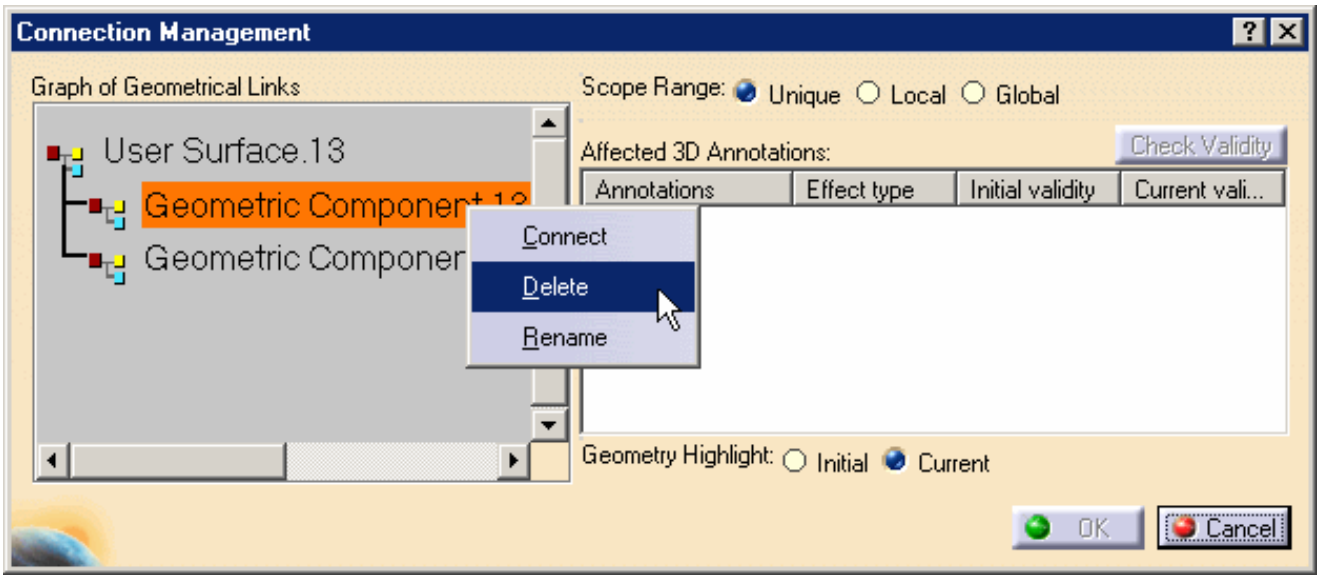
6. Click **OK**.

The two geometric components are now link to the annotation.



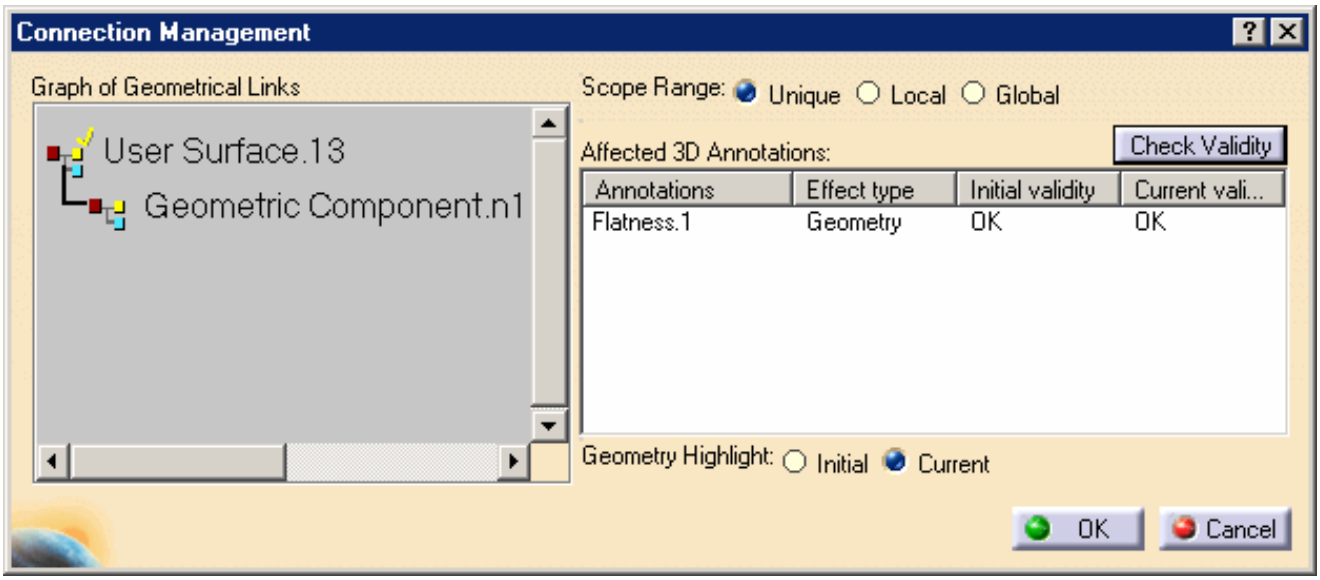
7. Right-click the annotation again and select the **Geometry Connection Management** contextual menu.

8. Right-click **Geometric Component.2** in the **Graph of Geometrical Links** as shown and select the **Delete** contextual menu.



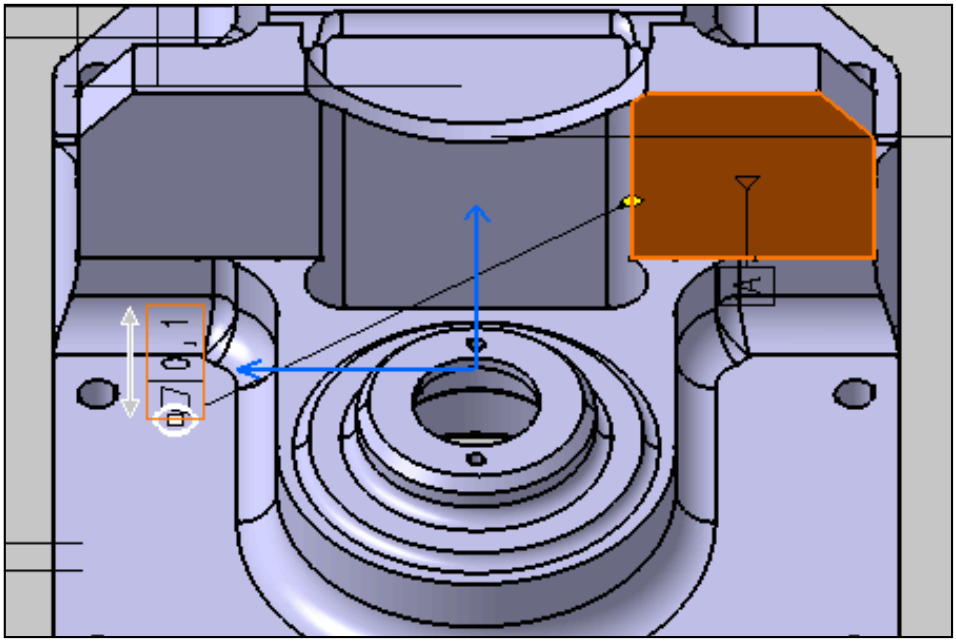
9. Click the **Check Validity**

The **Connection Management** dialog box displays the updated structure.



10. Click **OK**.

Only one geometric component is now linked to the annotation.



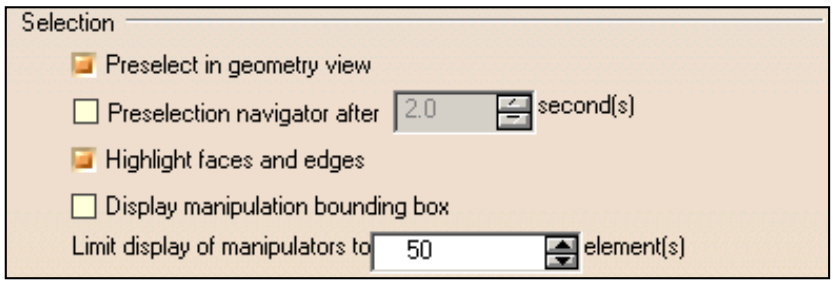
Back Up Next

Adding Component

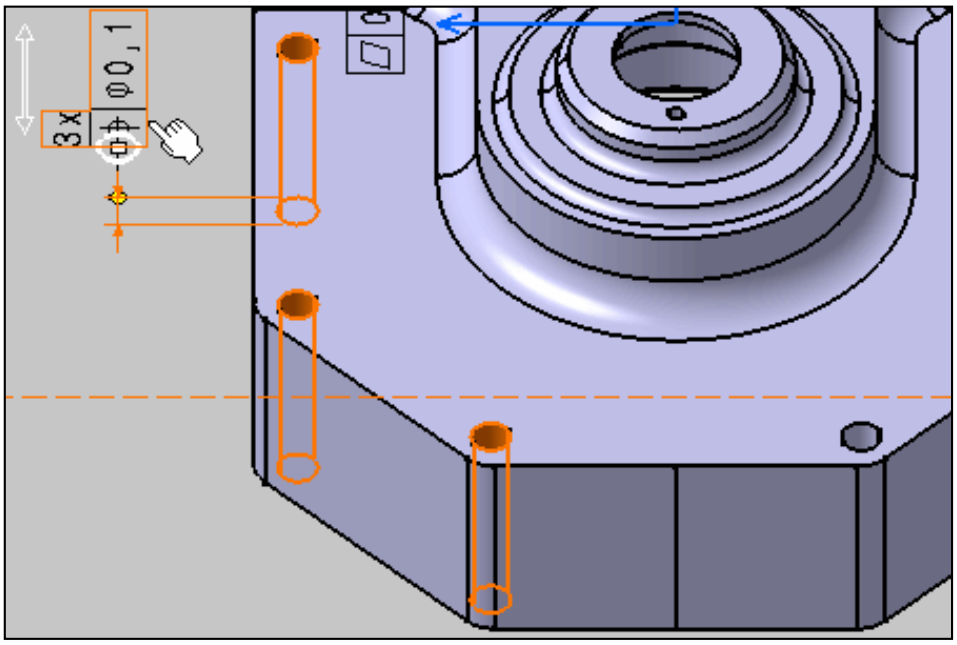
This task shows you how to add a user surface to the group of surfaces of an annotation.
See 3D Annotations and Annotation Planes concept.

Open the Tolerancing_Annotations_02 CATPart document.

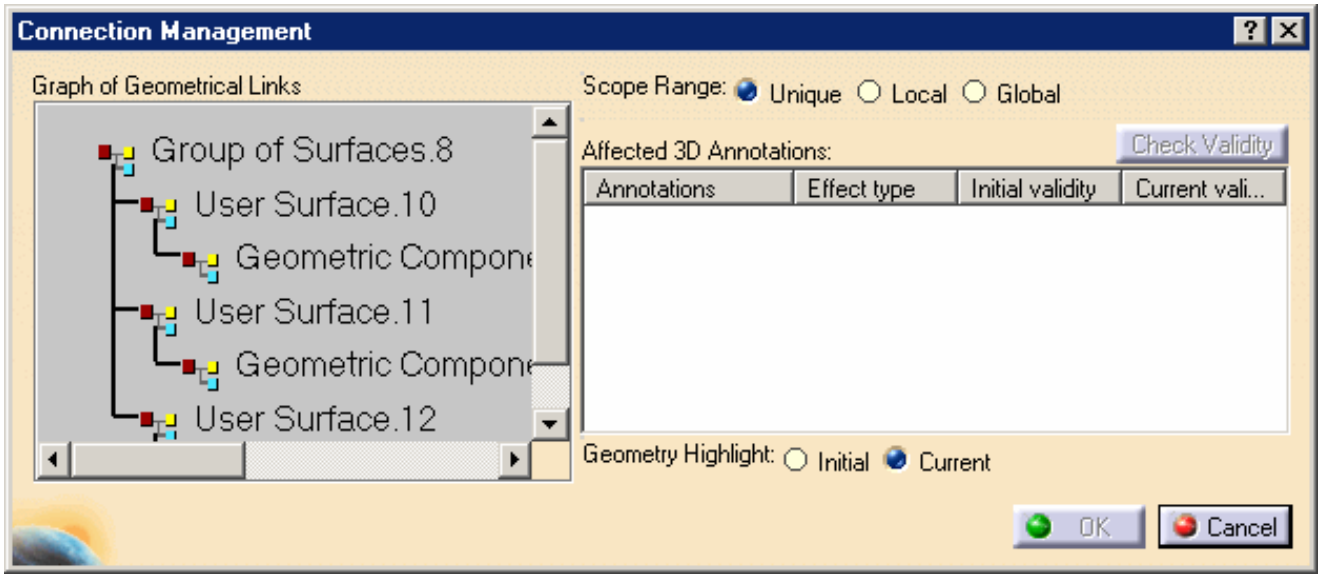
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



1. Right-click the annotation as shown on the part and select the **Associated Geometry ->Geometry Connection Management** from the contextual menu.



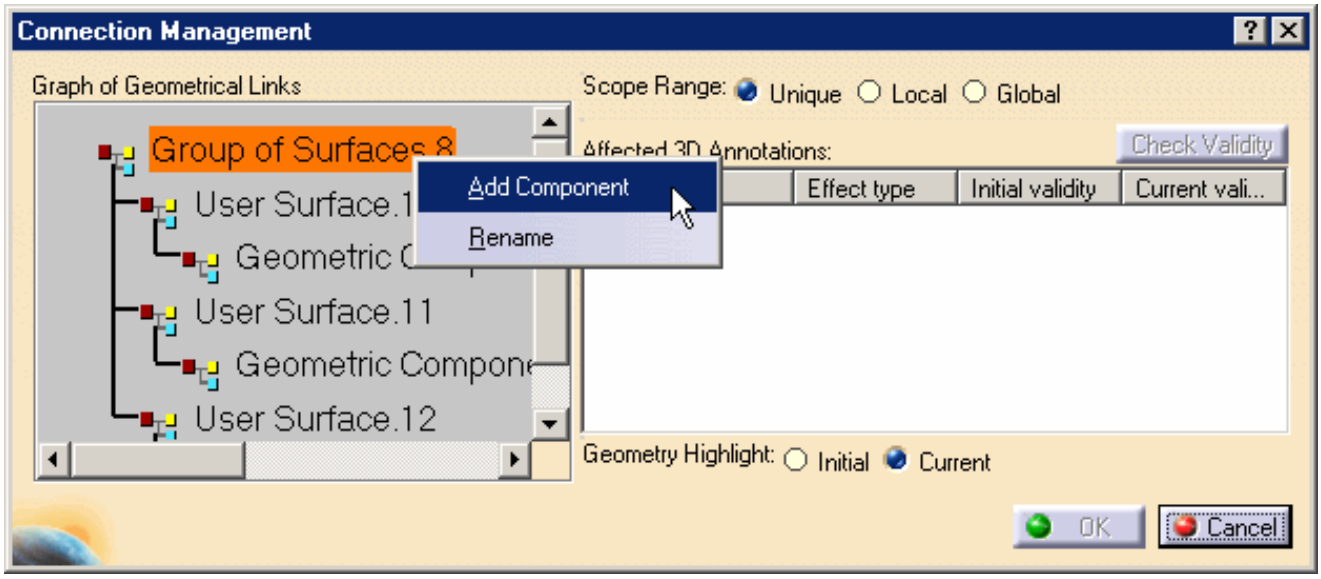
The **Connection Management** dialog box appears.



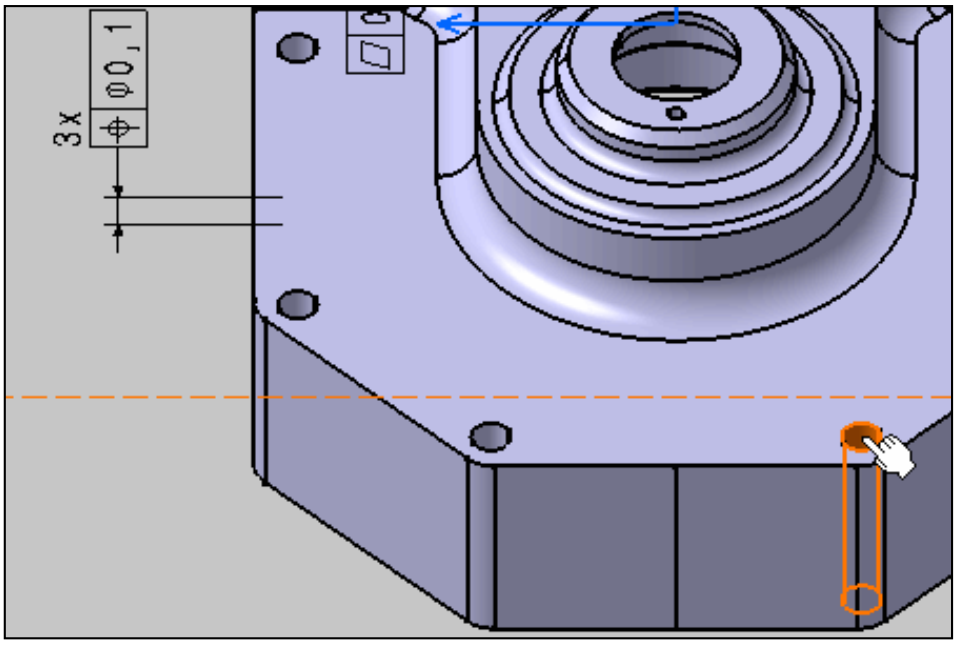
2. Check that **Unique** option is activated in **Scope Range**.

3. Right-click **Group of Surfaces.8** in the **Graph of Geometrical Links** as shown and select the **Add Component** contextual menu.

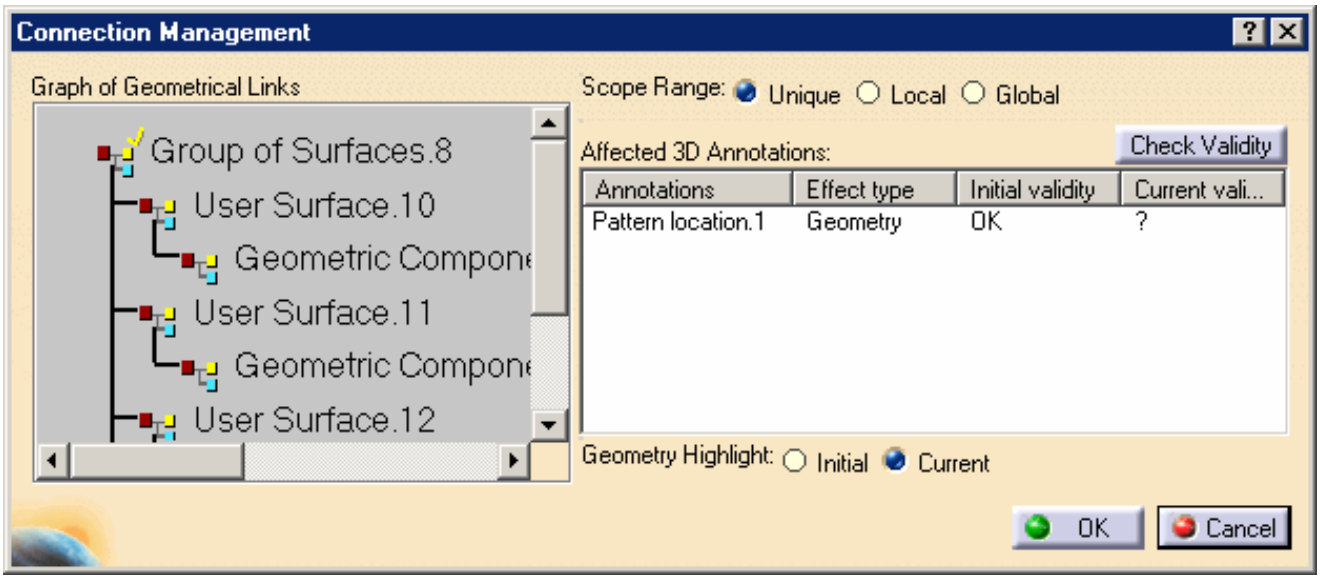
It will add a new component to the group of surfaces **Group of Surfaces.10** feature and prompt you to select the new geometrical element to be linked to the annotations.



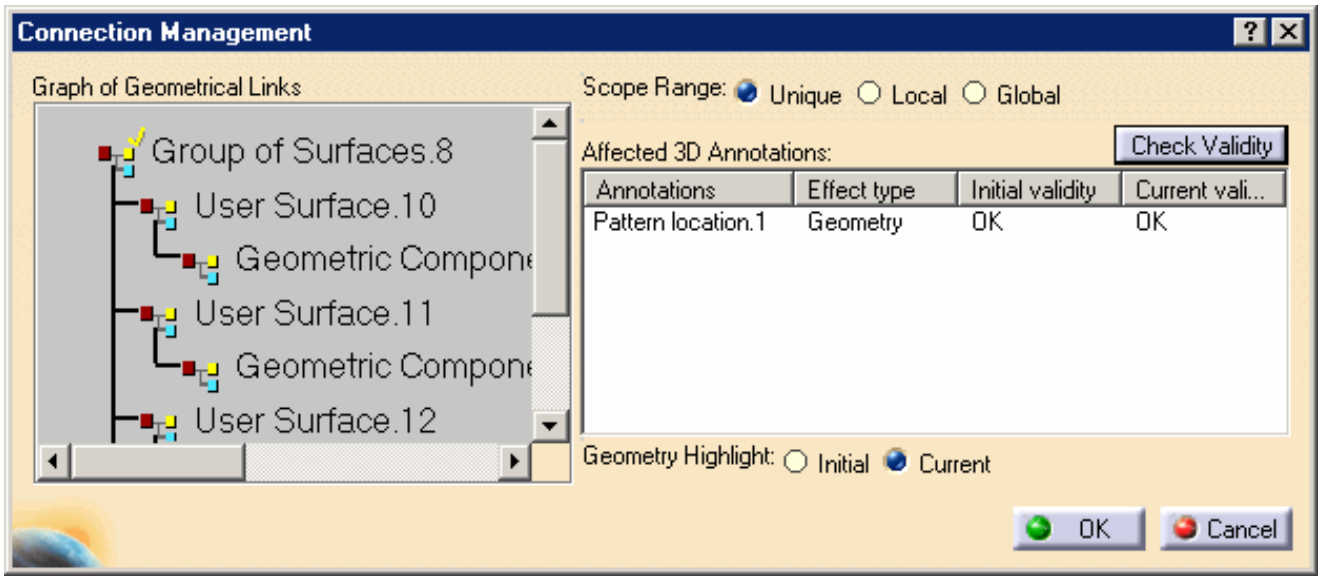
4. Select the hole surface as shown on the part.



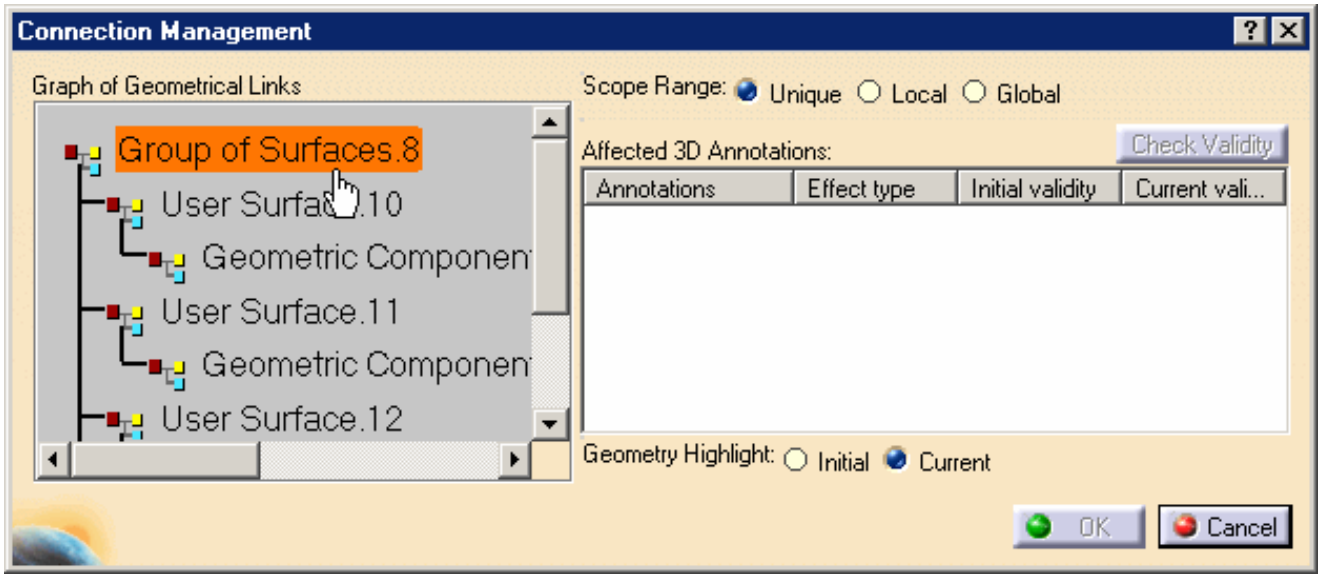
The **Connection Management** dialog box displays the new connected geometry: Geometric Component.1



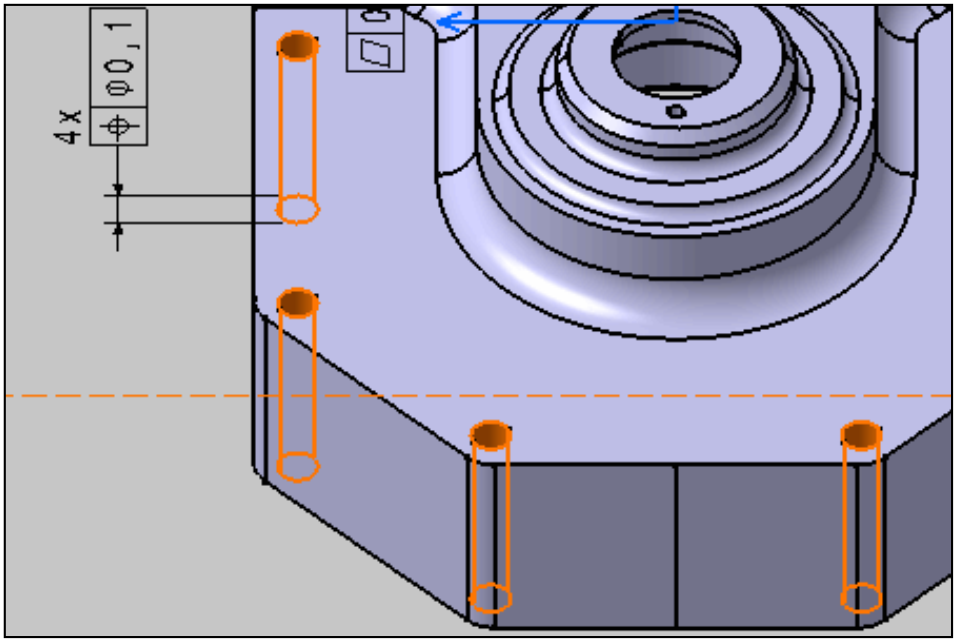
5. Click the **Check Validity** command to check the new geometry component validity relative to the selected annotation.



6. Select **Group of Surfaces.8** in the **Graph of Geometrical Links** as shown and select **Current** option in **Geometry Highlight**.

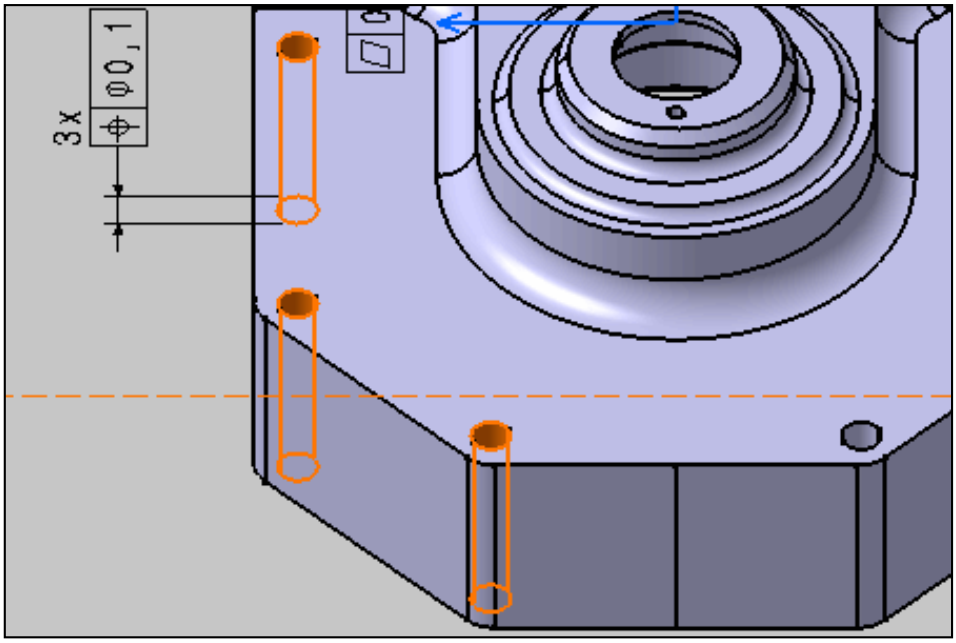


This option shows you the current group of surfaces of the selected annotation after the modification.



7. Select now **Initial** option in **Geometry Highlight**.

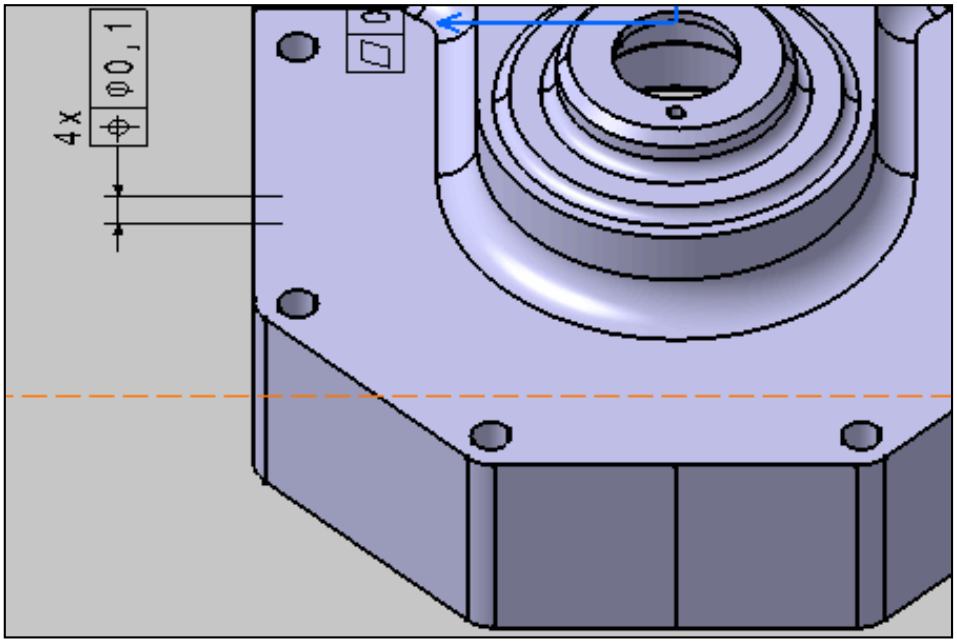
This option shows you the initial group of surfaces of the selected annotation before the modification.



Options in **Geometry Highlight** are always applied to the selection in the **Graph of Geometrical Links**.

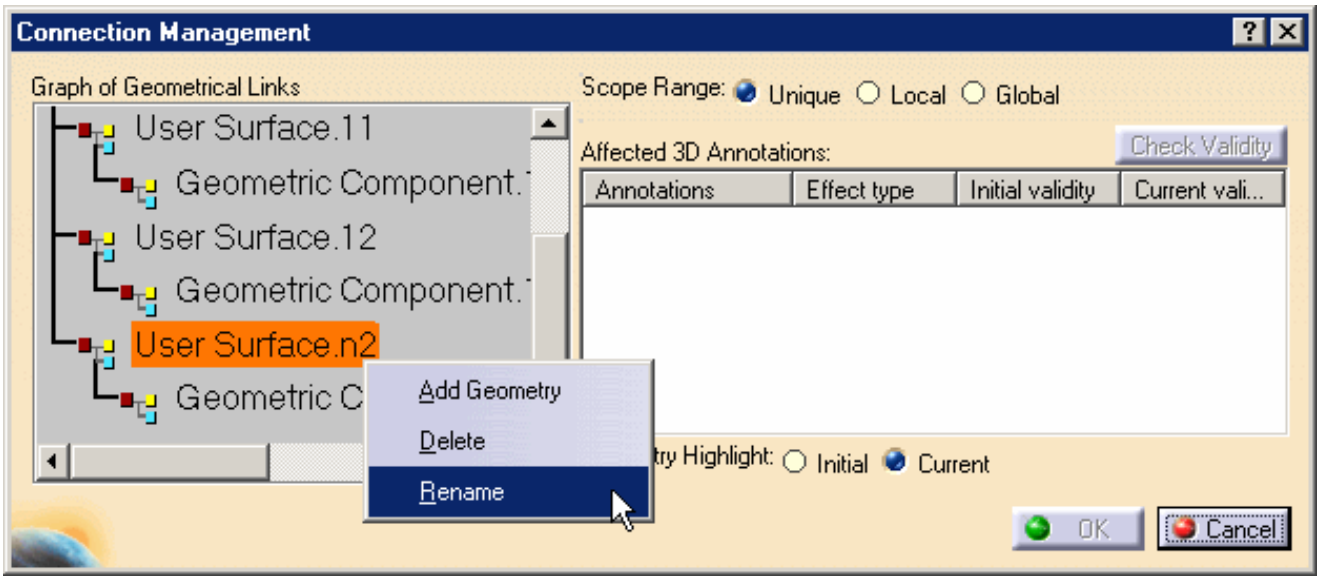
8. Click **OK**.

The four geometric components are now linked to the annotation.



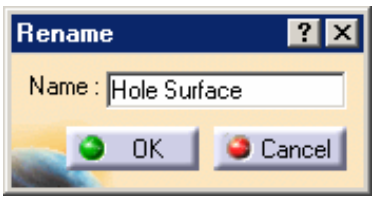
9. Right-click the annotation again and select the **Geometry Connection Management** contextual menu.

10. Right-click **Geometric Component.1** in the **Graph of Geometrical Links** as shown and select the **Rename** contextual menu.

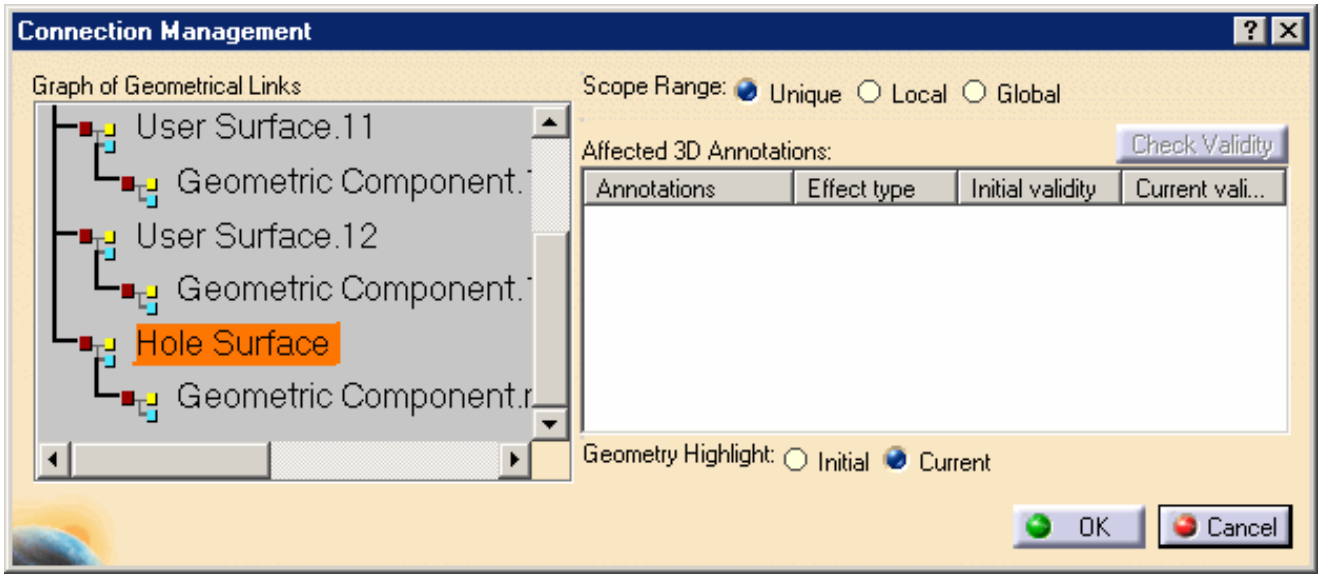


The **Rename** dialog box appears.

11. Enter the new name: Hole Surface and Click **OK**.



The feature has been renamed.



12. Click **OK**.

Back Up

Re-specifying Geometry Canonicity

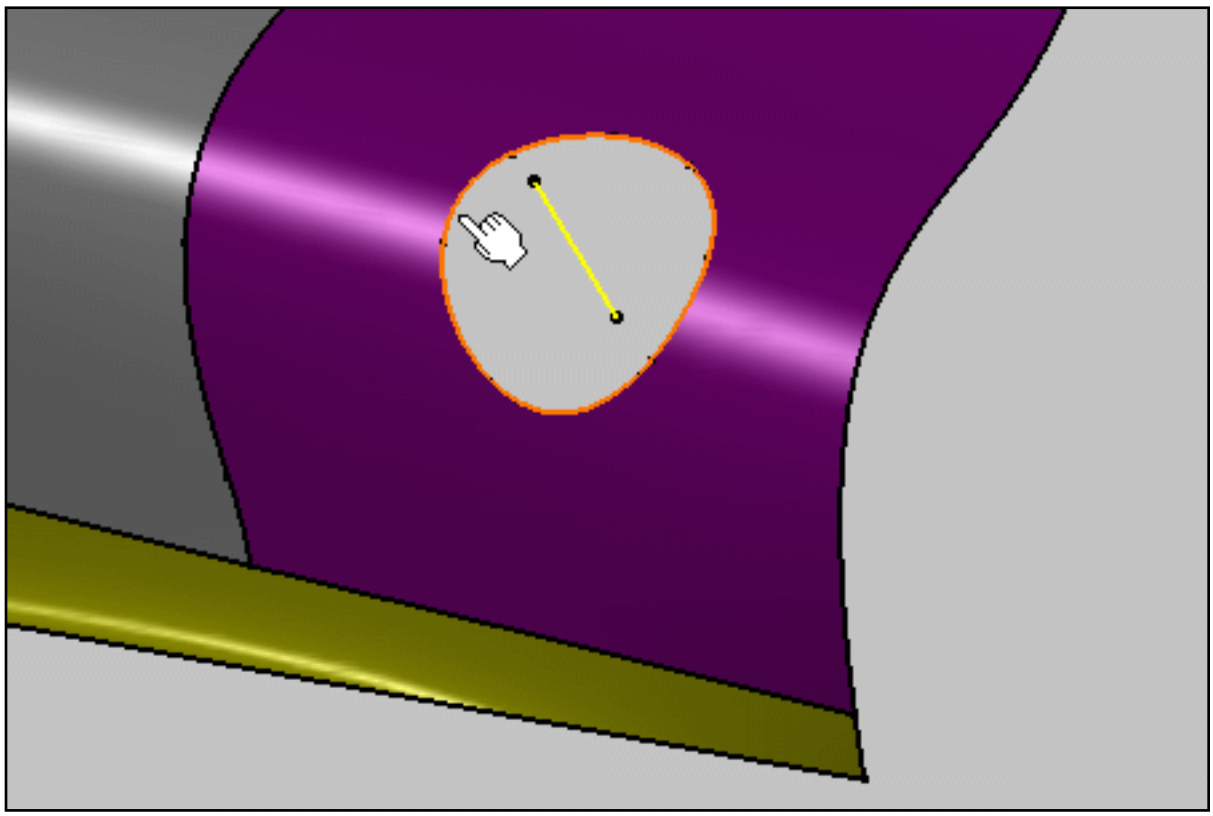
This task shows how to re-define a complex edge into a cylindrical surface and apply a size tolerance to the previous redefined geometry.

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.

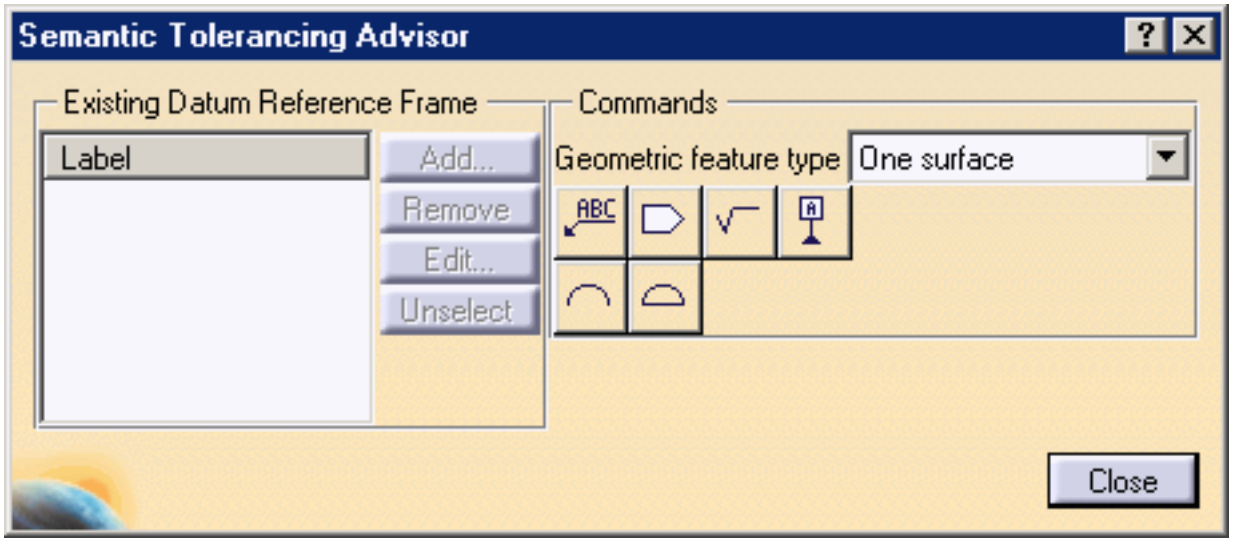
Open the Tolerancing_Annotations_07 CATPart document.

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.

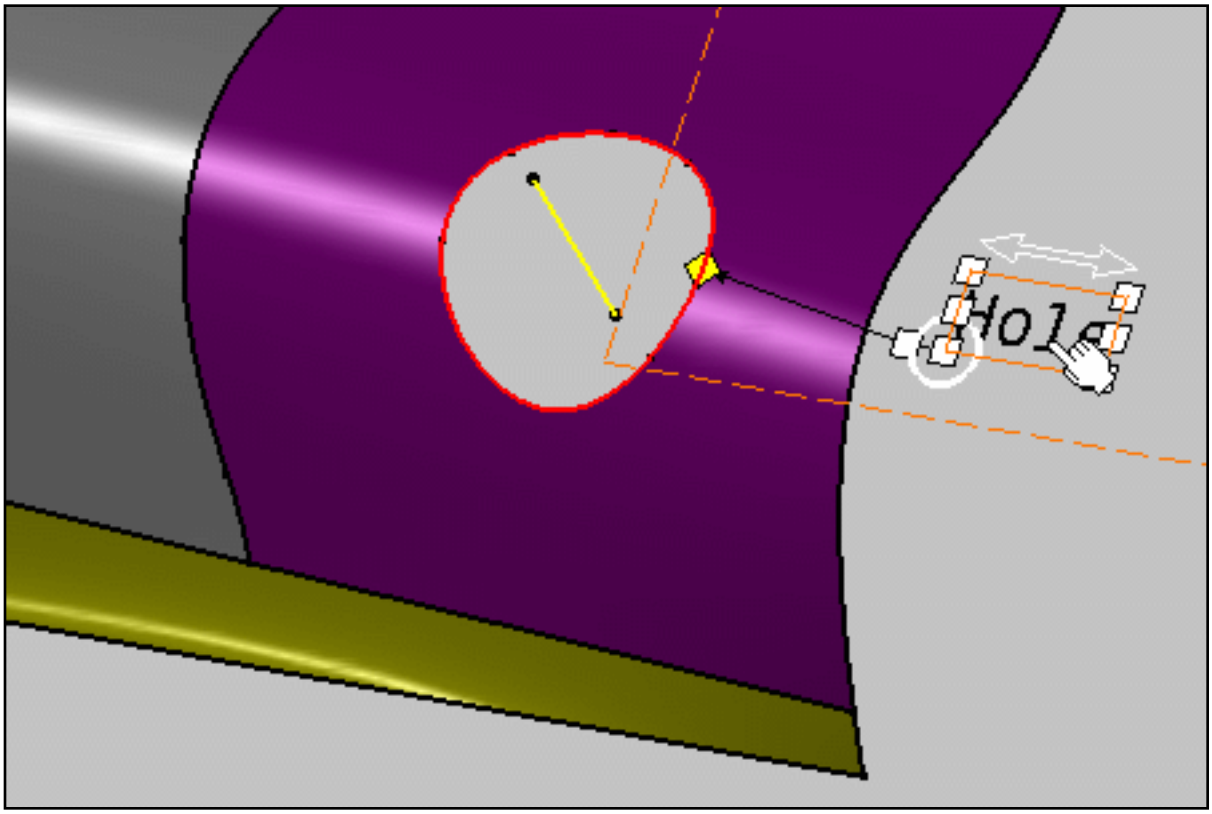


3. Click the **Text with Leader** icon (One surface): 

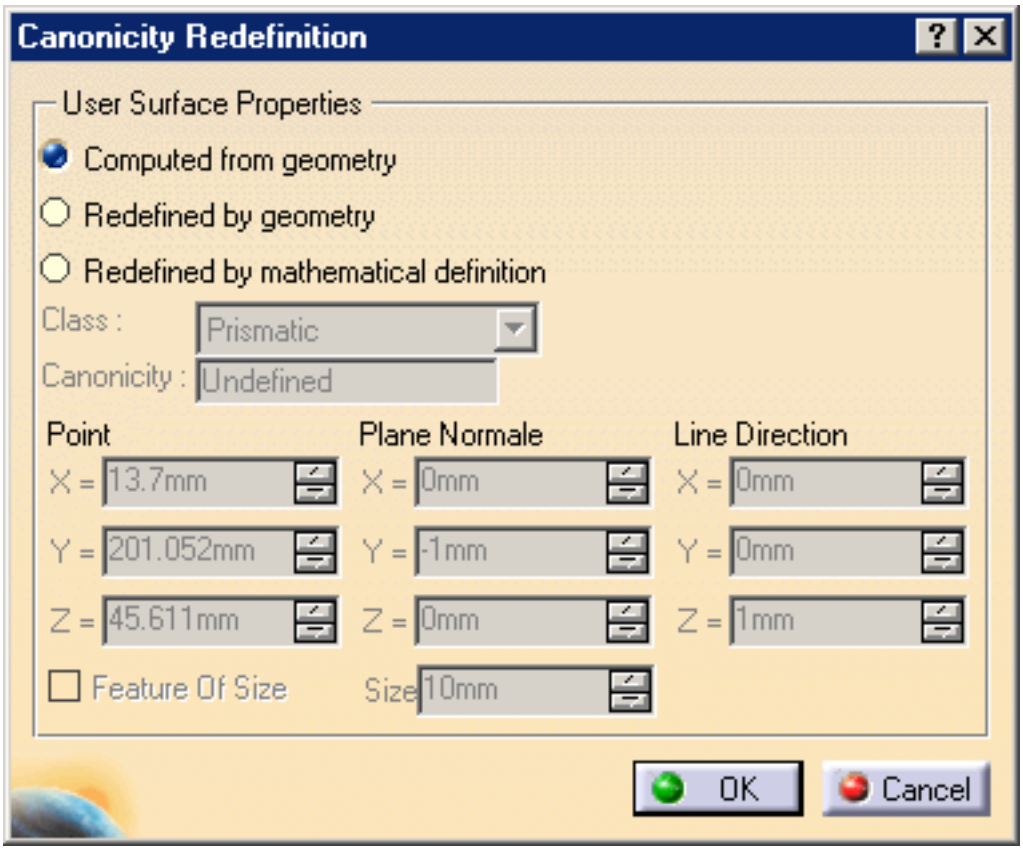
4. Enter Hole in the **Text Editor** dialog box field when appears.

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.

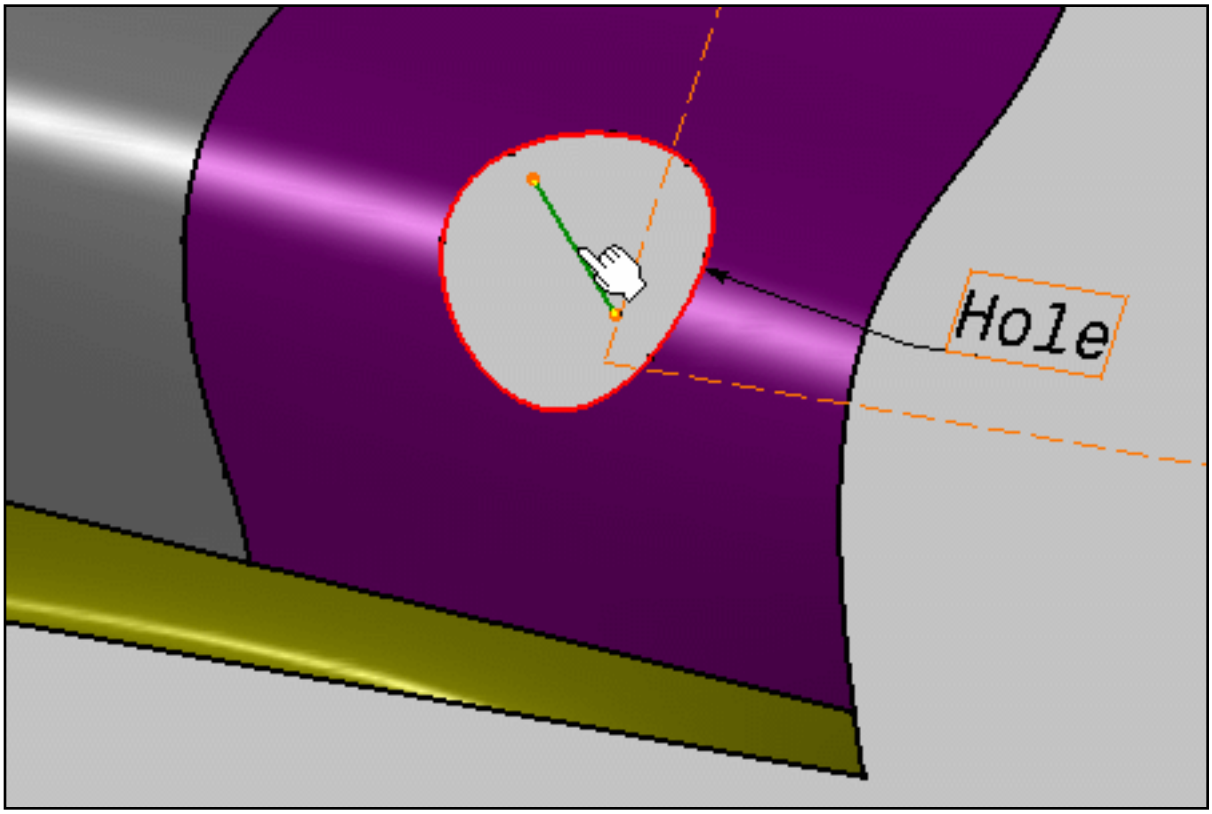


The **Canonicity Redefinition** dialog box appears.

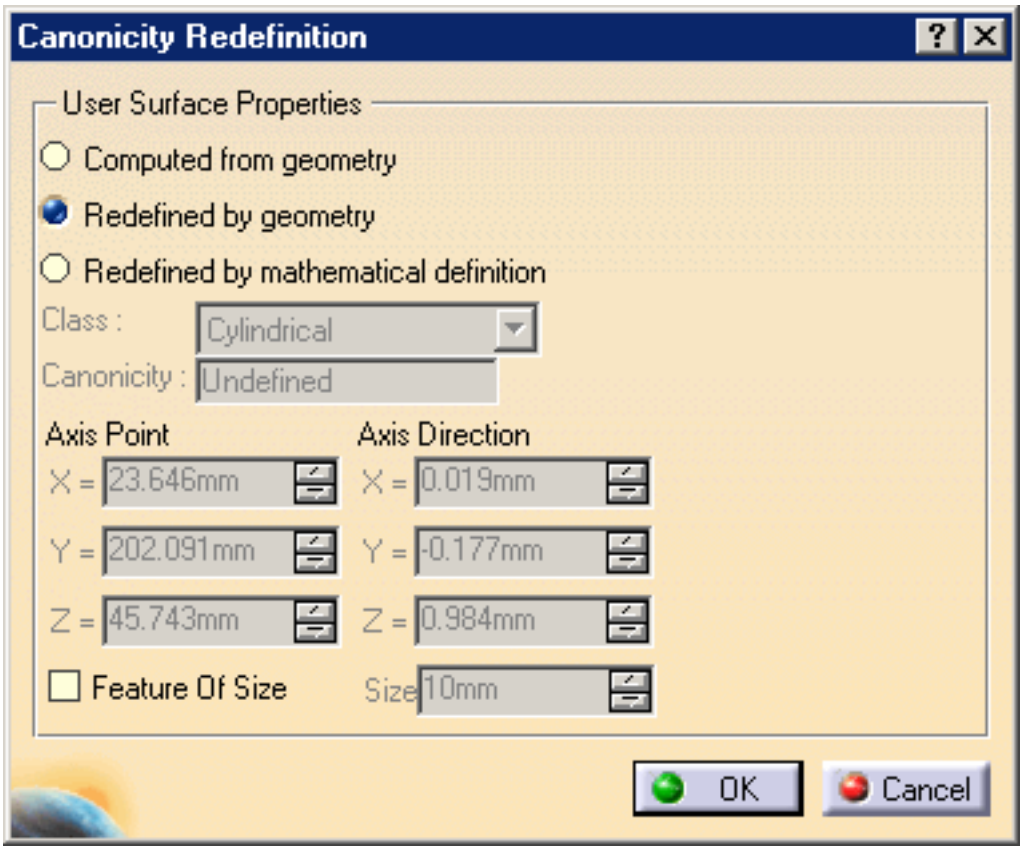


7. Select the **Redefined by geometry** option.

8. Select the line as shown on the part as axis line associated with the cylindrical surface definition.

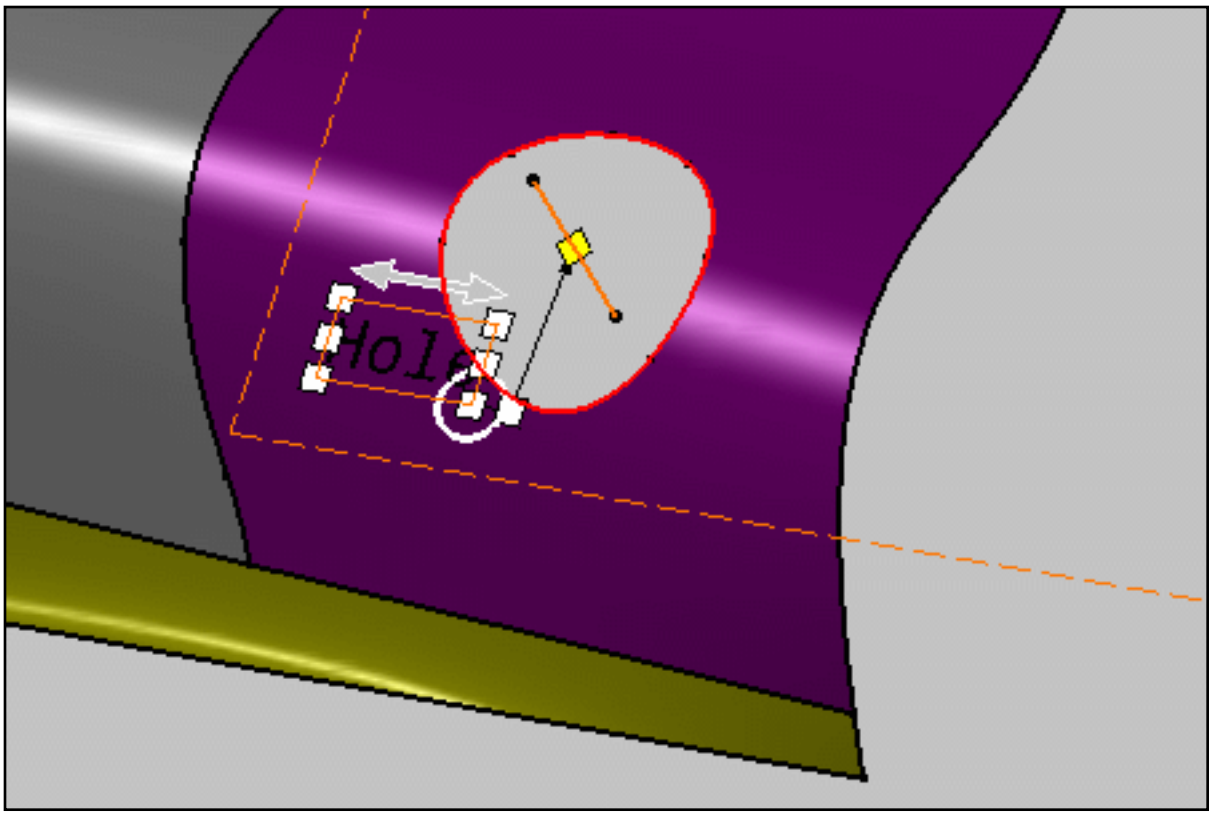


The **Canonicity Redefinition** dialog box is updated.



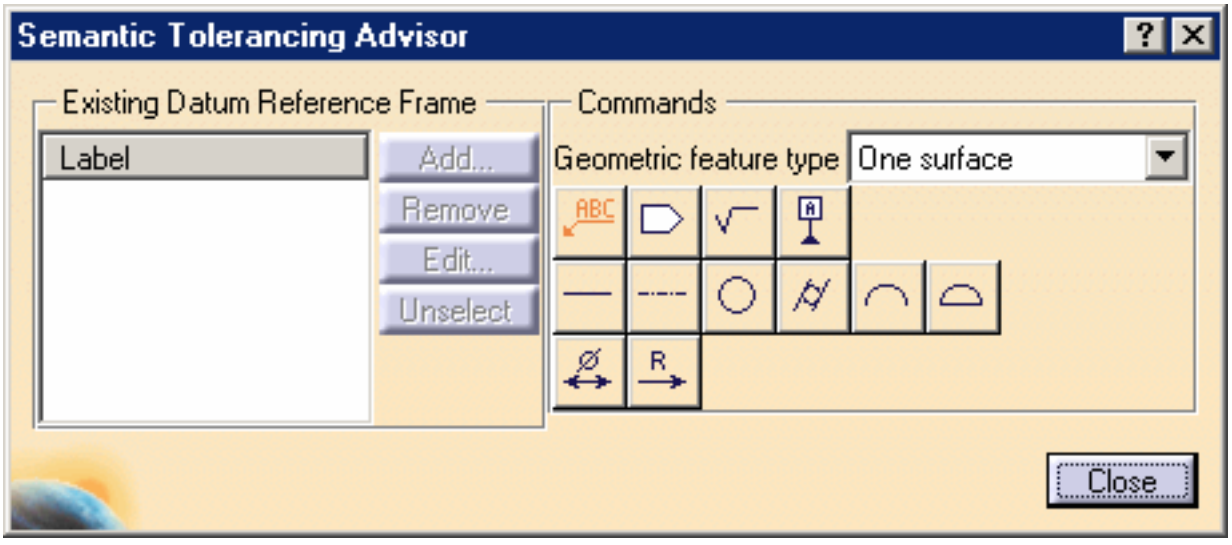
9. Select the **Feature of Size** option, the Size field is now enabled and set its value to 20mm (We "know" the cylindrical diameter) and click **OK**.

The annotation is now attached to the axis.



10. Click the **Tolerancing Advisor** icon: , the annotation is still selected.

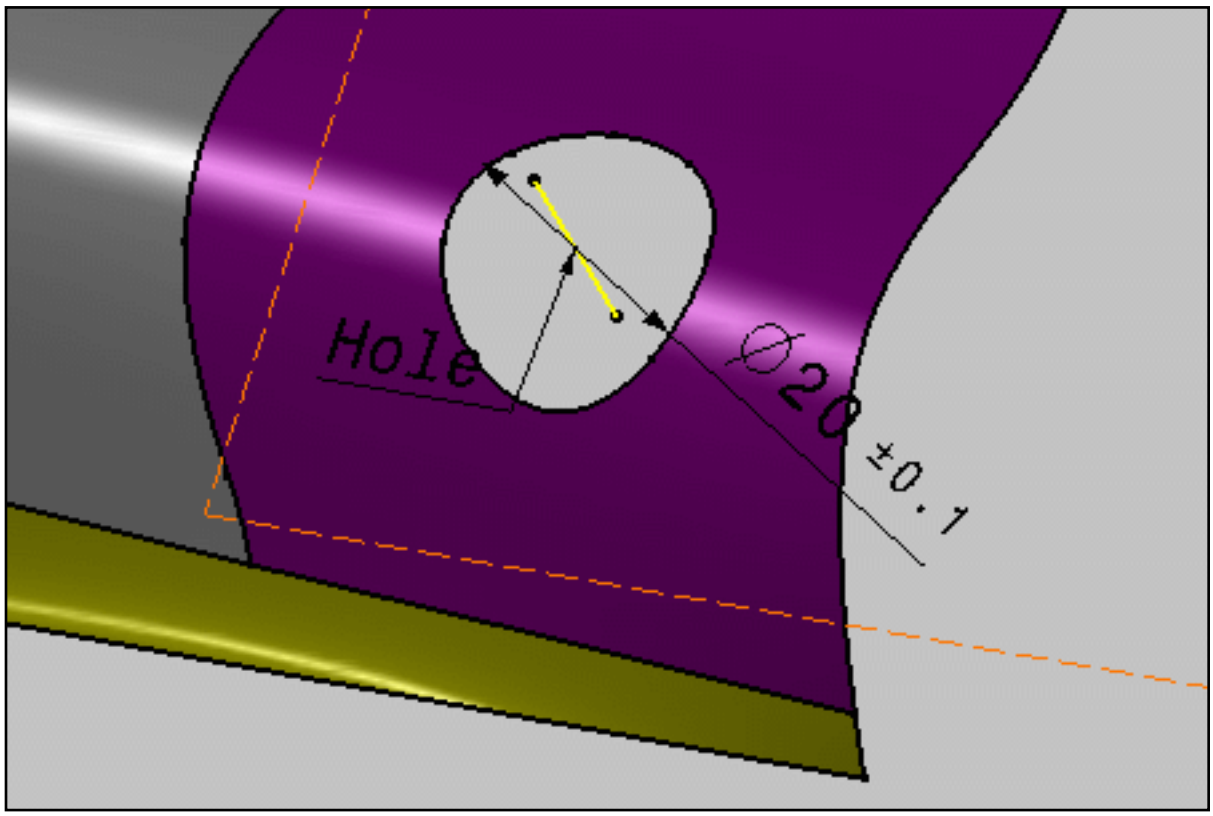
The **Semantic Tolerancing Advisor** dialog box displays now command related to a cylindrical surface.



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.



Back Up Next

Reporting Annotations



Generate a Check Report: click this icon to generate the report.



Customize the Reporting: click this icon and select the desired options.

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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 Tolerancing_Annotations_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.

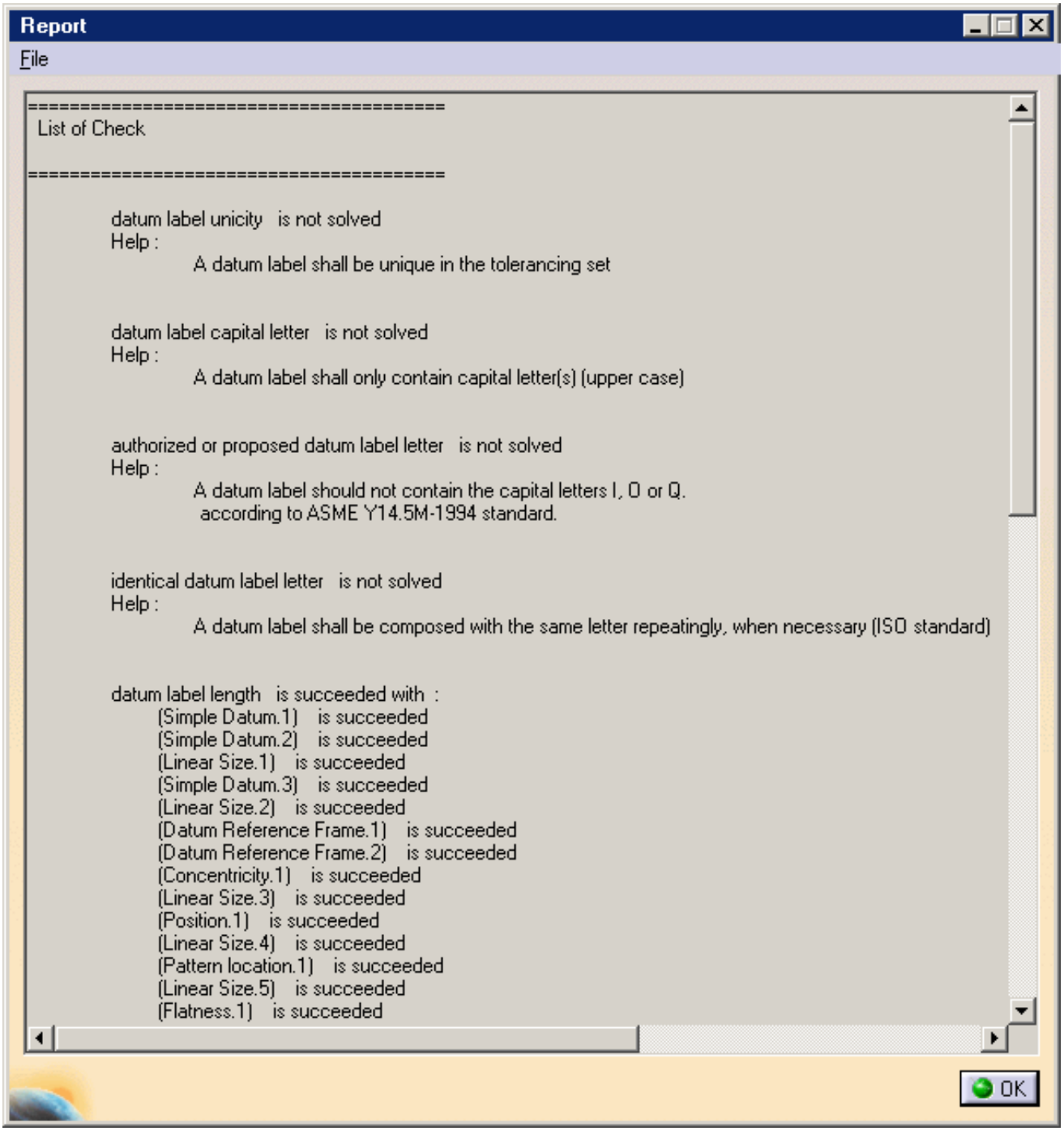
   The file provides you with the path of the CATPart document you are using and the date of generation. In our example, all the rules are respected as mentioned by the green symbols and the 100% success message.

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.

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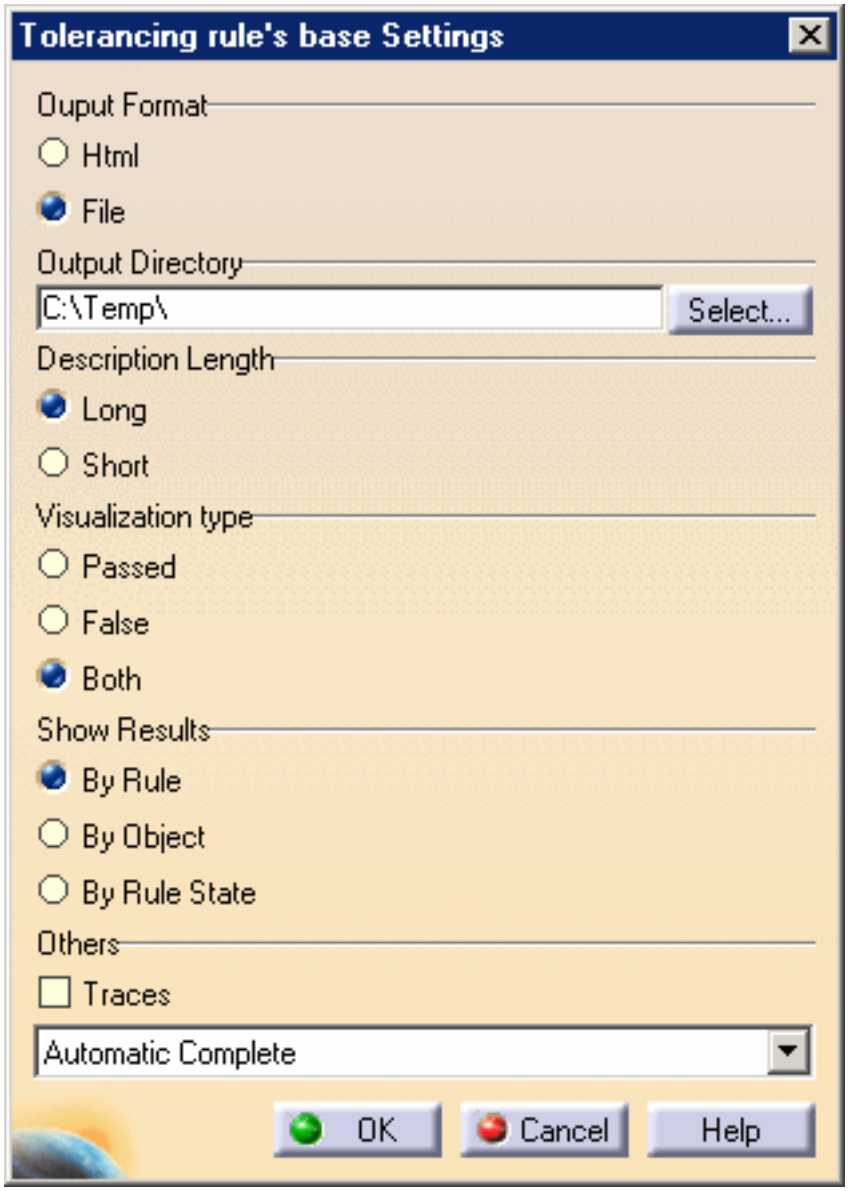
Customizing the Reporting

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.



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Annotation Associativity

Different types of associativity characterize the 3D Functional Tolerancing and Annotation workbench, 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: click this icon, select the annotation.

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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:

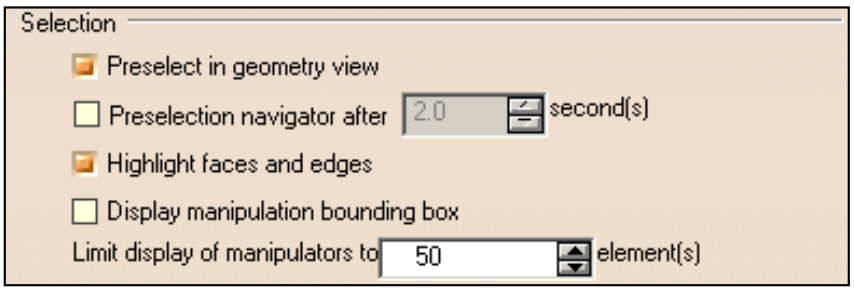
- The geometrical element itself.
- All the annotations that are applied (directly or indirectly) to it.
- All the framed (basic) dimensions that are related to the corresponding specifications.
- All the datum feature annotations that are related to the specifications (for semantic geometric tolerance only).

Selecting a Part Design or Generative Shape Design feature or a restricted area in the specification tree will highlight:

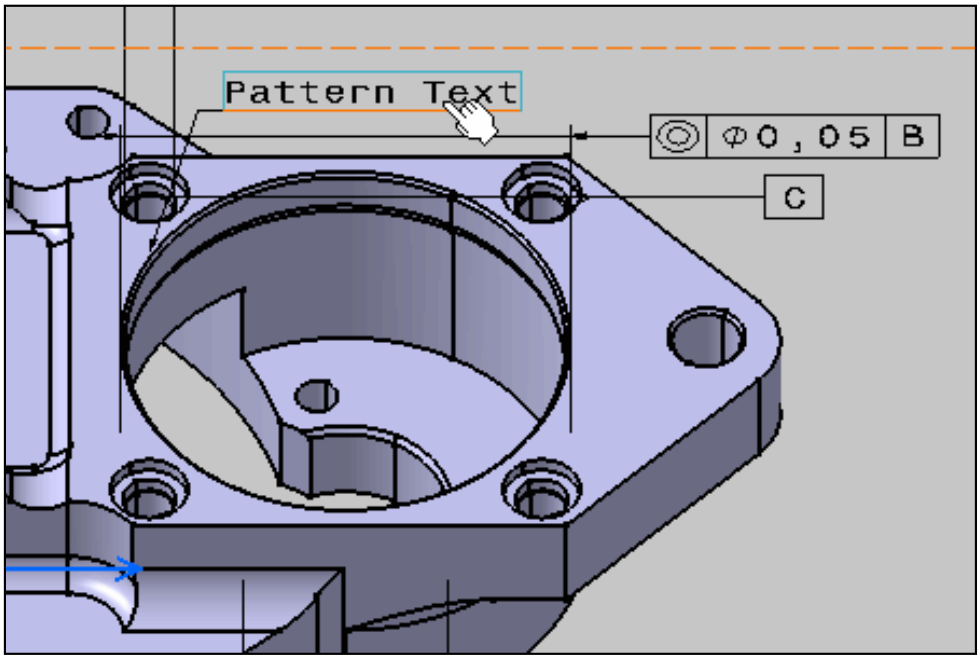
- The feature itself.
- All the annotations that are applied (directly or indirectly) to all its geometrical components.
- All the framed (basic) dimensions that are related to the specification.
- All the framed (basic) dimensions that are related to the corresponding specifications.
- All the datum feature annotations that are related to the specifications (for semantic geometric tolerance only).

Open the Tolerancing_Annotations_02 CATPart document, and make sure that the **3D Annotation Query Switch On/Switch Off** icon  is activated.

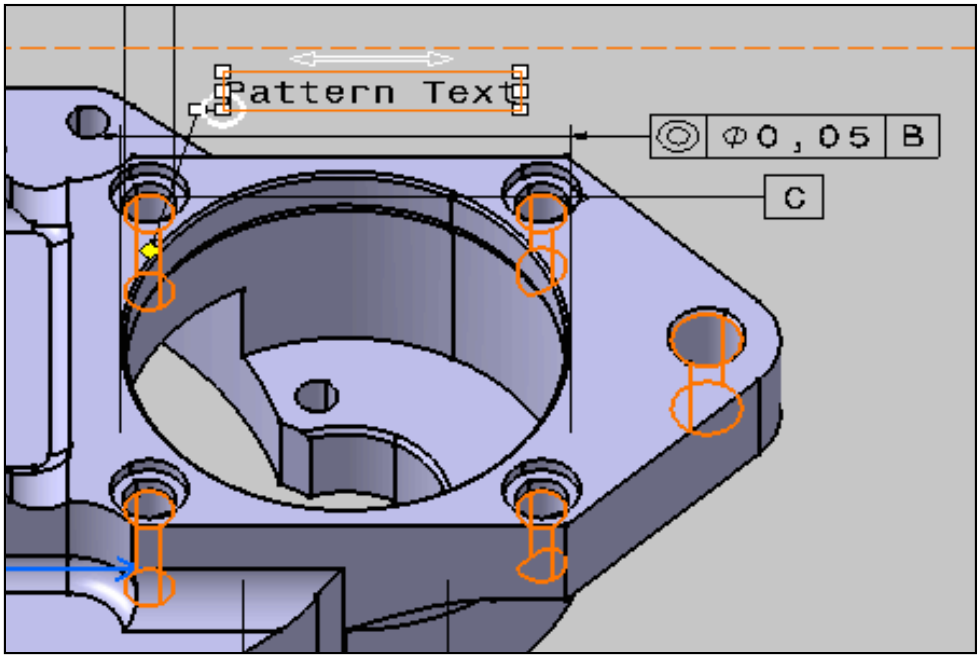
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



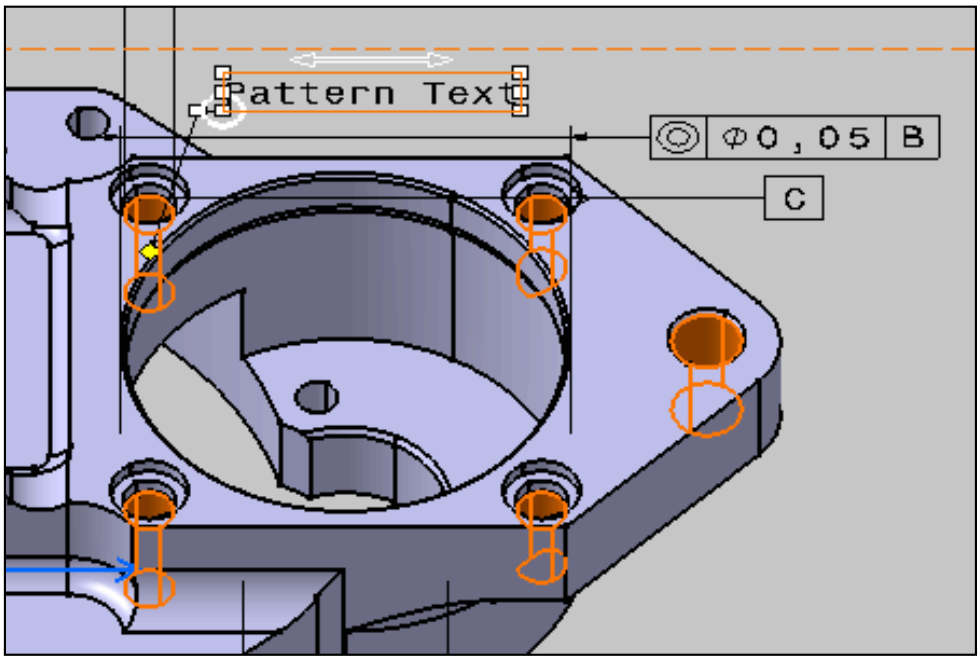
1. Select the annotation as shown:



This location specification is defined on a pattern of holes. Any toleranced hole of the pattern is then highlighted.

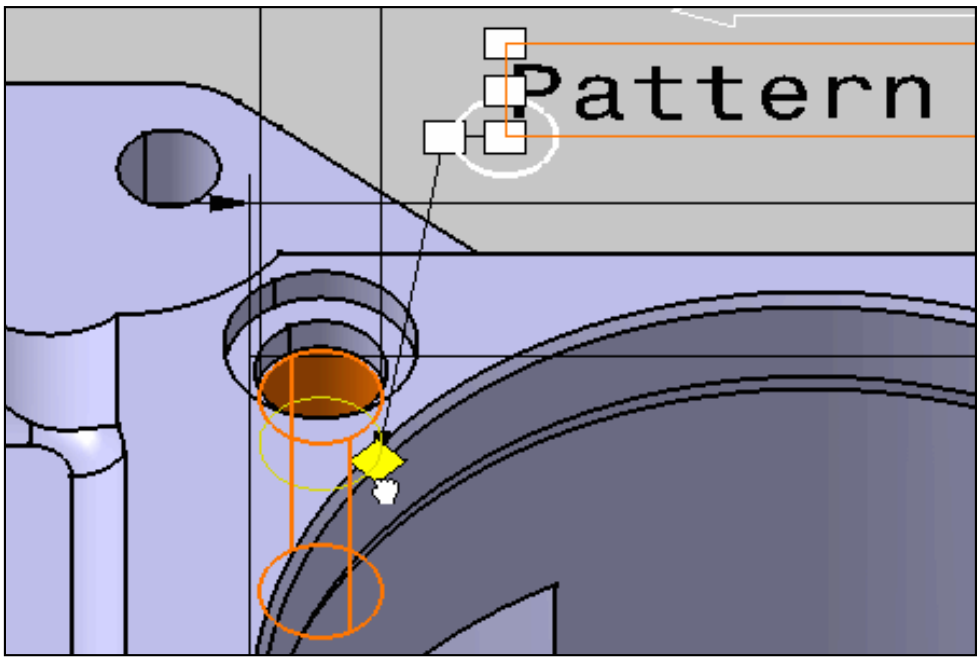


Any toleranced face hole of the pattern is then highlighted.

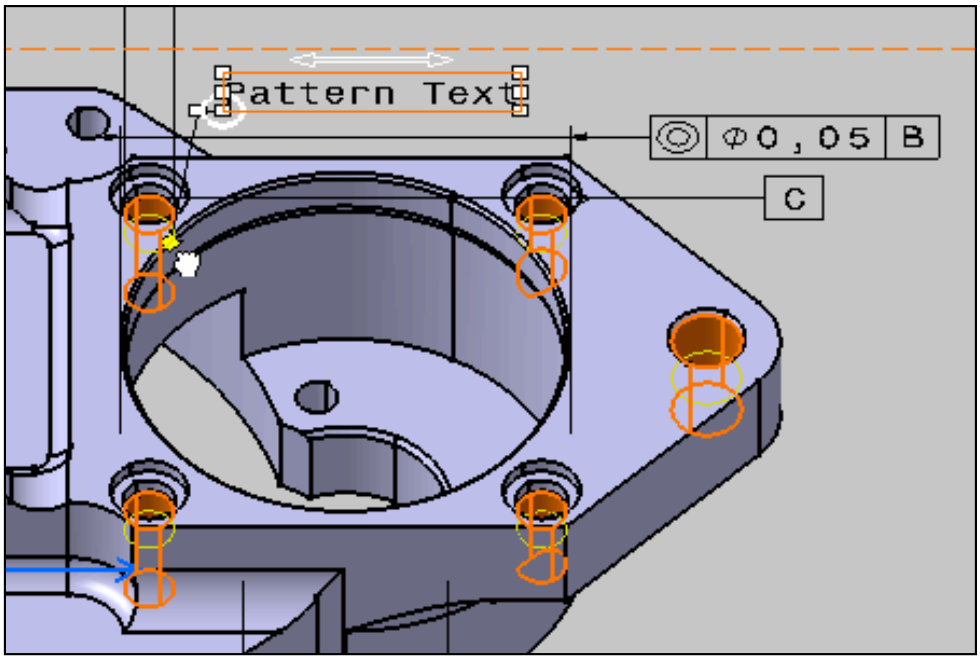


2. Select the yellow manipulator and start dragging the arrow head.

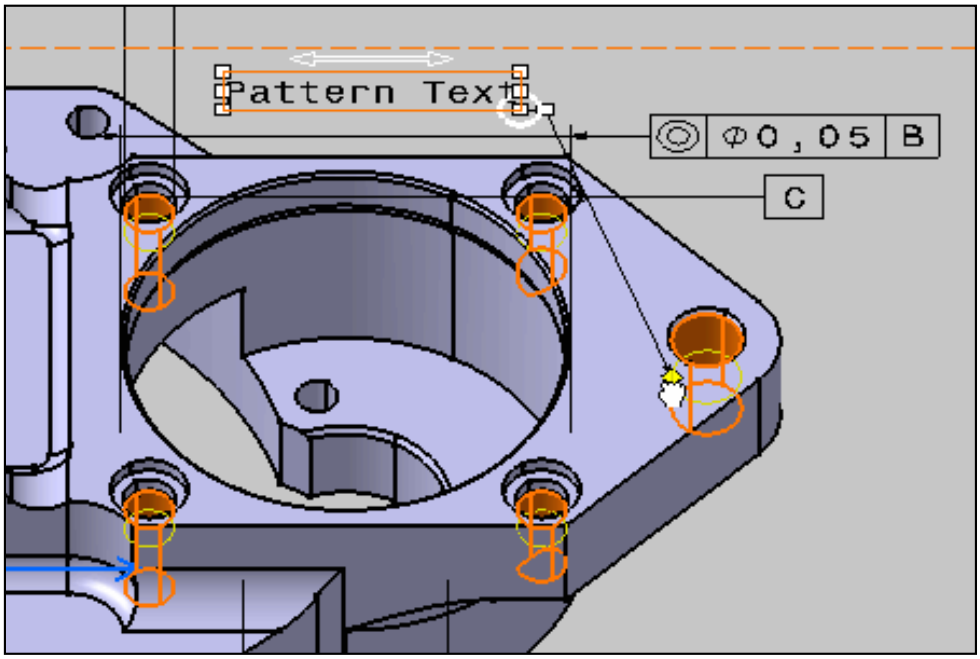
A yellow temporary trace is displayed. It corresponds to the intersection between the annotation plane and the corresponding toleranced elements. This trace defines all the possible positions for the arrow head that are semantically correct for the selected tolerancing annotation.



In our example, the toleranced elements are a group of five holes, this is why we obtain five yellow traces:

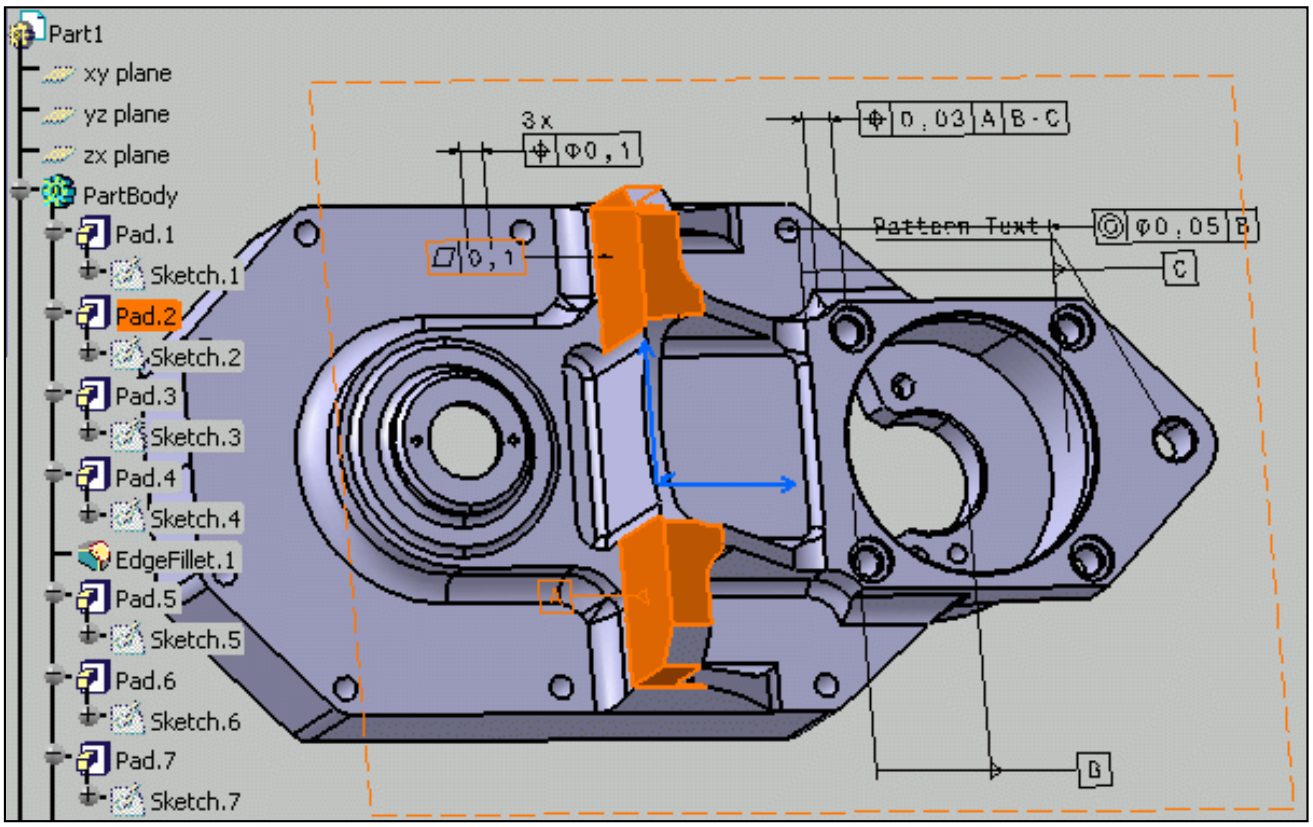


3. For example, drag and drop the arrow head on another trace to the right. You can obtain this:



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*.

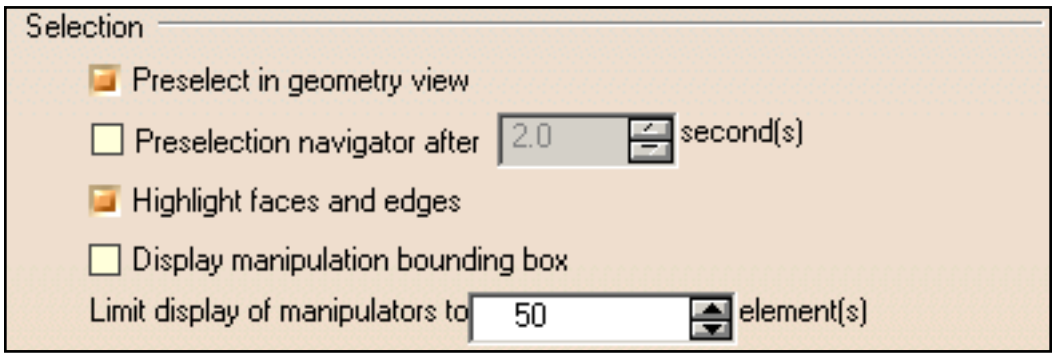
Up Next

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.

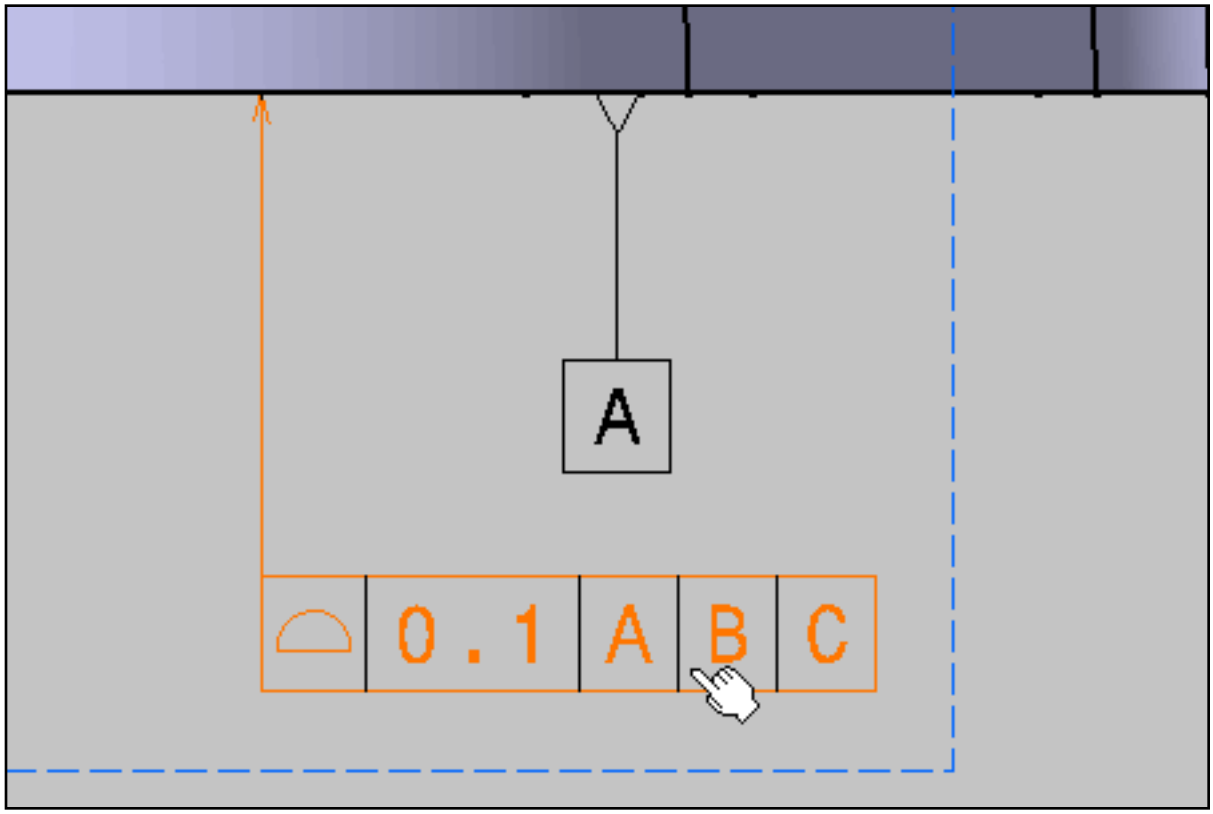
Open the Tolerancing_Annotations_03 CATPart document.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.

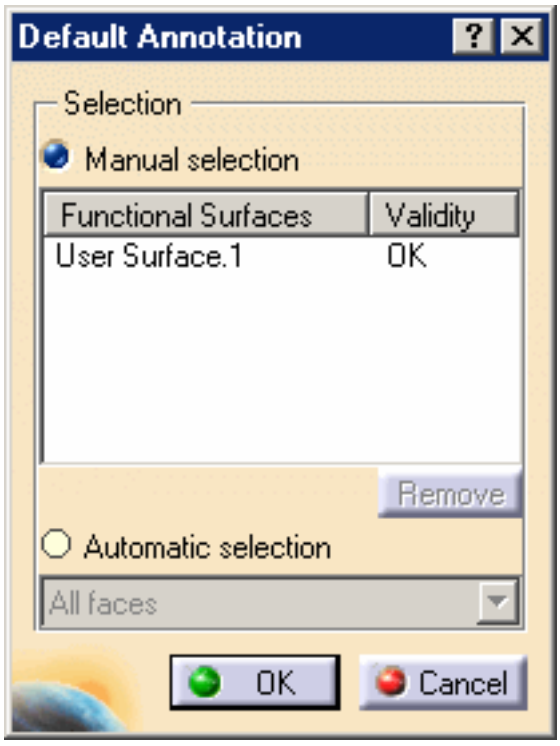


1. Select the **Insert -> Annotations -> Default Annotation** menu.

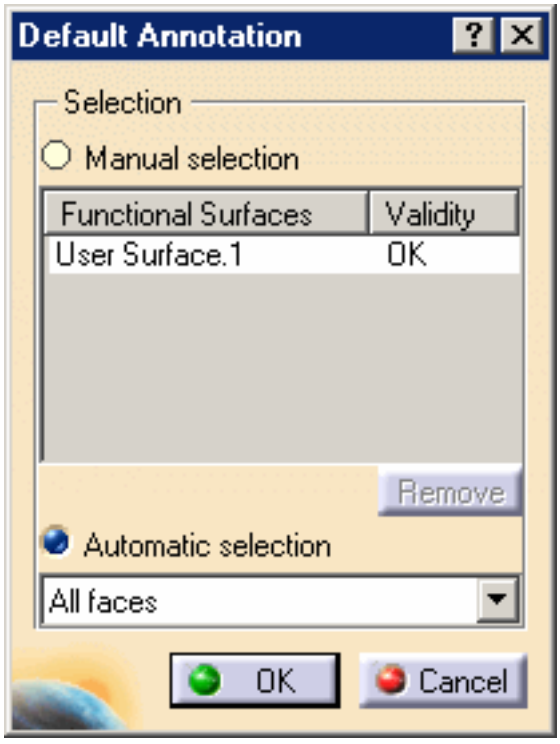
2. Select the **Position surfacic profile.1** annotation as shown



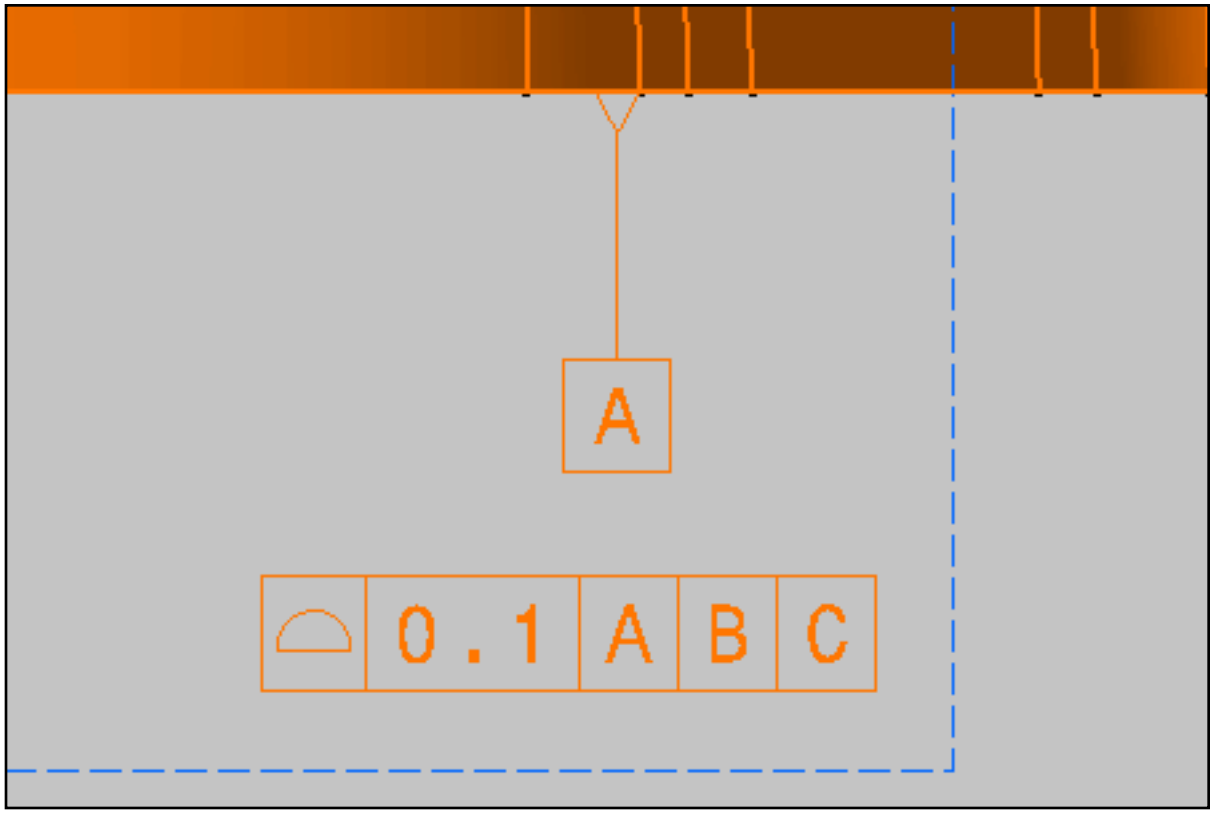
The **Default Annotation** dialog box appears.

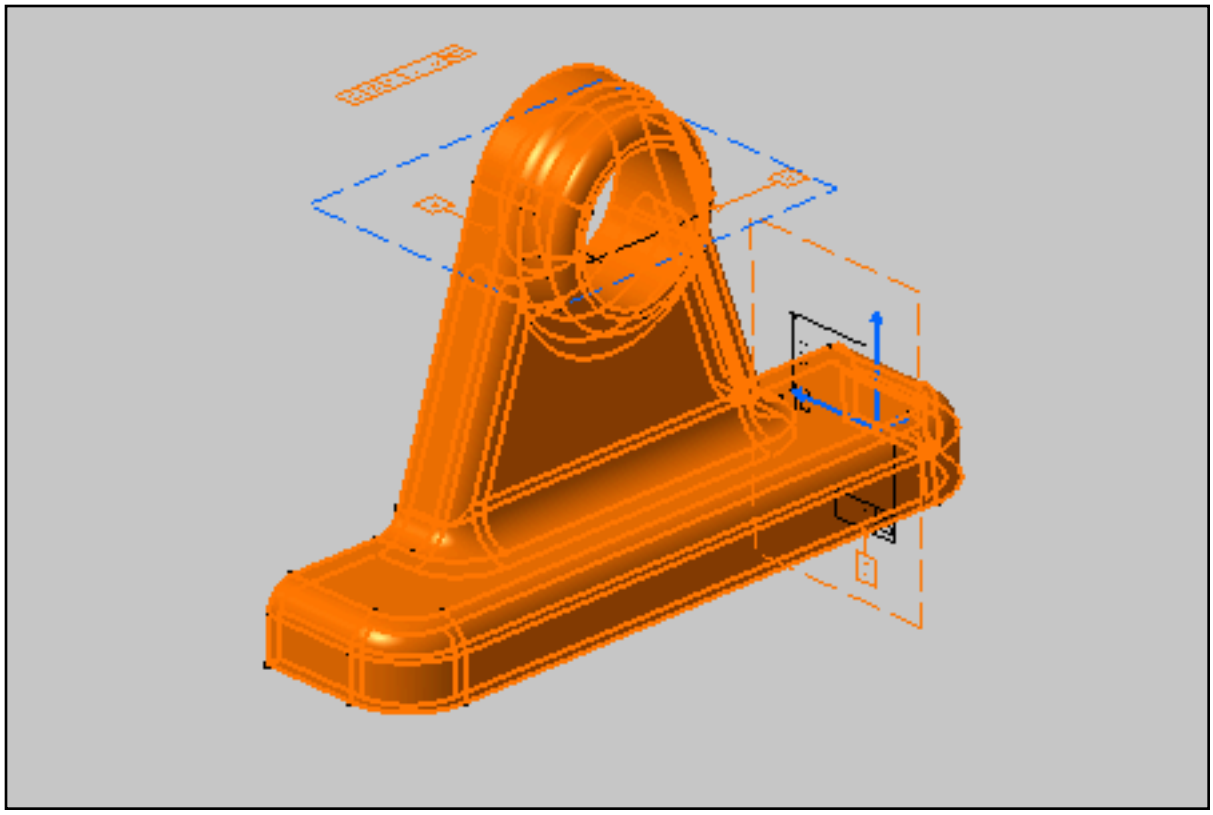


3. Select **Automatic** in the **Default Annotation** dialog box, **All faces** option in the combo box and click **OK**.



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



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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.

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Creating Power Copy

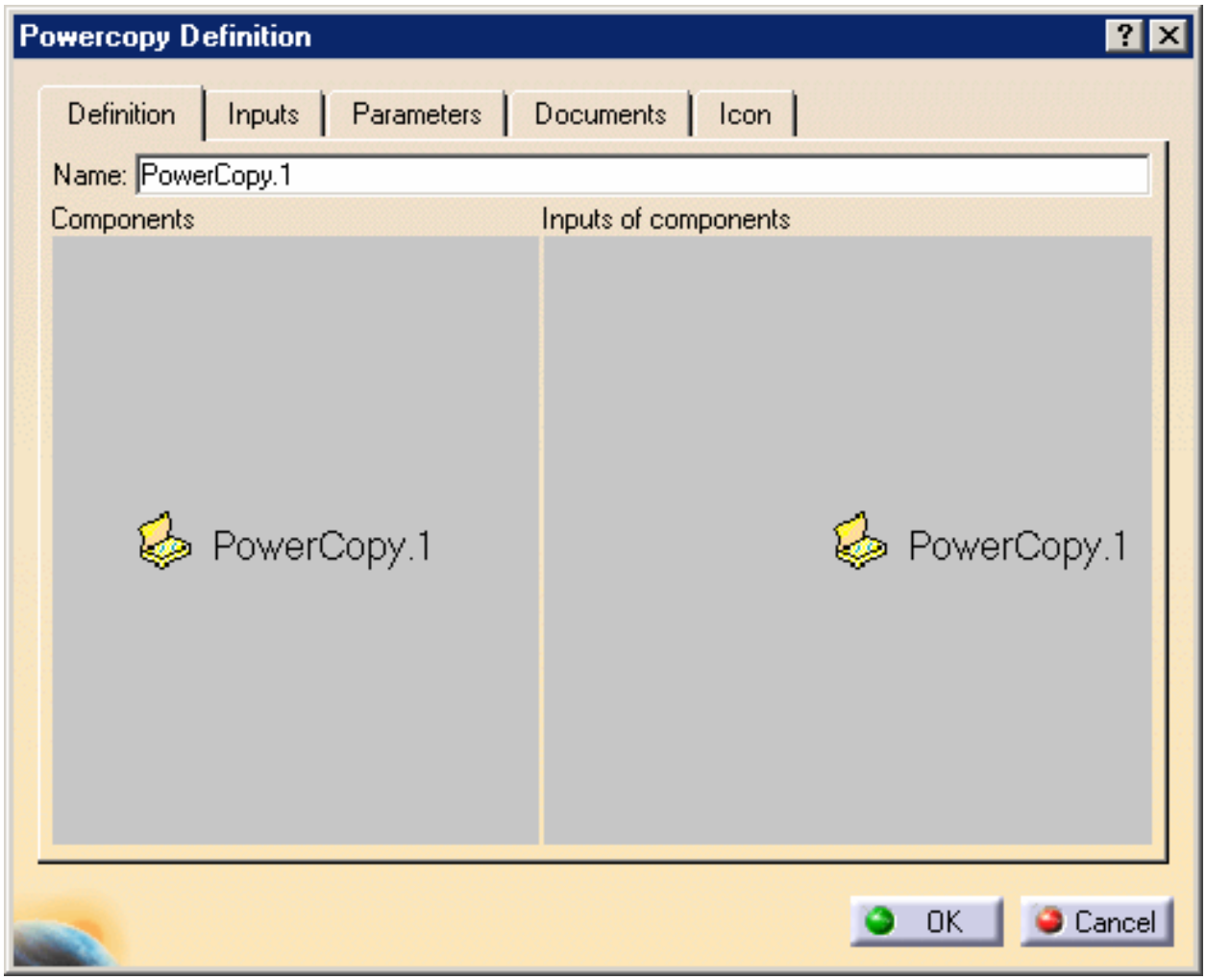
This task shows how to create Power Copy elements, to be reused later.

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.

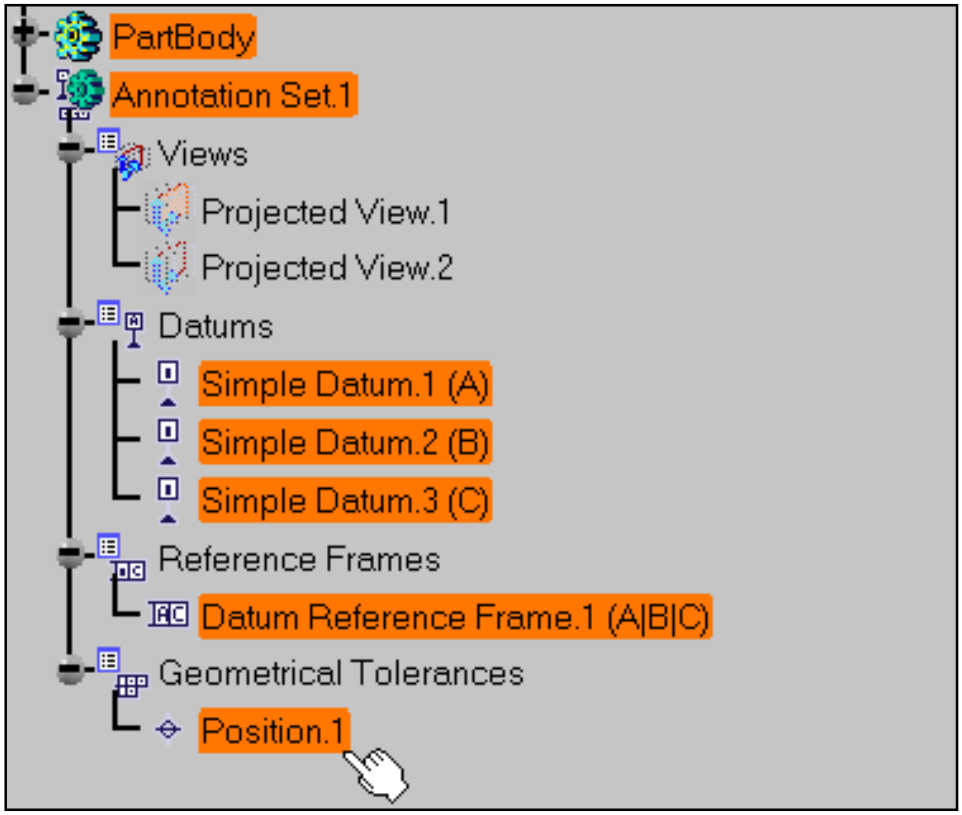
Open the PowerCopy1.CATPart document.

1. Select the **Insert ->Advanced Replication Tools -> Powercopy Creation** command.

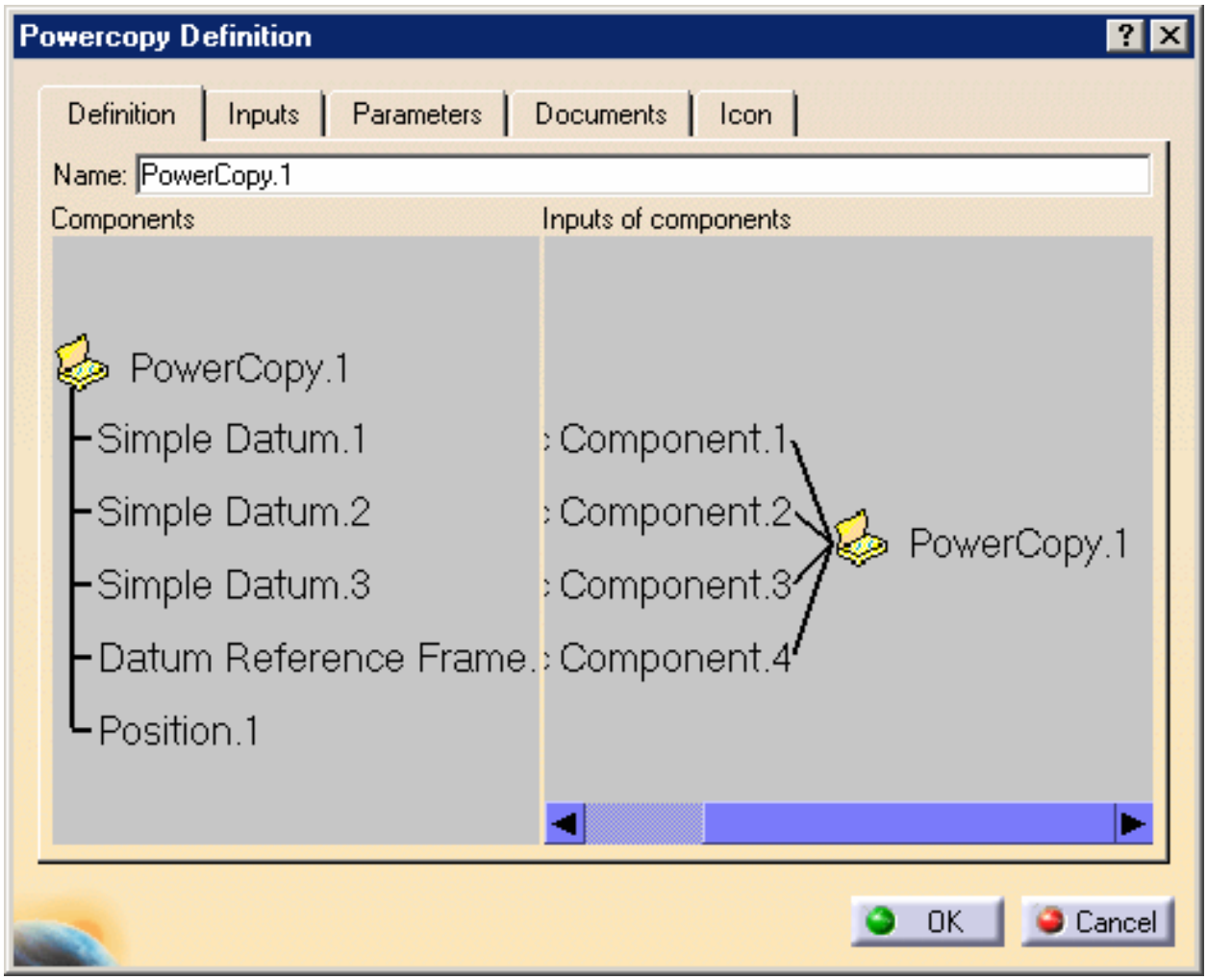
   The **Powercopy Definition** dialog box appears.



2. Select the following elements making up the PowerCopy from the specification tree.

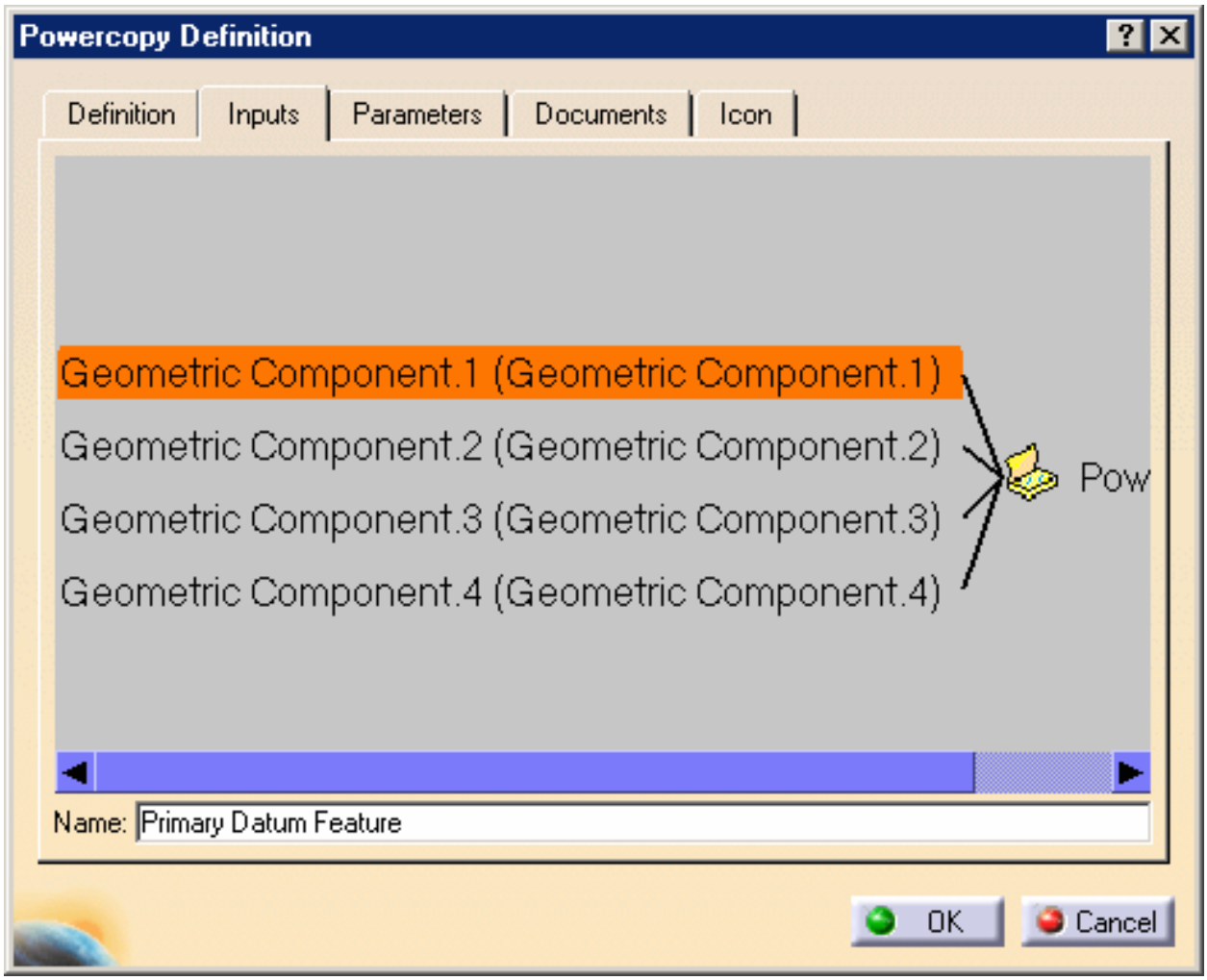


The **Powercopy Definition** dialog box is updated.

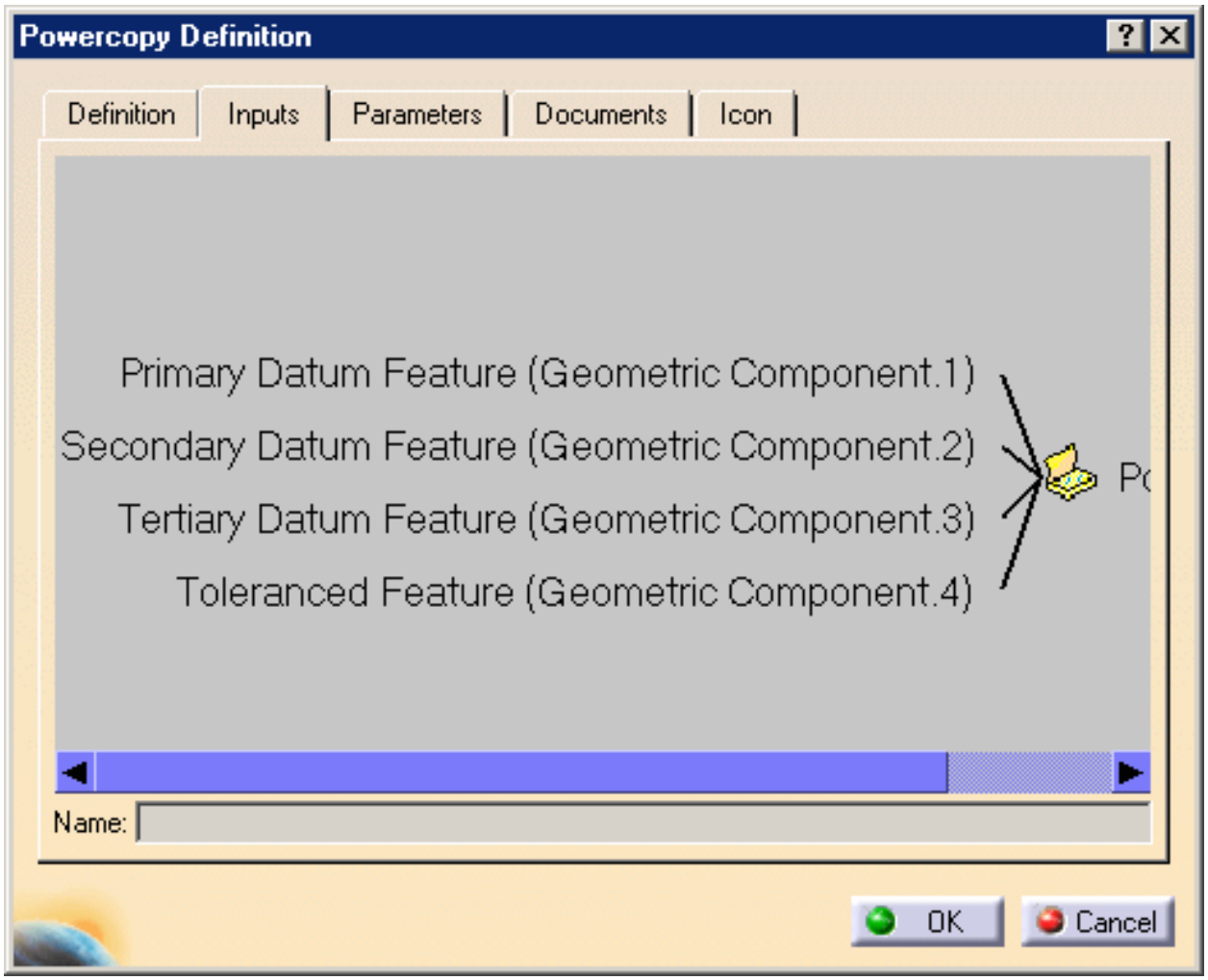


3. Select the **Inputs** tab.

4. Select the **Geometric Component.1** input and rename it into Primary Datum Feature.

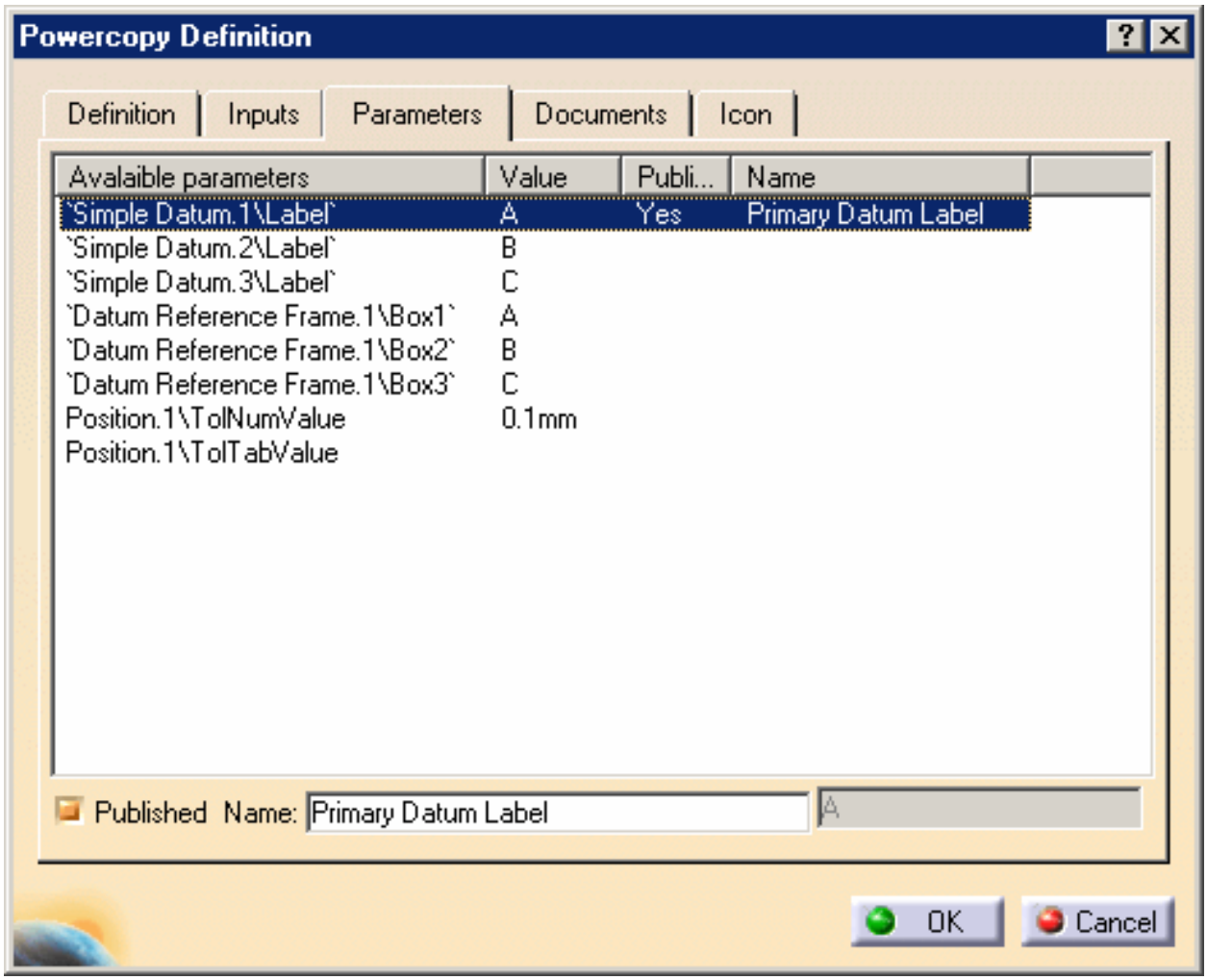


5. Repeat the operation with others inputs like this.

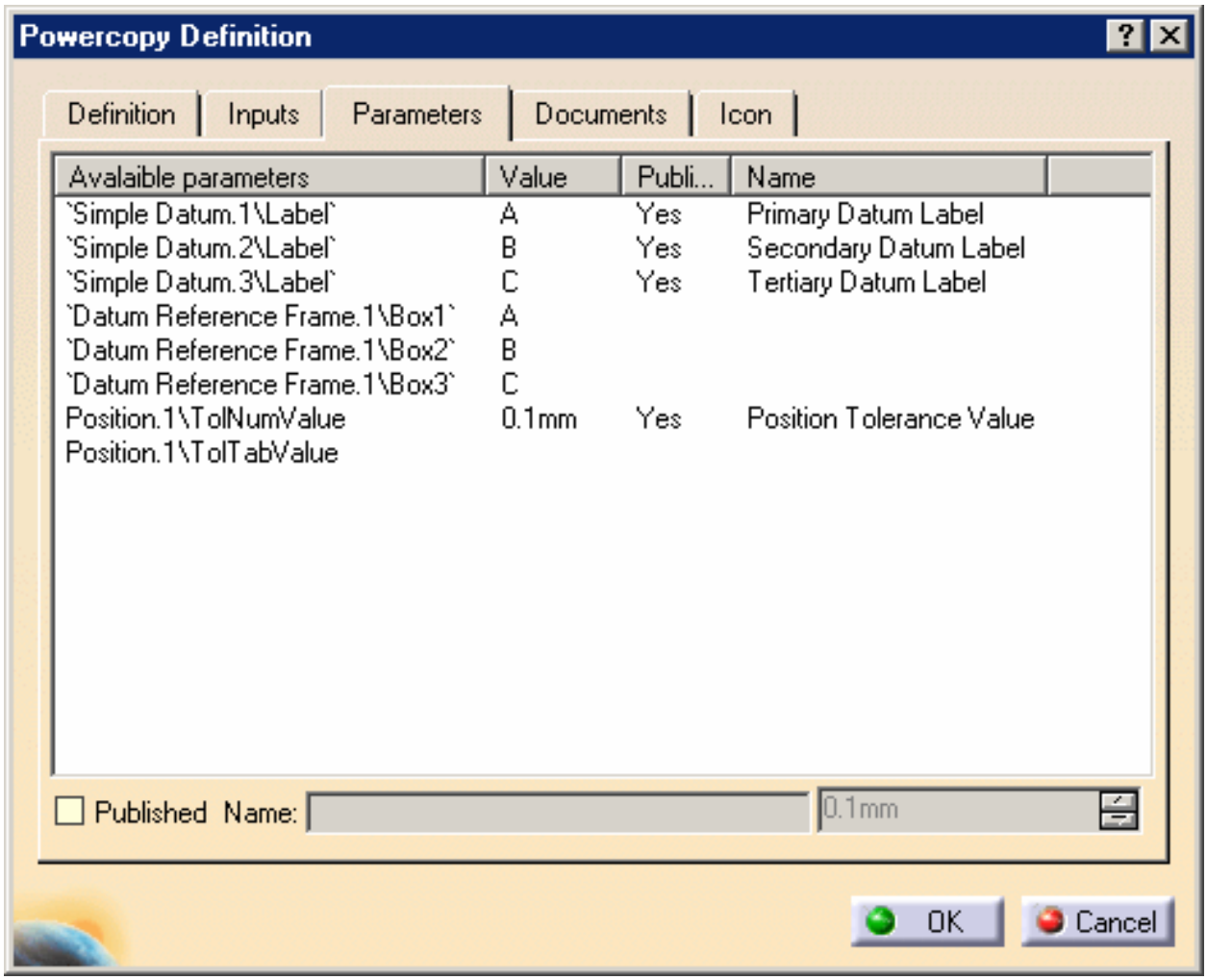


6. Select the **Parameters** tab.

7. Select the **Geometric Component.1** parameters and check **Publish Name** to rename it into Primary Datum Label.

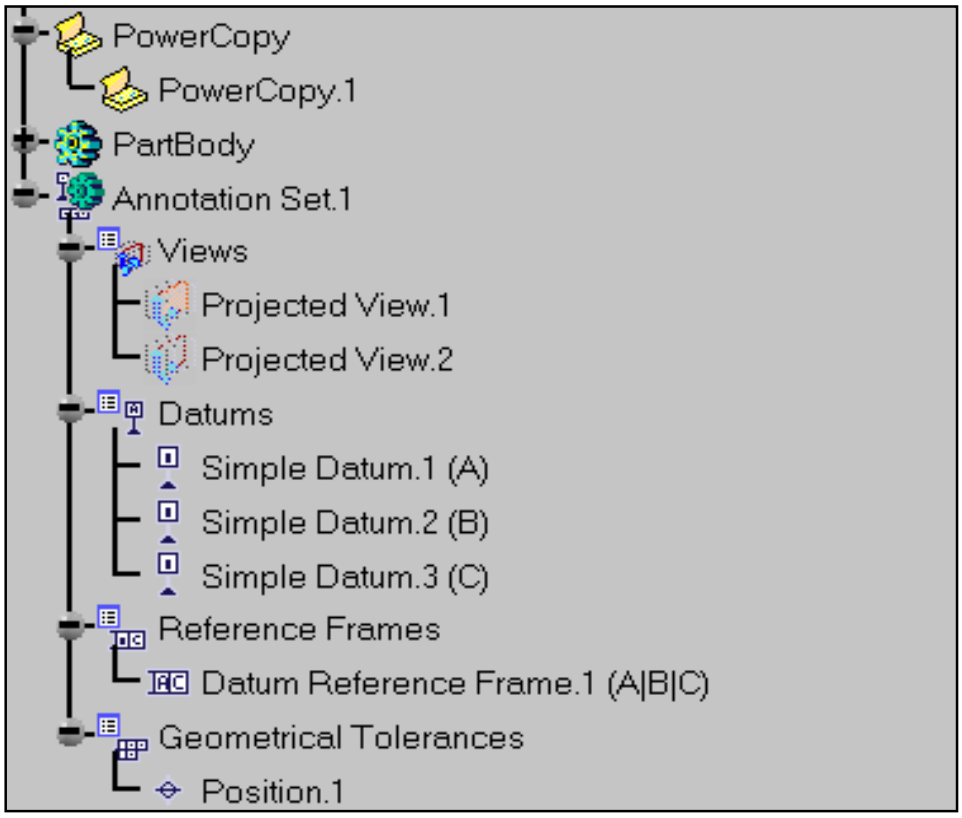


8. Repeat the operation with the following parameters like this.



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

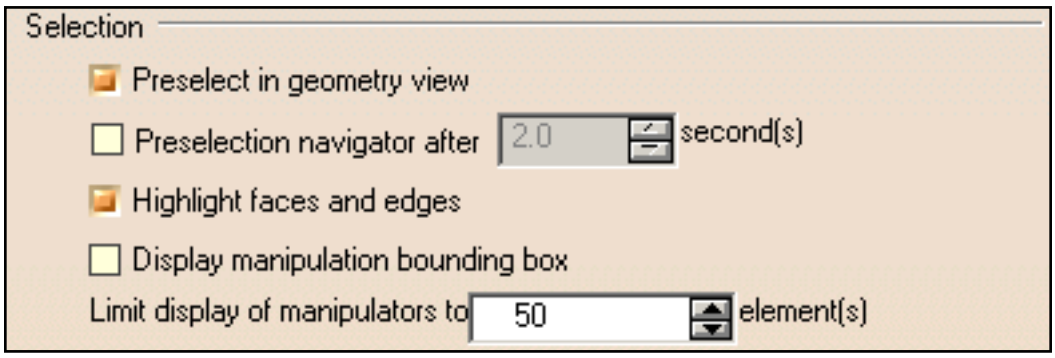
Up Next

Instantiating Power Copy

This task shows how to instantiate Power Copies once they have been created.
See Creating Power Copy.

Open the Tolerancing_Annotations_01 CATPart document.

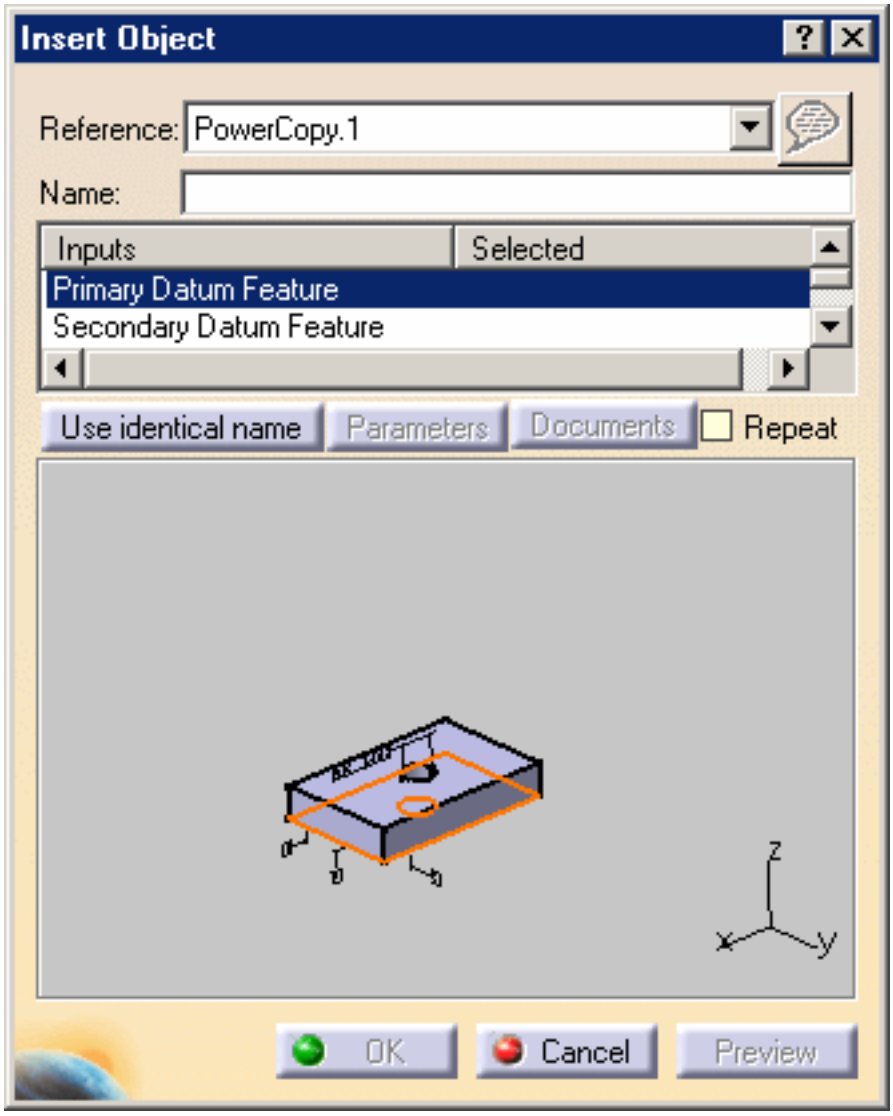
To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



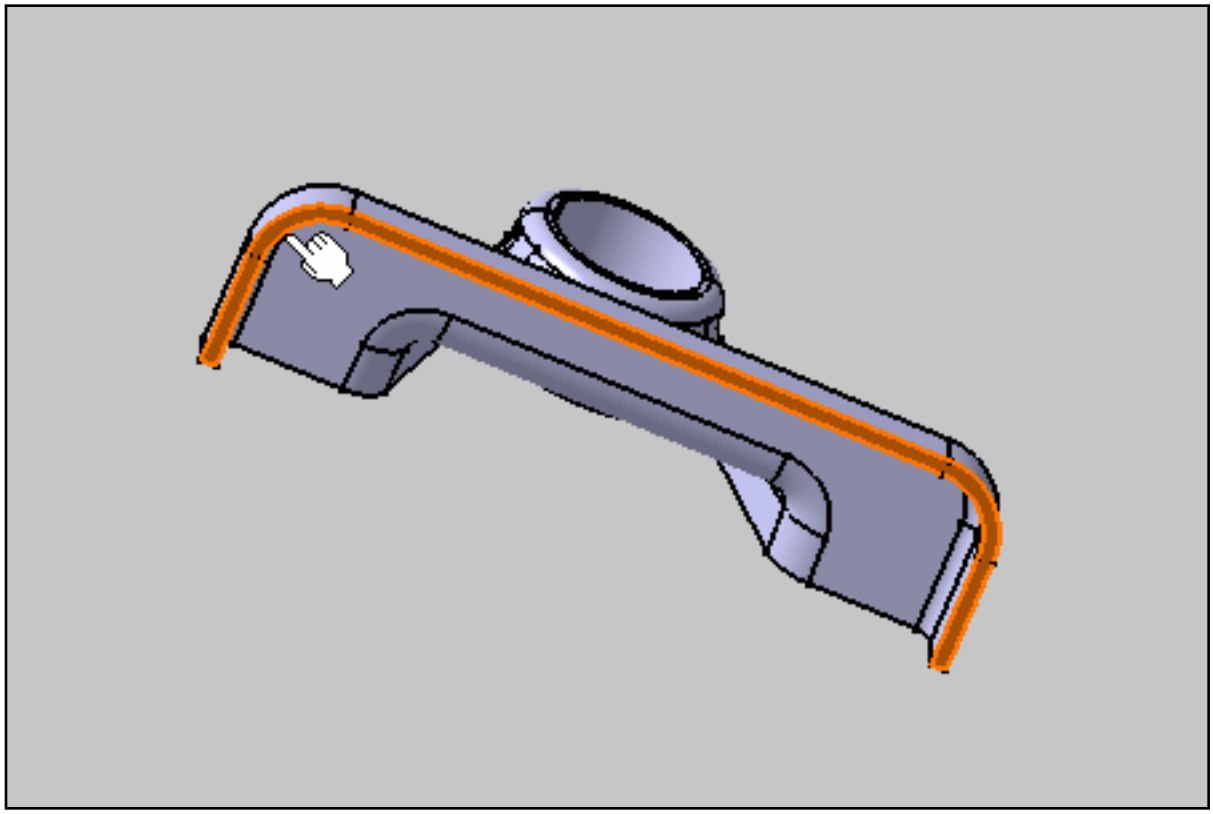
1. Select the **Insert -> Instantiate From Document...** command.

2. Select select PowerCopy2.CATPart from file selection dialog box.

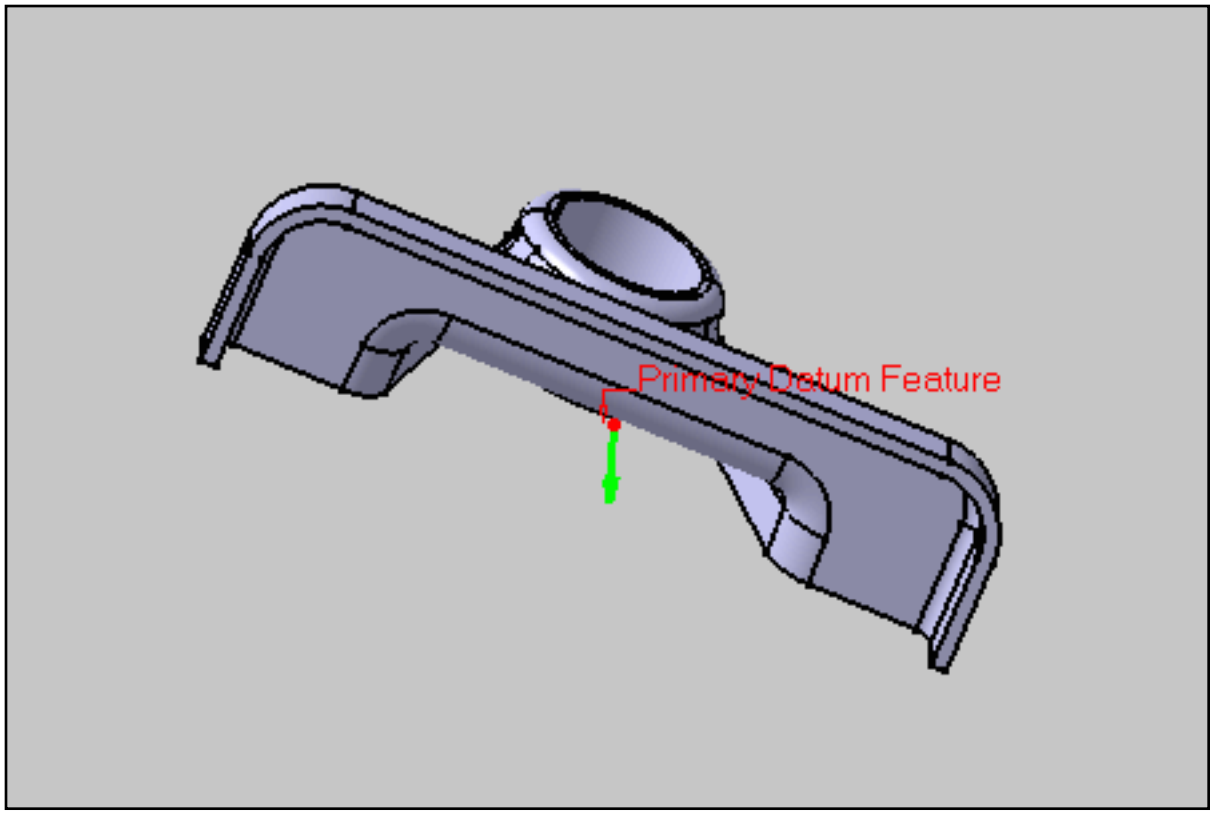
   The **Insert Object** dialog box appears. **Primary Datum Feature** is pre-selected.



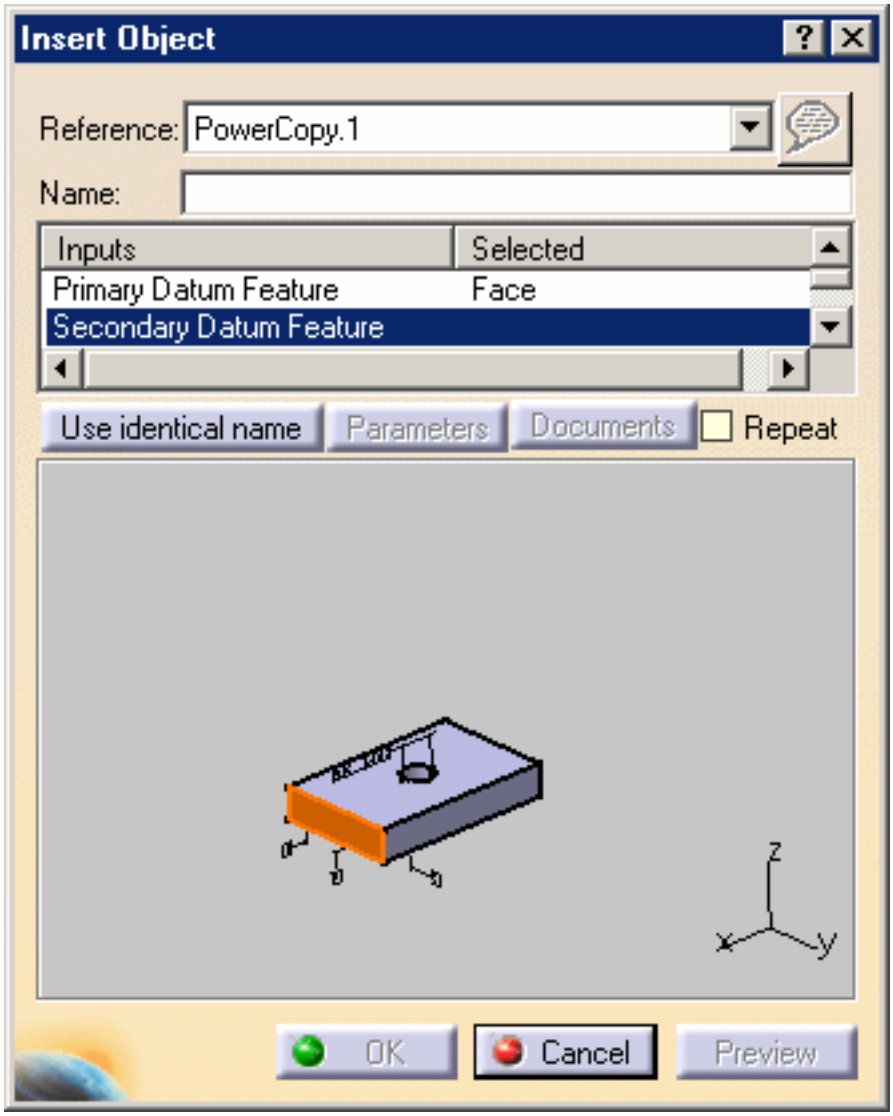
3. Select the surface as shown.



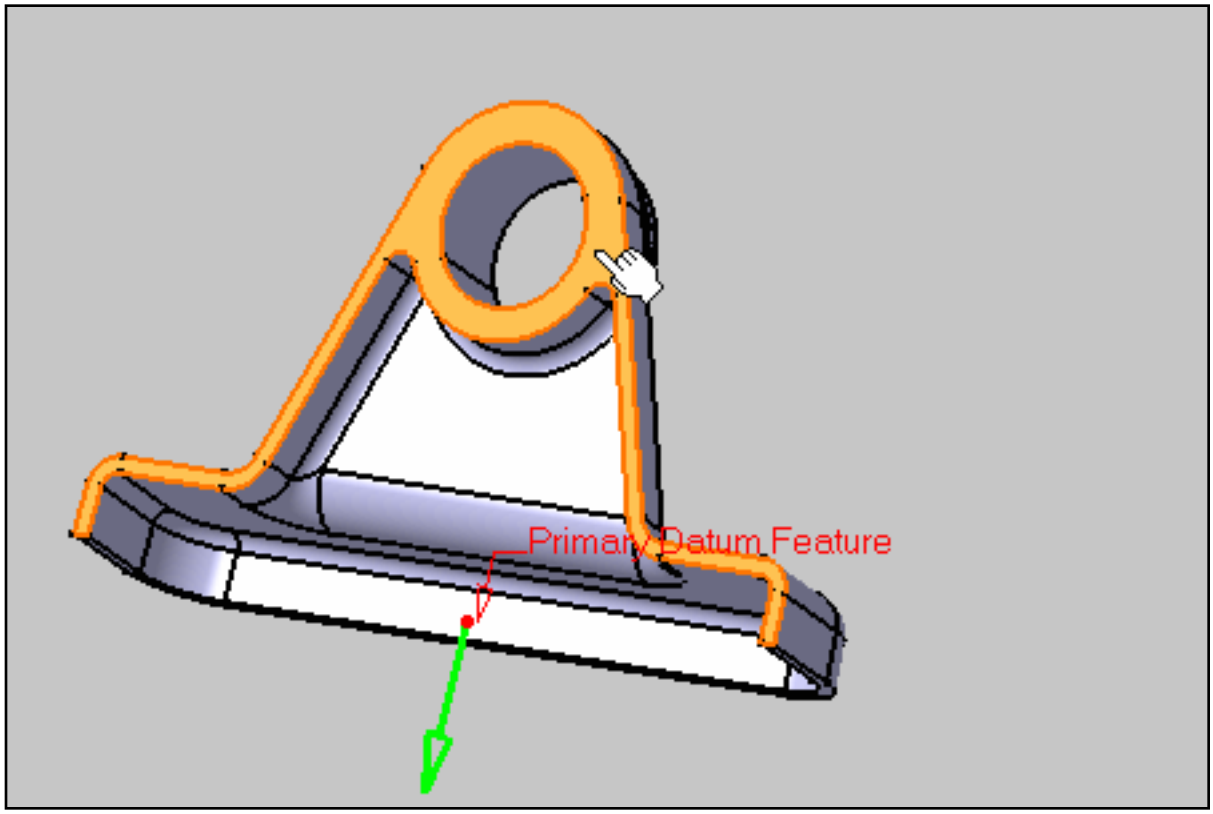
The **Primary Datum Feature** is associated with this surface.



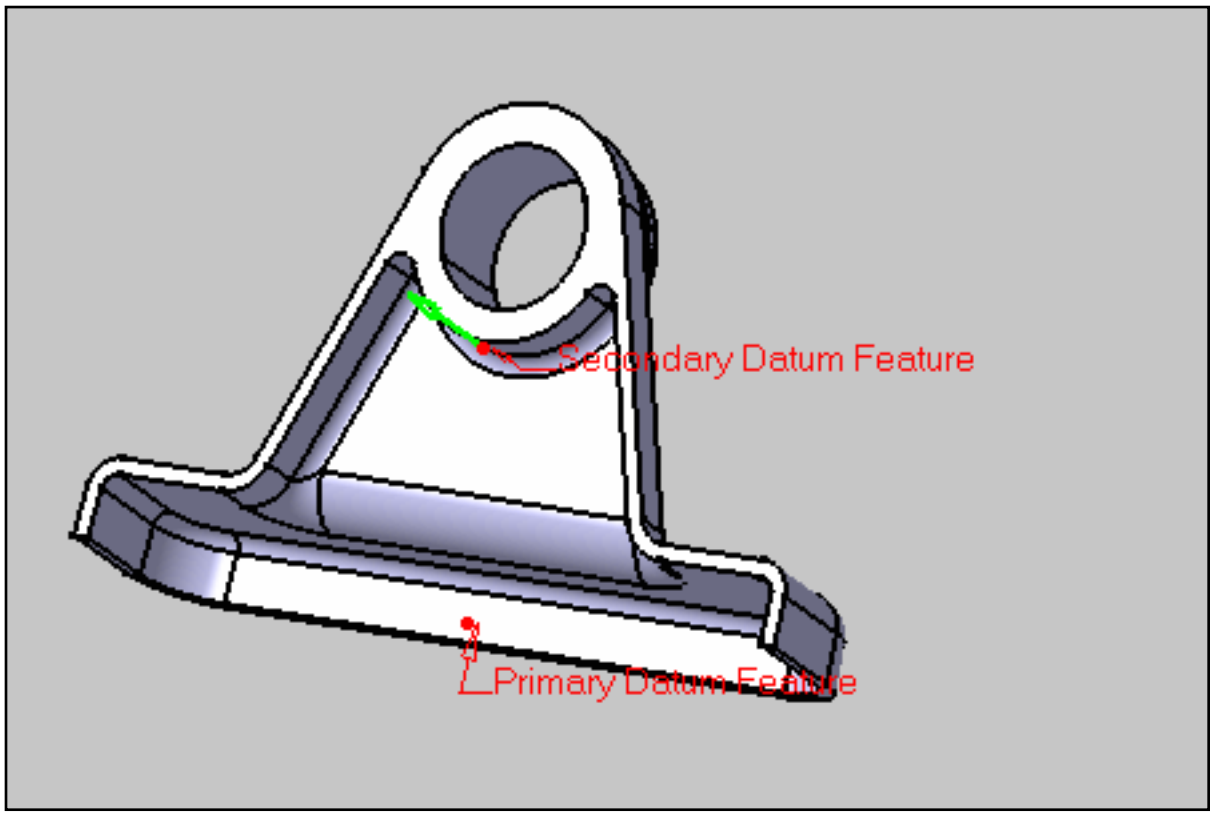
The next input **Secondary Datum Feature** is pre-selected in the **Insert Object** dialog box.



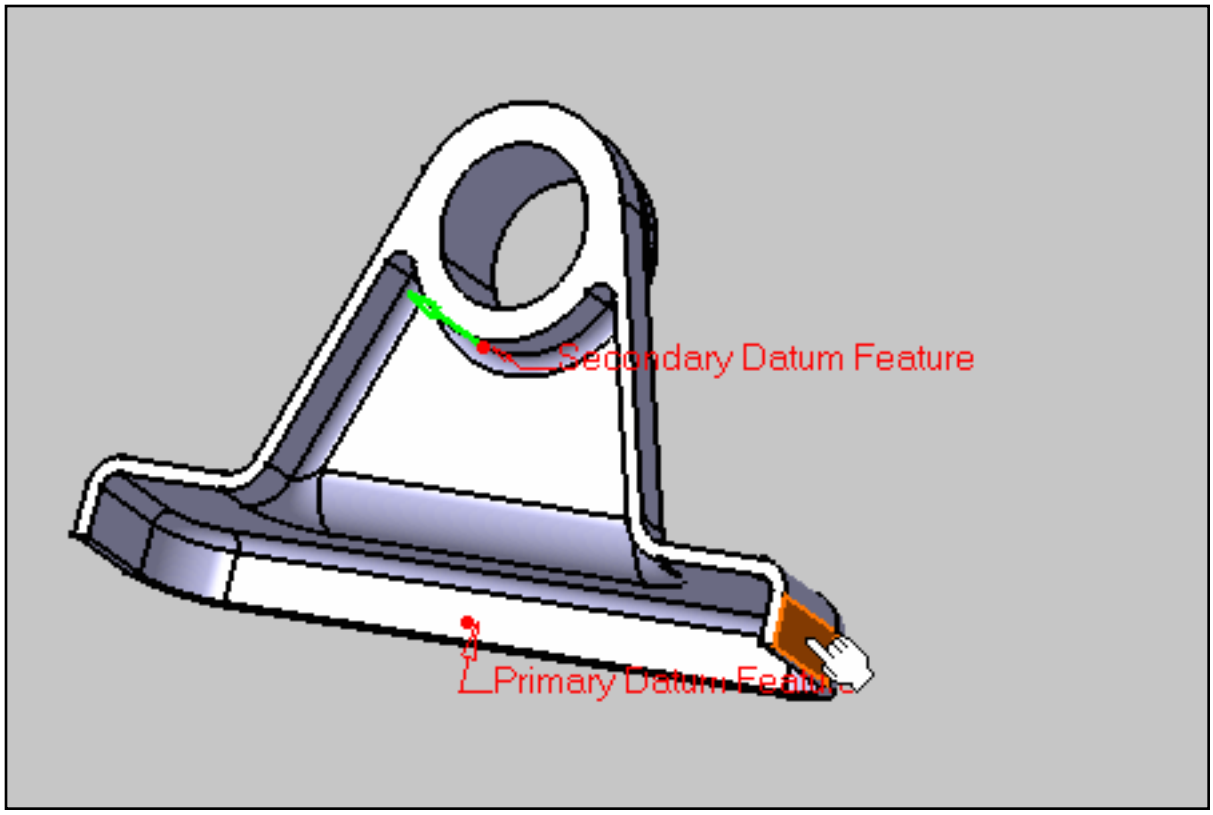
4. Select the surface as shown.



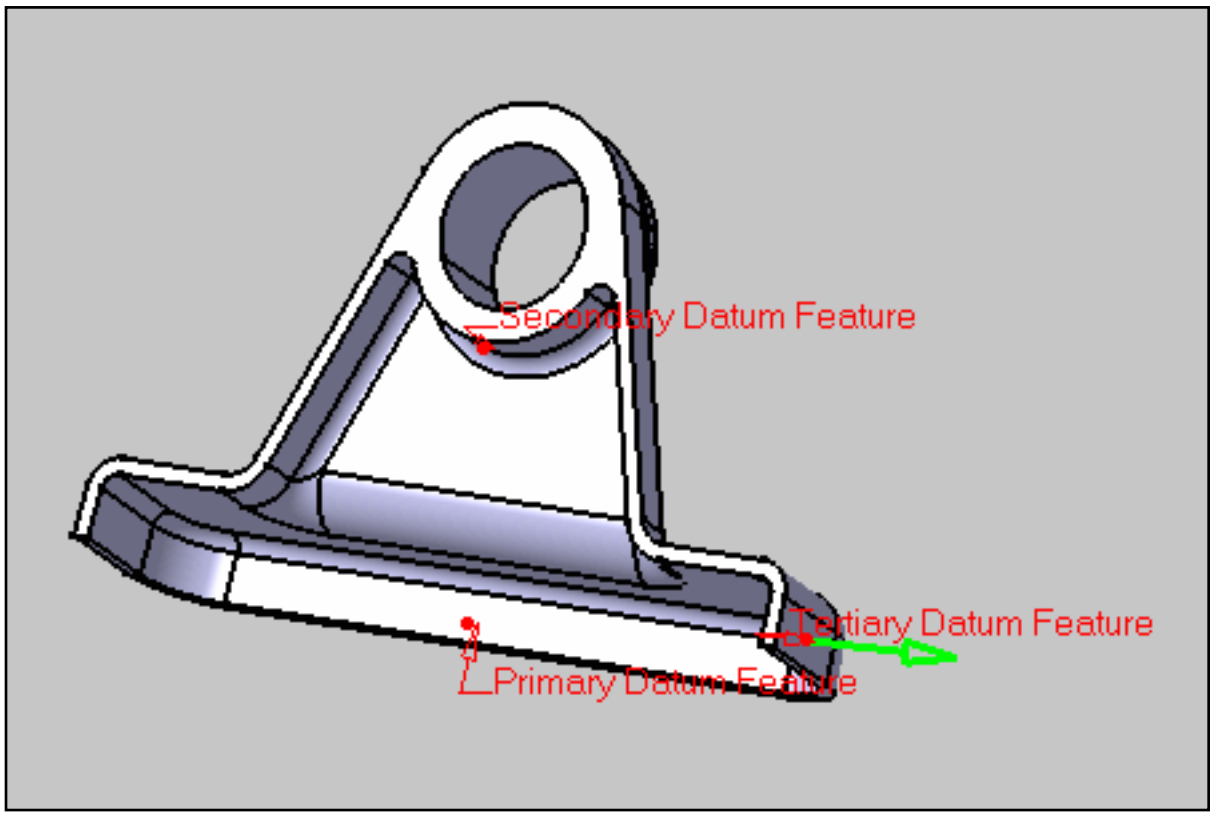
The **Secondary Datum Feature** is associated with this surface.
The next input **Tertiary Datum Feature** is pre-selected in the **Insert Object** dialog box.



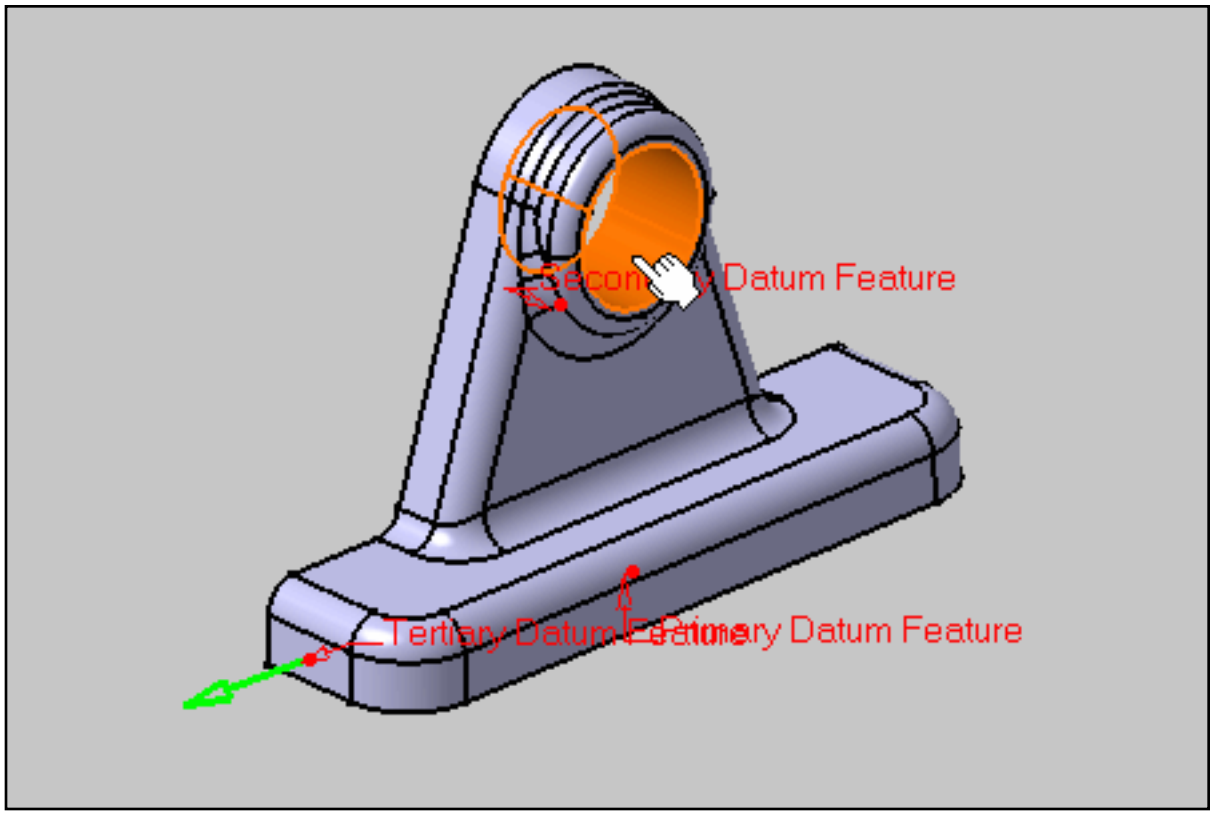
5. Select the surface as shown.



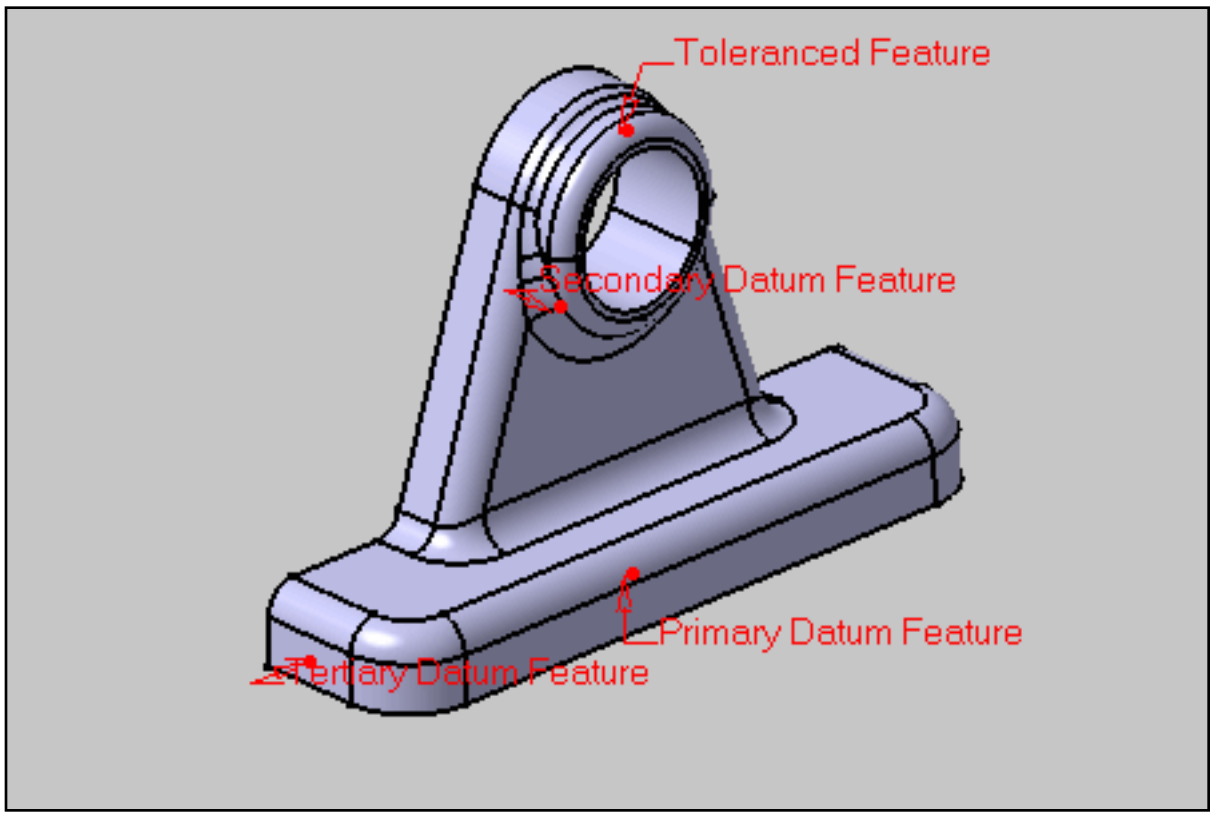
The **Tertiary Datum Feature** is associated with this surface.
The next input **Toleranced Feature** is pre-selected in the **Insert Object** dialog box.



6. Select the surface as shown.

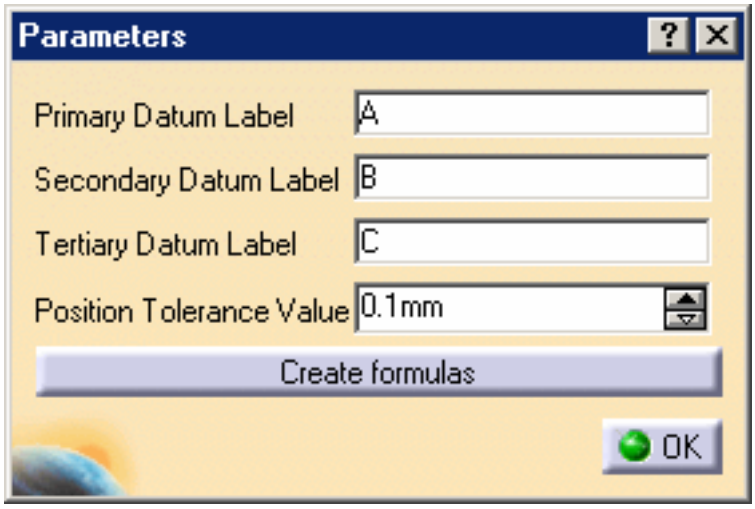


The **Toleranced Feature** is associated with this surface.

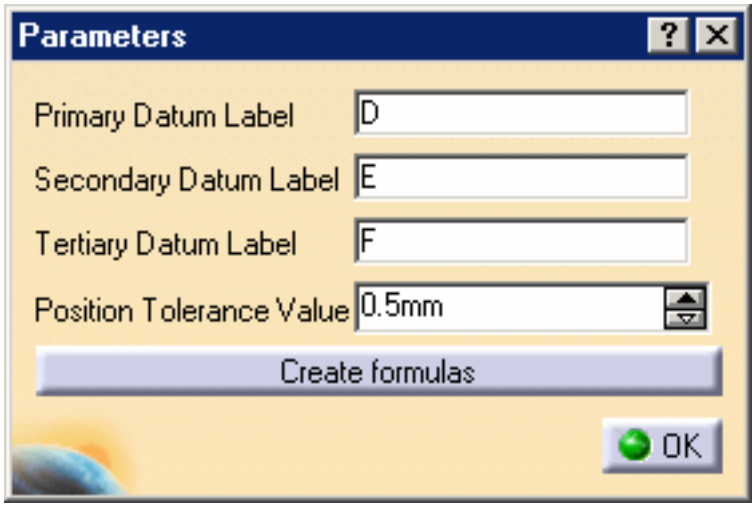


7. Click the **Parameters** in the **Insert Object** dialog box.

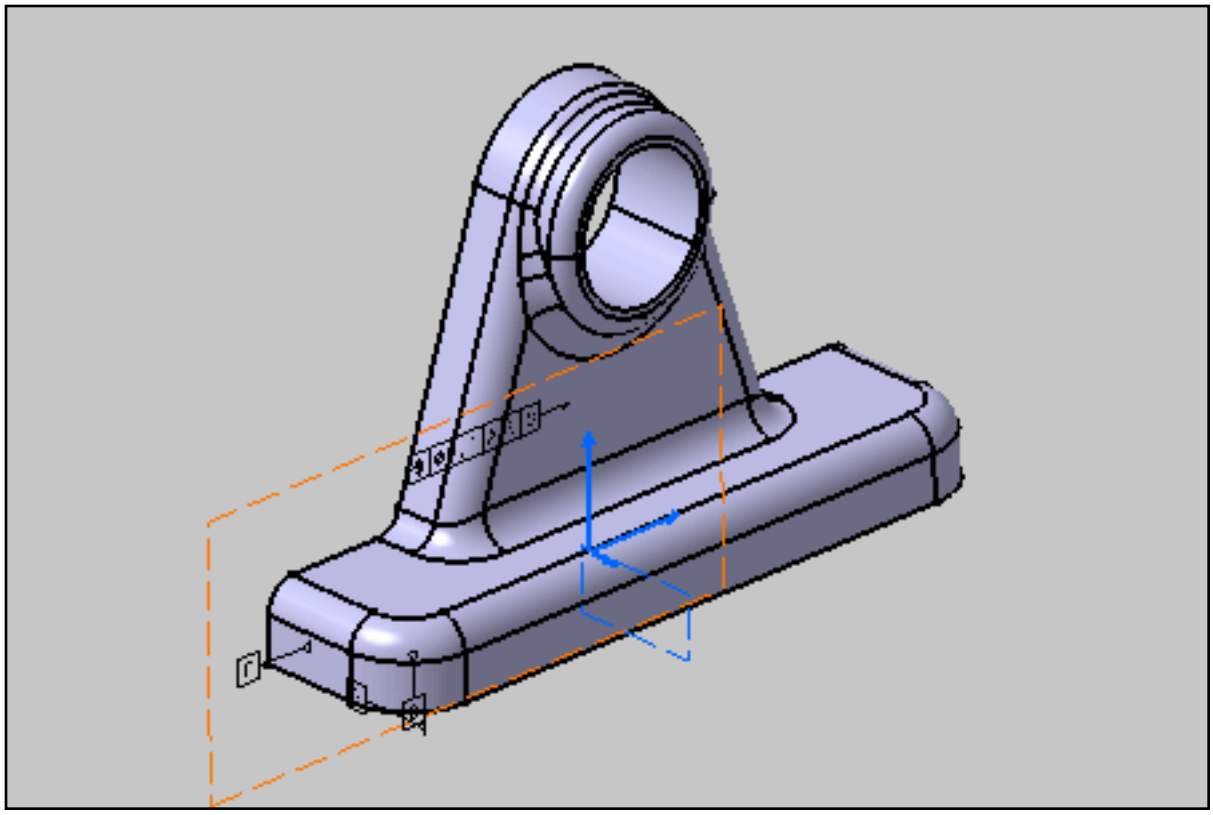
   The **Parameters** dialog box appears.



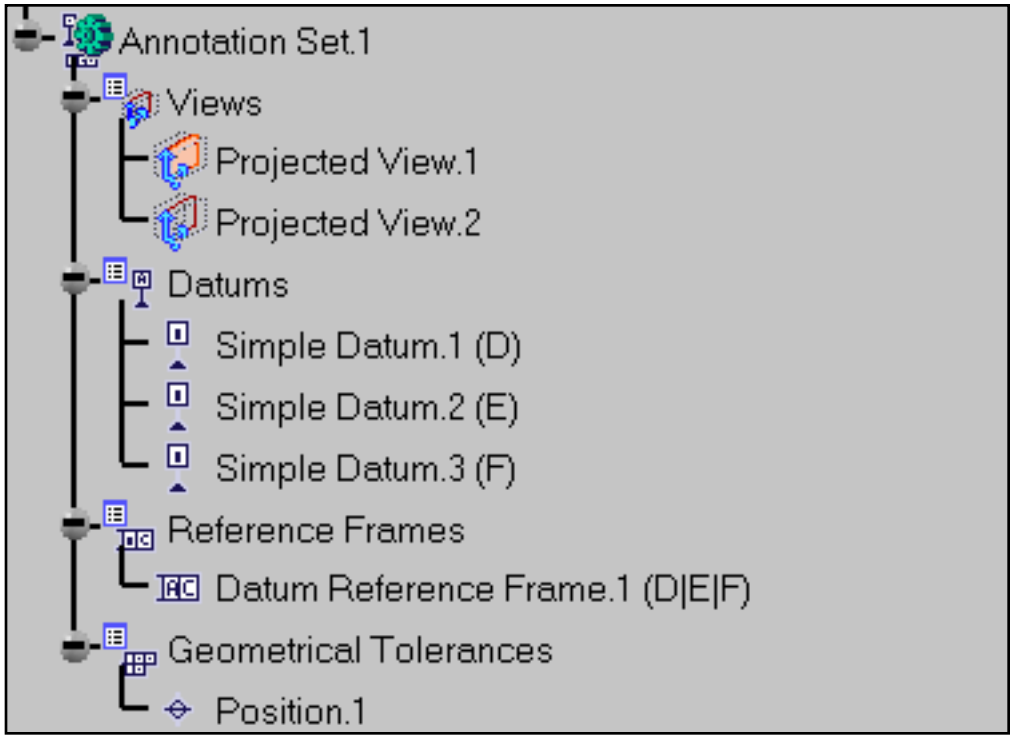
8. Change parameters as shown.



9. Click **OK** in the **Parameters** and **Insert Object** dialog boxes.



Annotations are created.



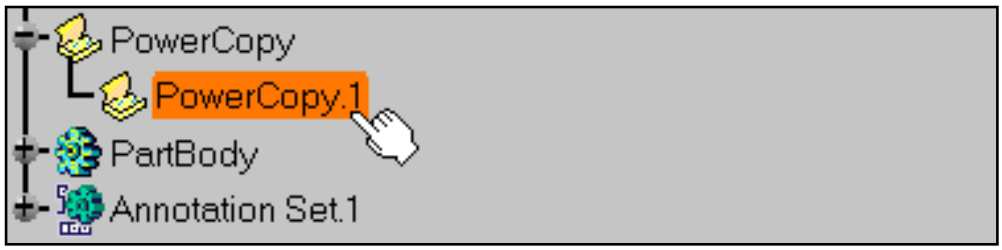
Back Up Next

Saving Power Copy into a Catalog

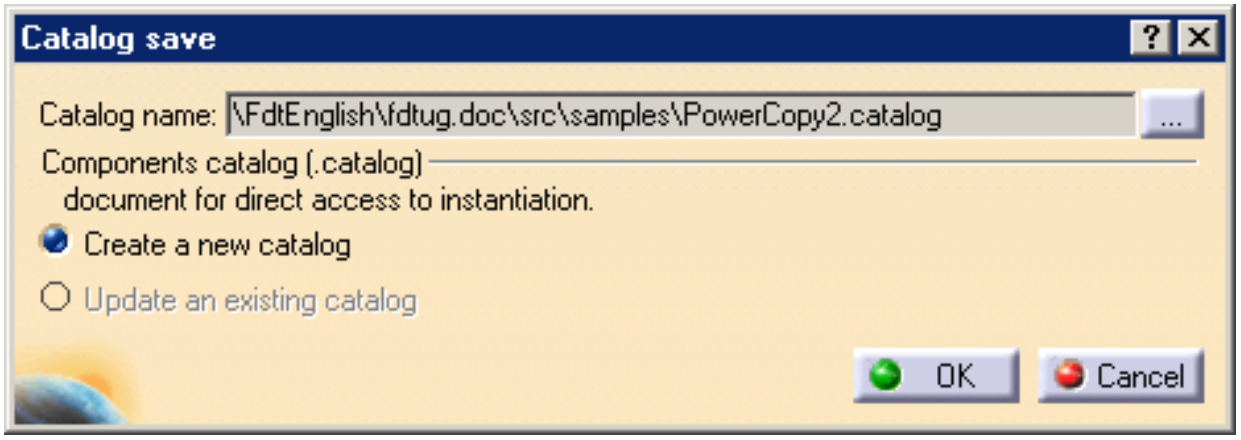
This task shows how to store PowerCopy elements into a catalog.

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.



Back Up

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.

Back Up

Creating an Automatic Constructed Geometry

This task shows you how to create an automatic constructed geometry.

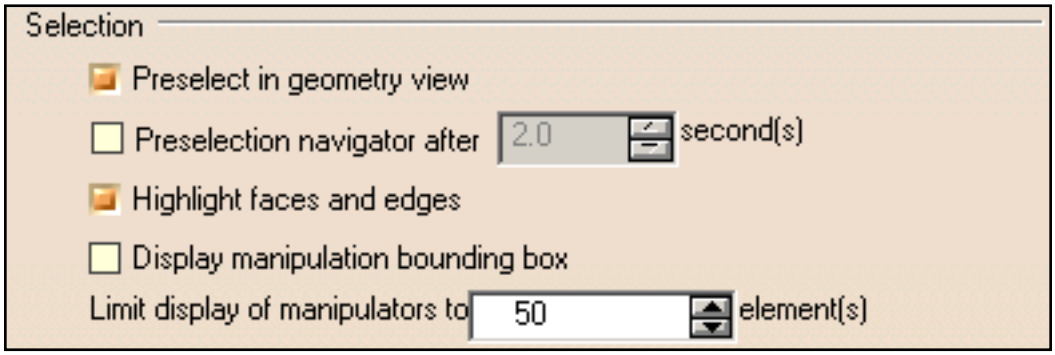
Creating an automatic constructed geometry allows you to create associative wireframe geometry according to the context or represented geometry. The lifecycle of an automatic constructed geometry is managed by the application.

The following constructed geometry elements are available:

- Line
- Point
- Plane
- Cylinder
- Circle

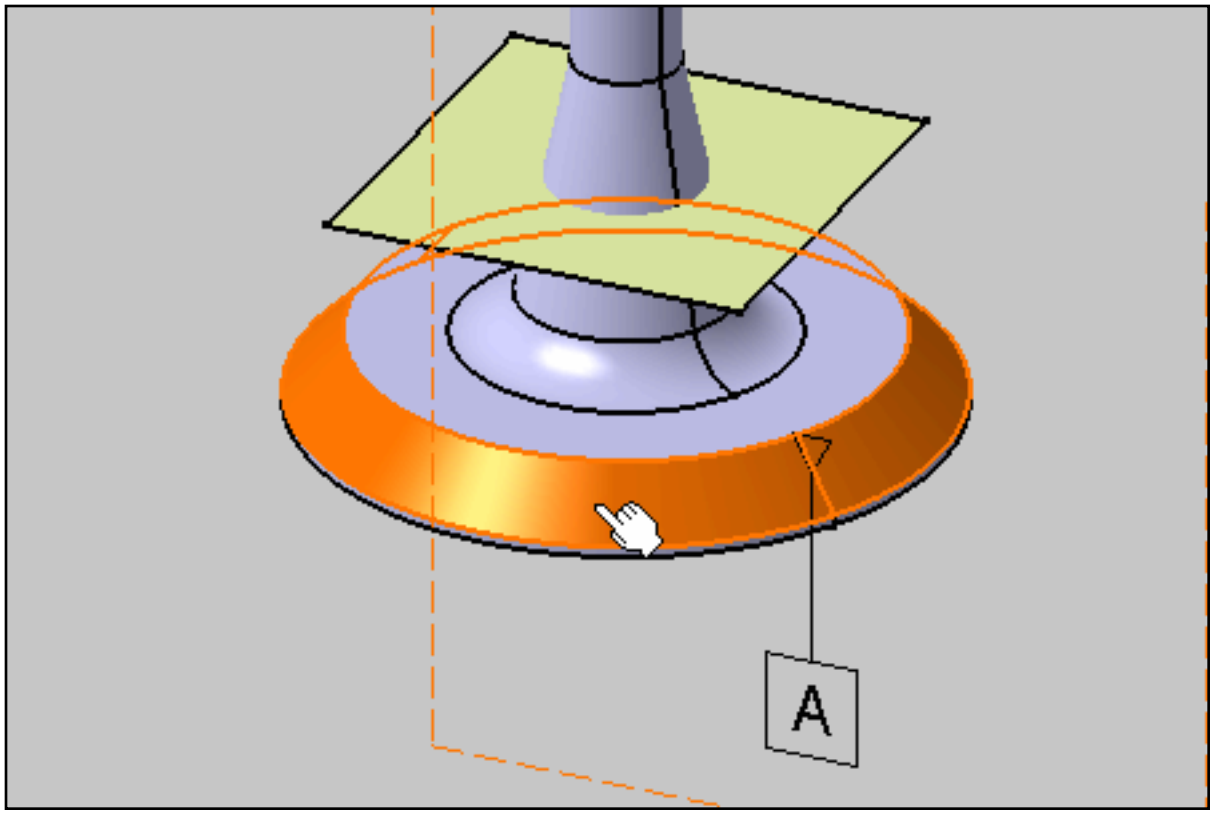
Open the Tolerancing_Annotations_08.CATPart CATPart document.

To improve the highlight of the geometry selection, select the **Tools->Options...** command.
In the **General** category, select the **Display** sub-category then the **Navigation** tab and check the option **Highlight faces and edges**.



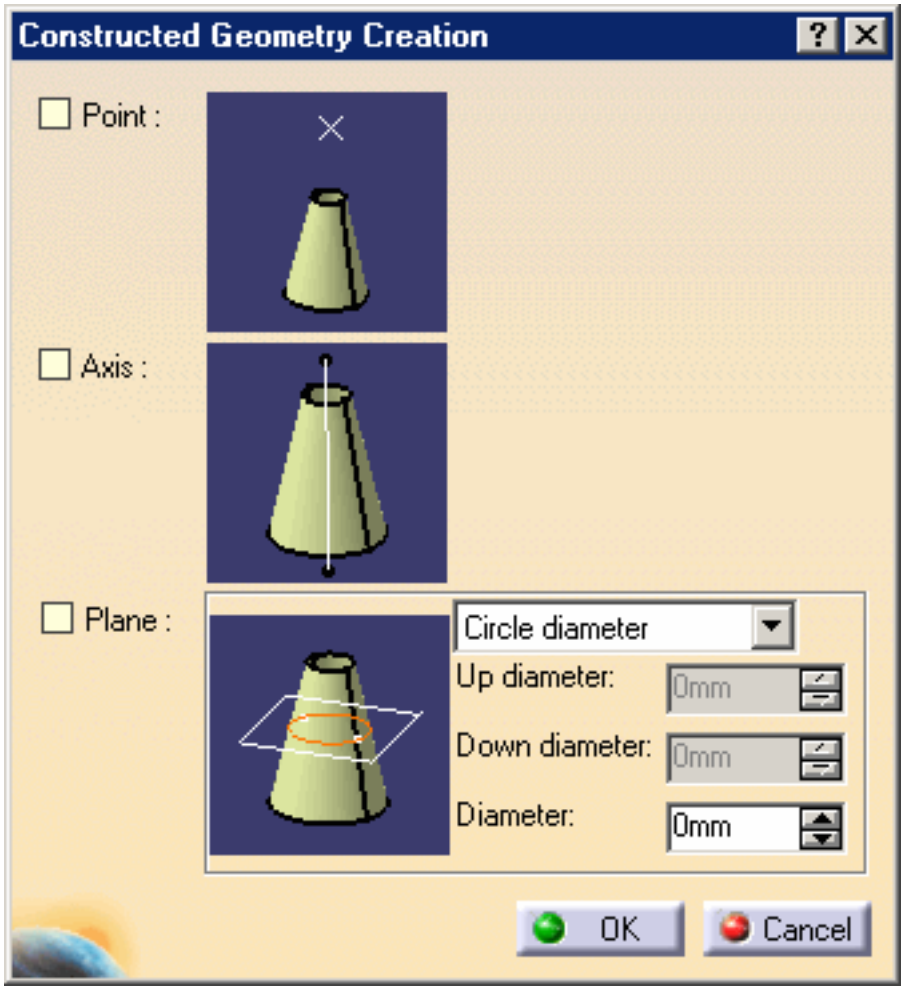
1. Click the **Constructed Geometry Creation** icon: 

2. Select the surface as shown.



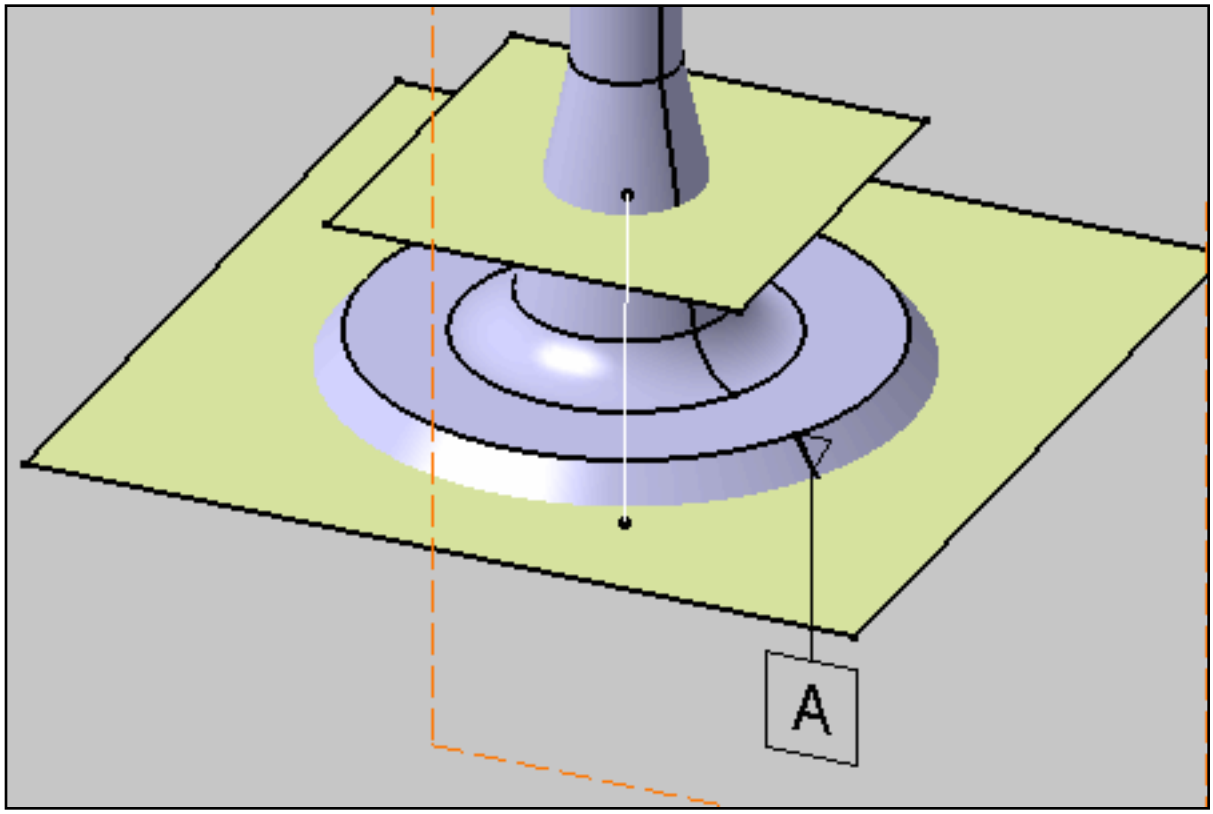
The **Constructed Geometry Creation** dialog box appears.

Double-clicking any automatically constructed geometry displays this dialog box.

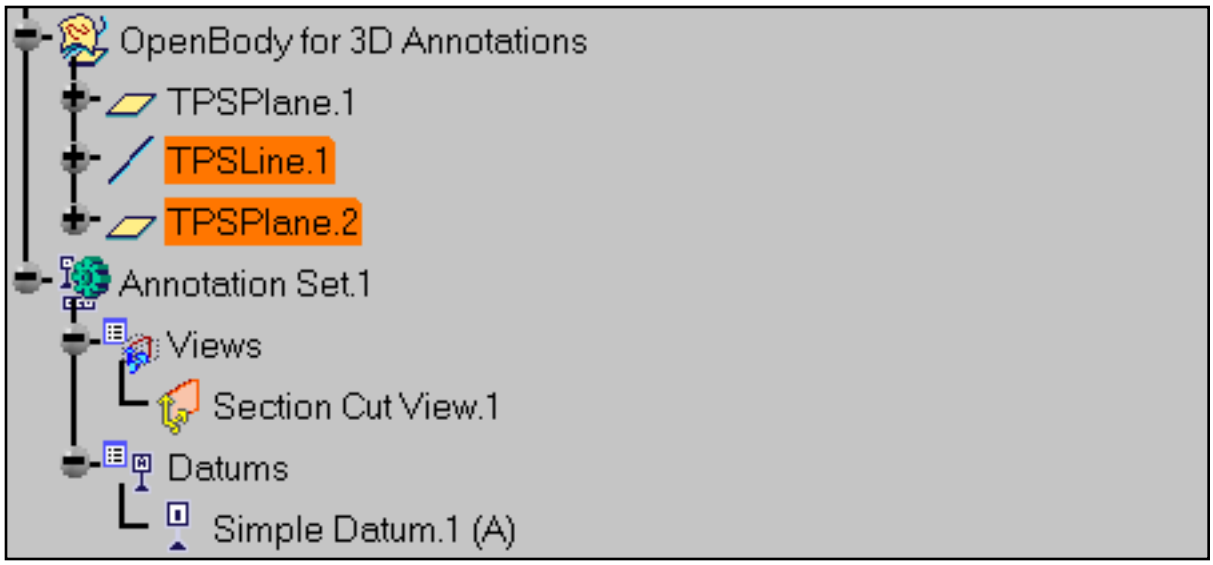


3. Check **Axis** and **Plane** as geometries to be generated and click **OK** in the dialog box.

The axis and the plane are created in the geometry. See also Constructed Geometry options for graphic properties and limits.

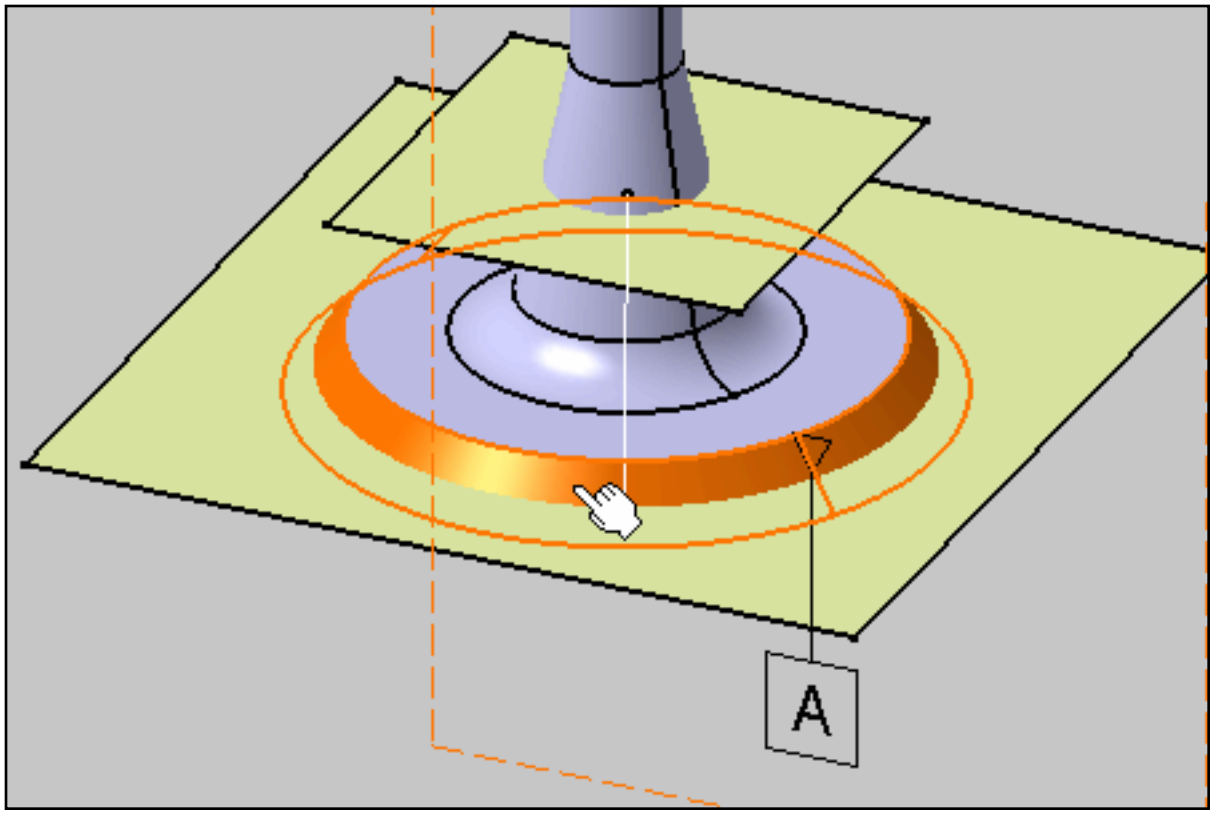


In the specification tree, an open body dedicated to the constructed geometry is added.

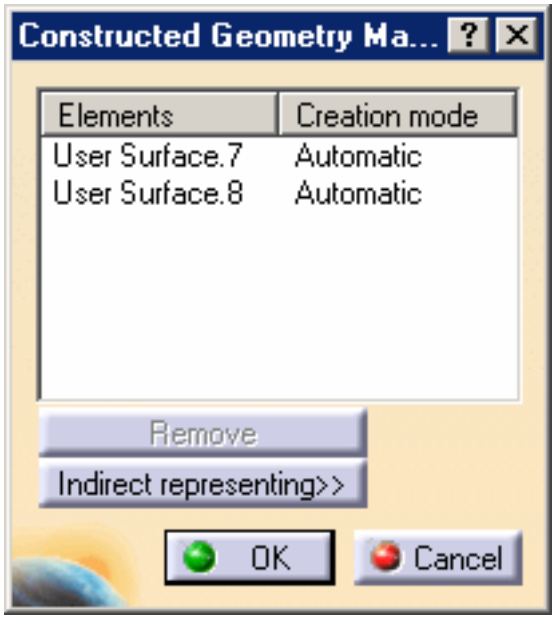


4. Click the **Constructed Geometry Management** icon: 

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**.

Up Next

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 an automatically constructed geometry is not managed by the application.

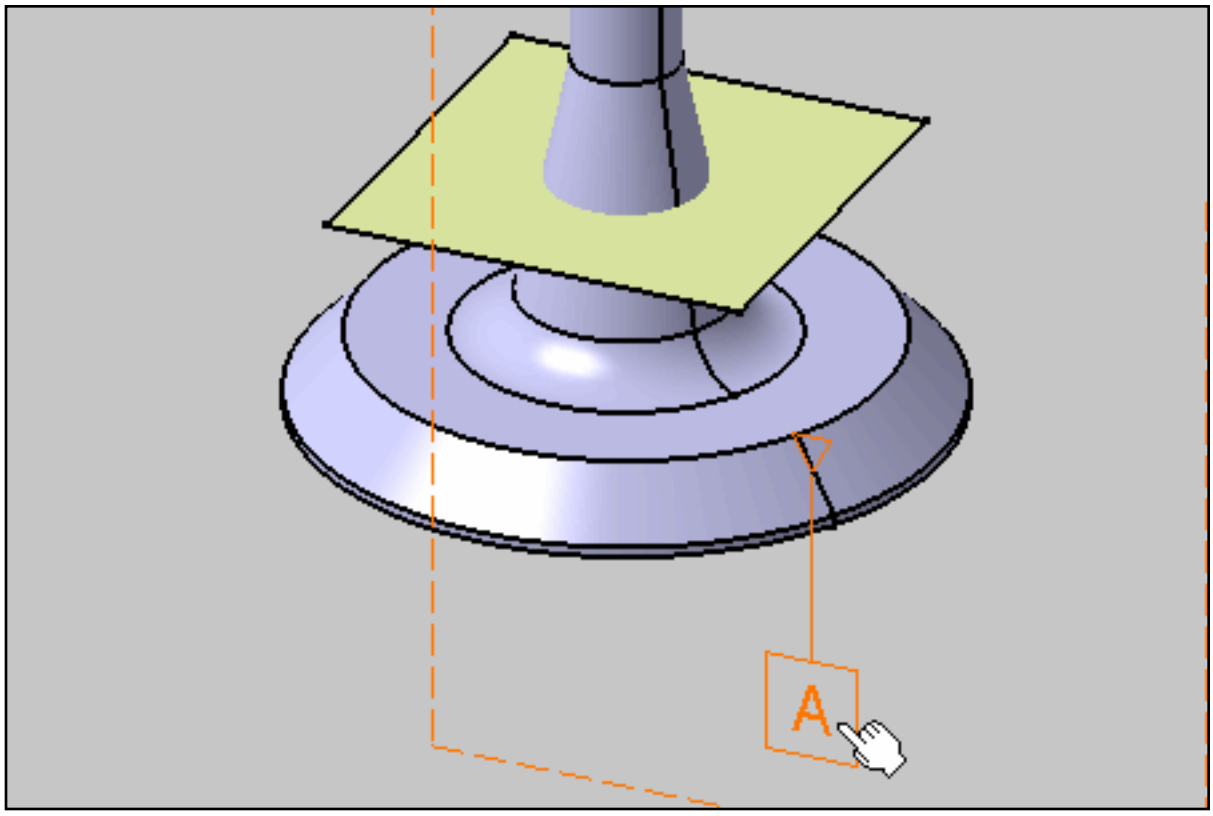
For this, you can select:

- A 3D annotation: the corresponding tolerancing feature is retrieved.
- A geometrical element: the corresponding tolerancing feature is retrieved or created.

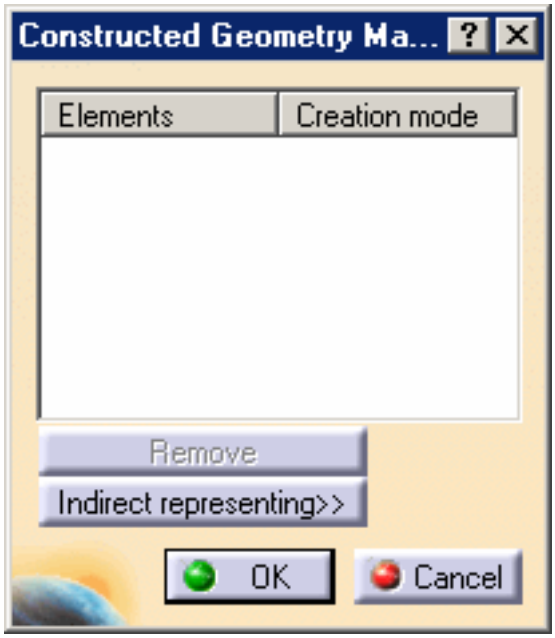
Open the Tolerancing_Annotations_08.CATPart CATPart document.

1. Click the **Constructed Geometry Management** icon: 

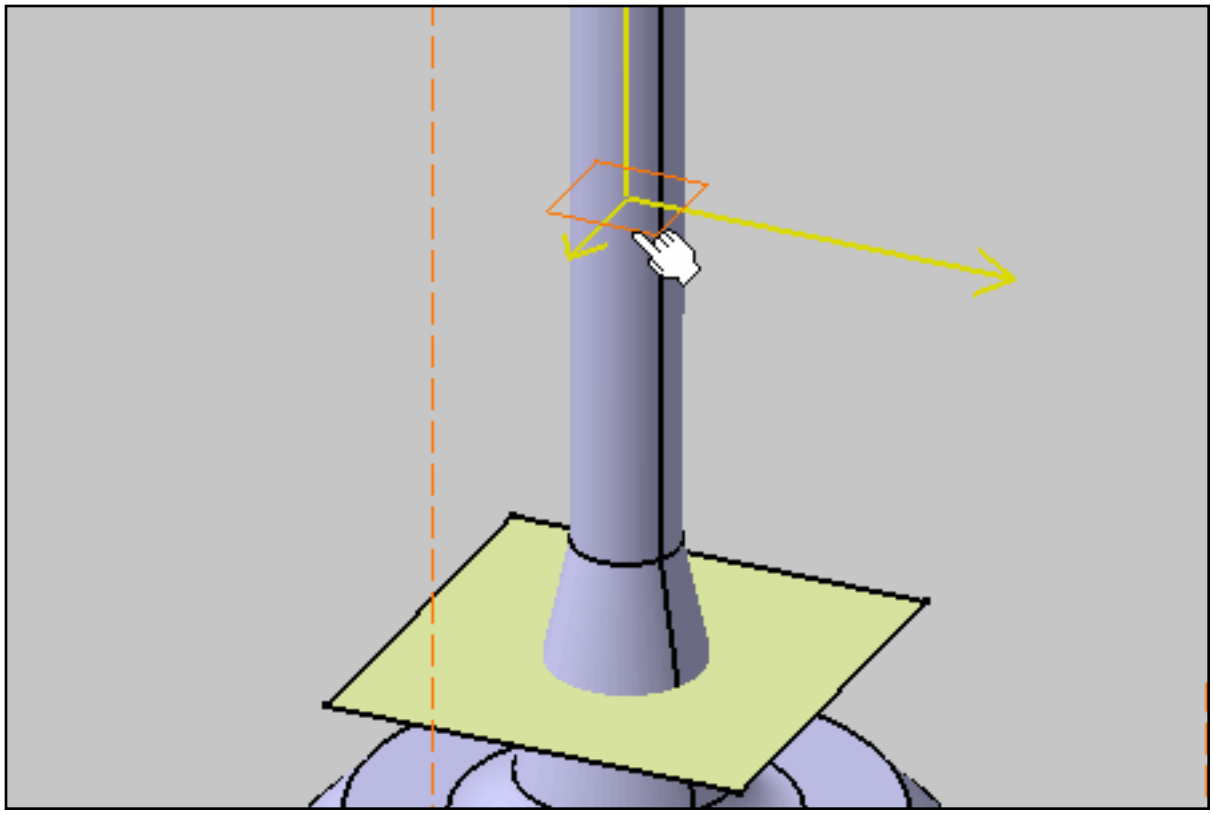
2. Select the datum.



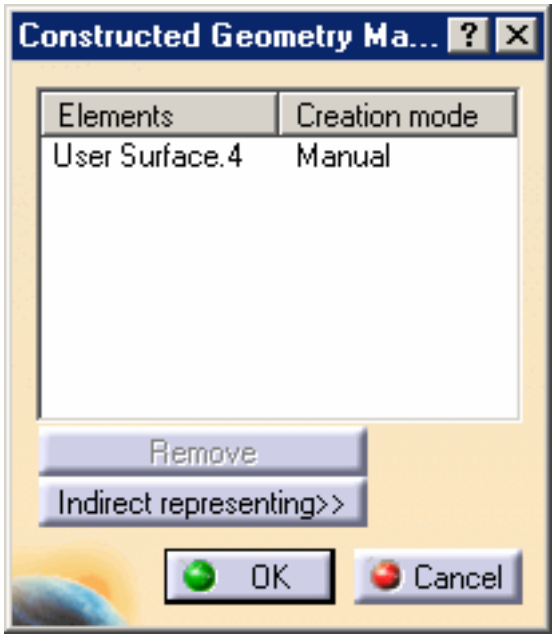
The **Constructed Geometry Management** dialog box appears.



3. Select the reference plane as shown to define the gage plane of the related surface of the annotation.



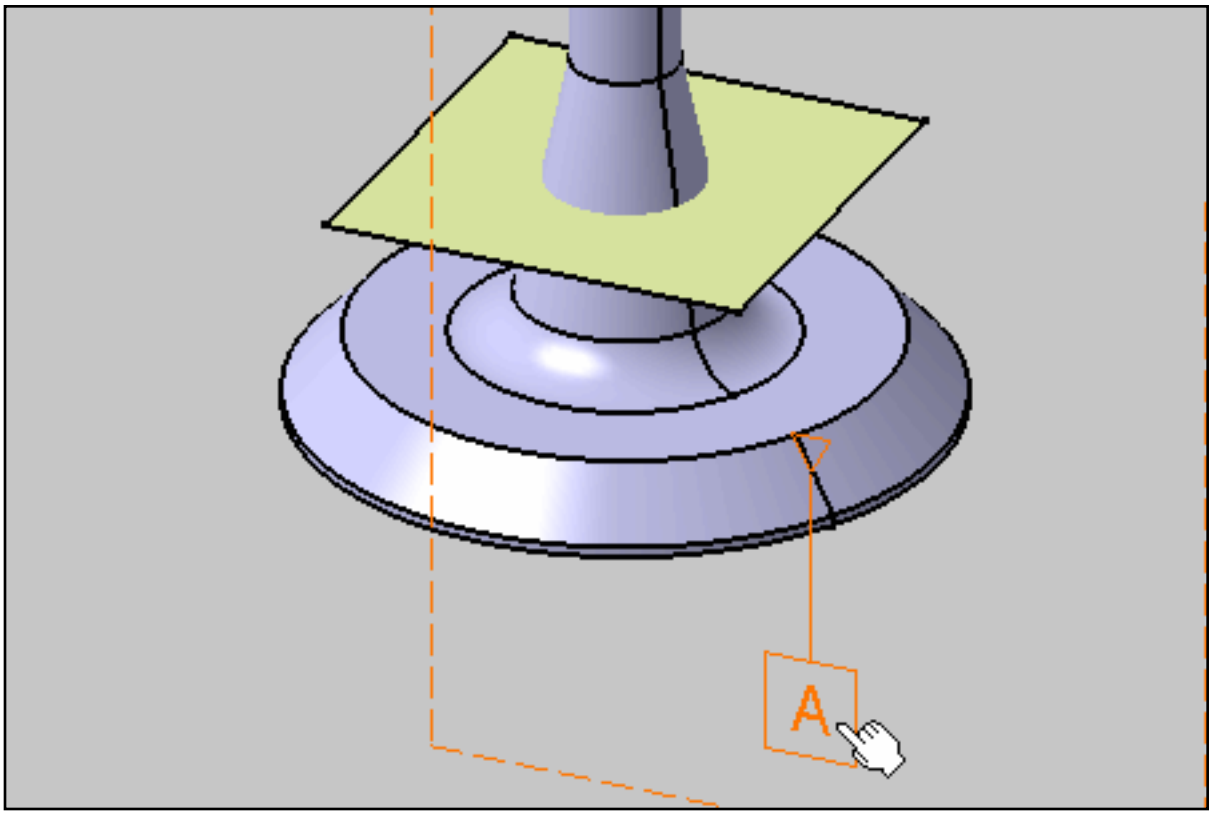
The **Constructed Geometry Management** dialog box is updated, the constructed geometry is created in manual mode: the associativity is managed by the user.



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

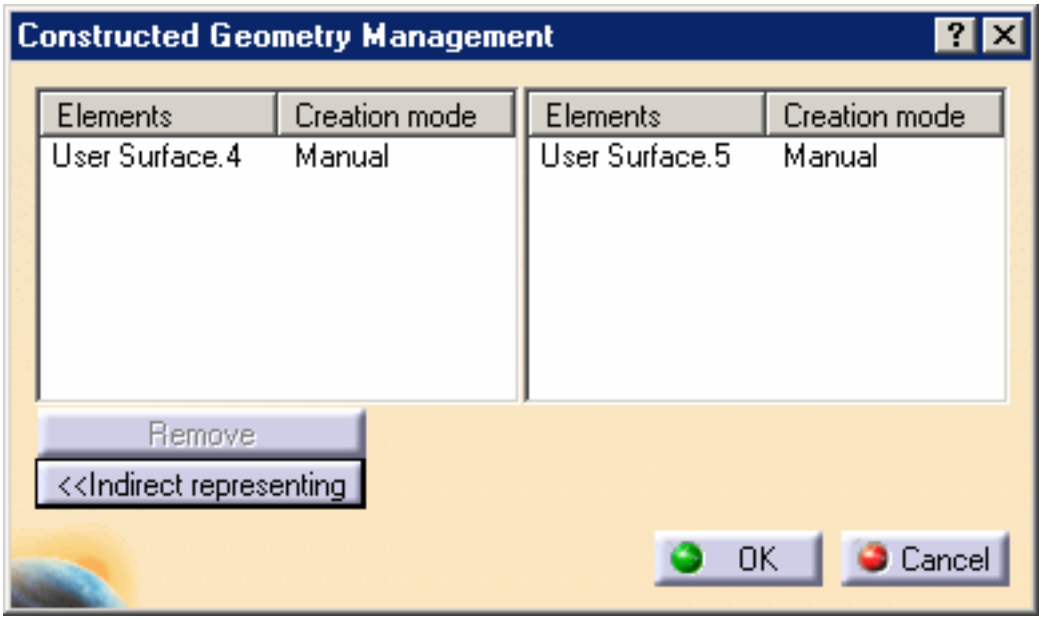
5. Click the **Constructed Geometry Management** icon: 

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).



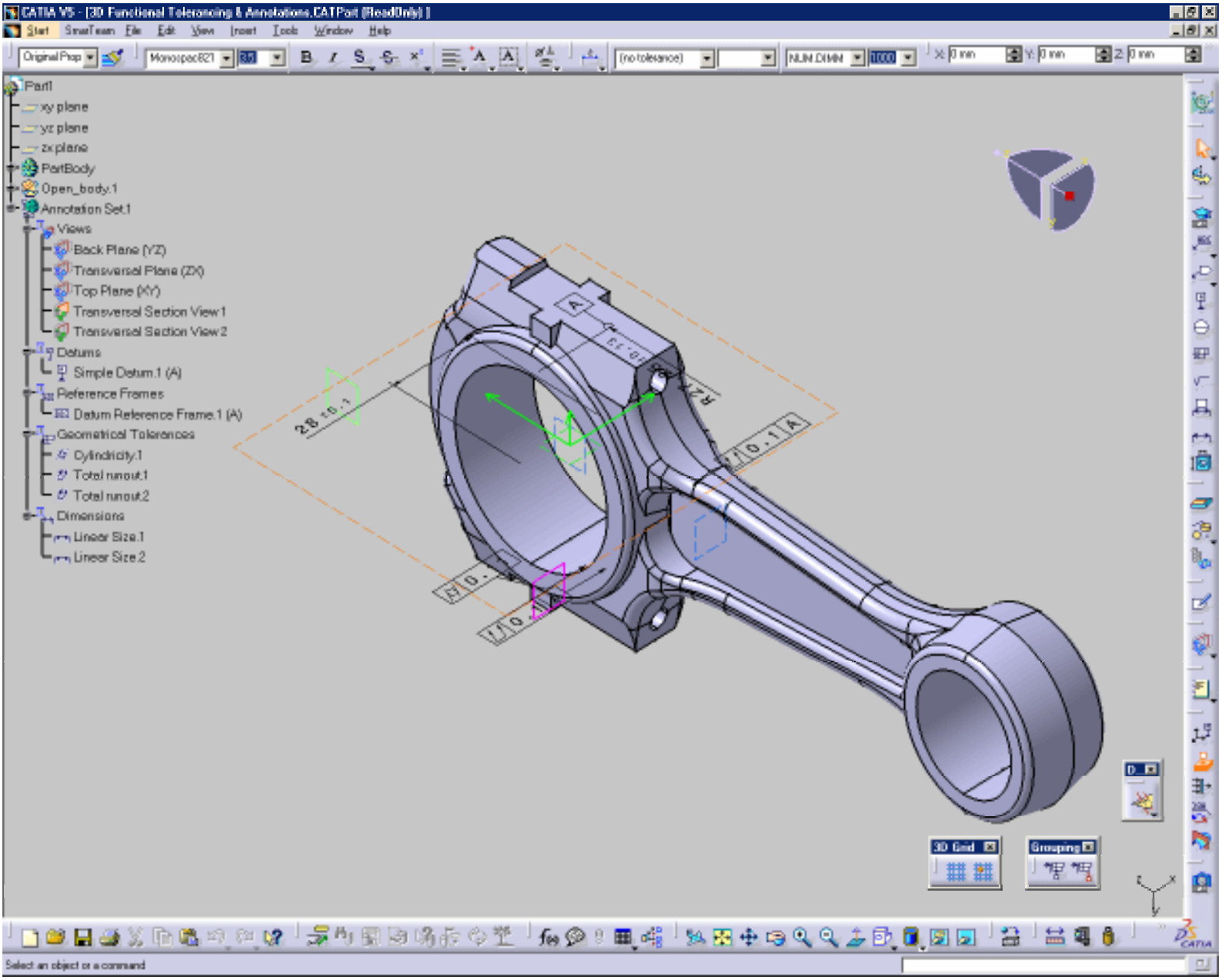
8. Click **OK** in the dialog box.



Back Up

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

This section presents the main menu bar available when you run the application and before creating or opening a document:



Insert

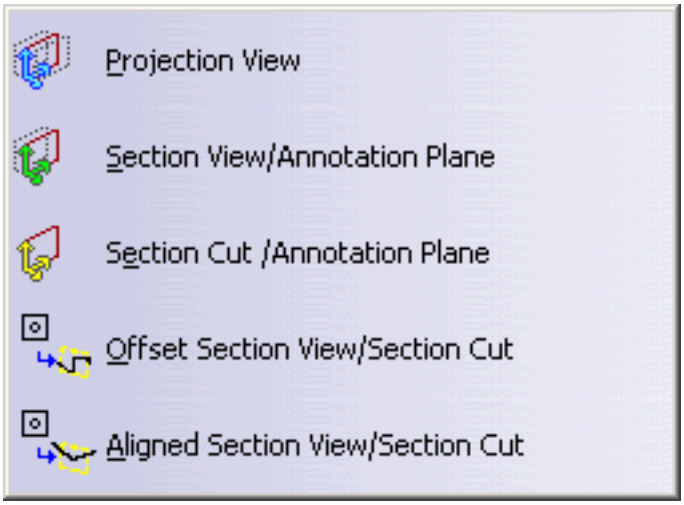


For...	See...
Views/ Annotation Planes	Insert -> Views/Annotation Planes Menu
Annotations	Insert -> Annotations Menu
Advanced Replication Tools	Insert -> Advanced Replication Tools Menu
Instantiate From Document...	Instantiating Power Copy

Up Next

Insert -> Views/Annotation Planes Menu

This section presents the Insert -> Views/Annotation Planes menu:

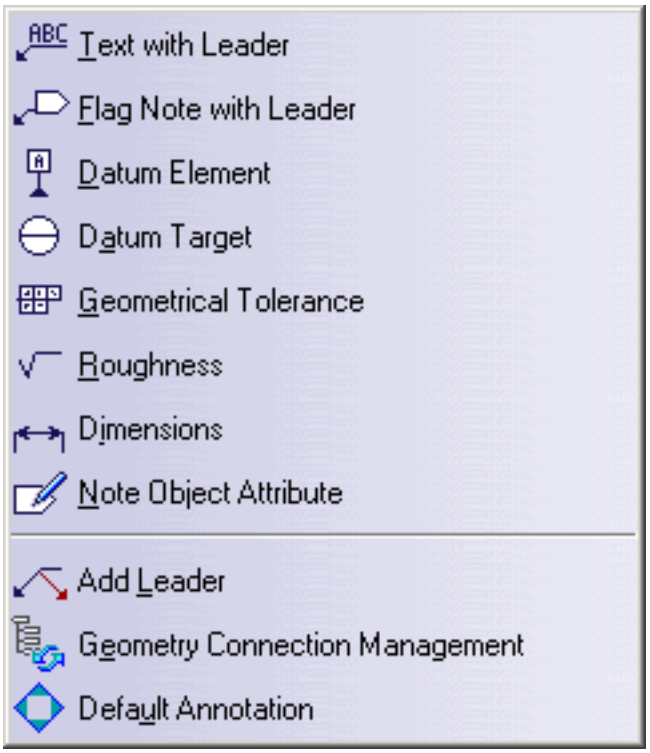


For...	See...
Projection View	Creating a Projection View/ Annotation Plane
Section View	Creating a Section View/ Annotation Plane
Section Cut	Creating a Section Cut View/ Annotation Plane
Offset Section View/Section Cut	Creating an Offset Section View/Section Cut
Aligned Section View/Section Cut	Creating an Aligned Section View/Section Cut

Up Next

Insert -> Annotations Menu

This section presents the Insert -> Annotations menu:



For...	See...
Text with Leader	Creating a Textual Annotation
Flag Note with Leader	Creating Flag Notes
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

Back Up Next

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

Back Up

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

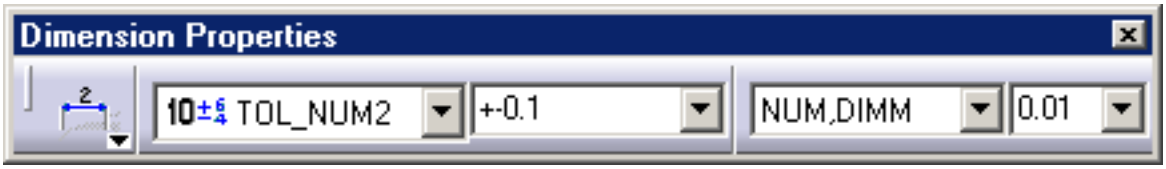
 See Creating Stacked Dimensions

 See Creating Coordinate Dimensions

 See Creating Curvilinear Dimensions

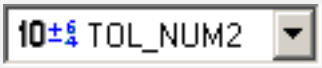
Back Up Next

Dimension Properties Toolbar





See Creating Dimensions



See Dimension Tolerance Display



See Tolerance Display



See Dimension Numerical Display , see Dimension Units reference.



See Precision Display

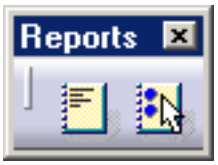
Back Up Next

Reporting Toolbar



Jump to Reports Sub-Toolbar

Reports Sub-Toolbar

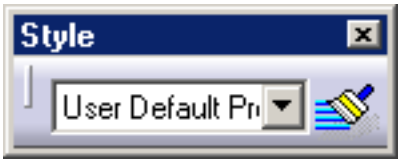


See Generating a Check Report

See Customizing the Reporting

Back Up Next

Style Toolbar





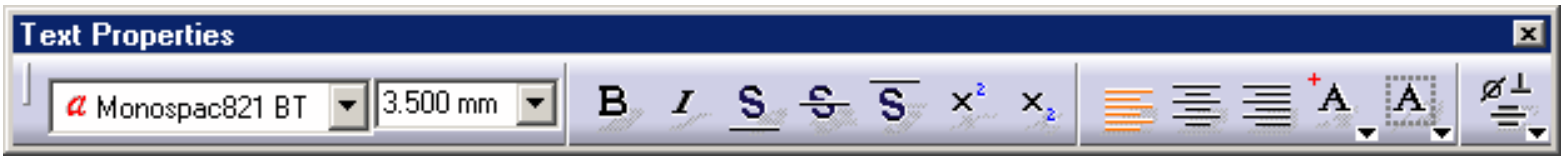
See Defining Default Properties



See Copying Graphic Properties

Back Up Next

Text Properties Toolbar



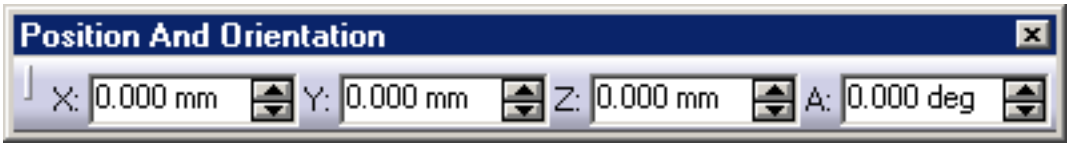
Monospac821 BT	See Setting Graphic Properties
3.500 mm	See Setting Graphic Properties
B	See Setting Graphic Properties
I	See Setting Graphic Properties
S	See Setting Graphic Properties
S	See Setting Graphic Properties
S	See Setting Graphic Properties
x²	See Setting Graphic Properties
x₂	See Setting Graphic Properties
	See Setting Graphic Properties
	See Setting Graphic Properties
	See Setting Graphic Properties
	See Setting Graphic Properties
	See Setting Graphic Properties

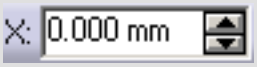
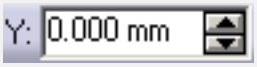
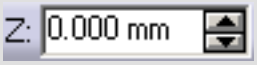
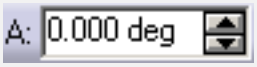


See Setting Graphic Properties

Back Up Next

Position and Orientation Toolbar



	See Moving Annotations
	See Moving Annotations
	See Moving Annotations
	See Moving Annotations

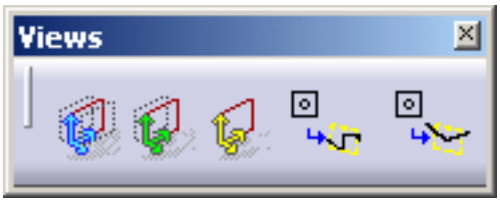
Back Up Next

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

Back Up Next

Visualization Toolbar




See Disabling 3D Annotations


See Managing Associativity Between Elements


See Using Annotation Filter


See Mirroring Annotations


See Clipping Annotations Plane

Back Up Next

Note Object Attribute Toolbar

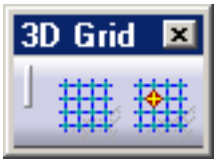




See Creating Note Object Attribute

Back Up Next

3D Grid Toolbar




See Using 3D Grid


See Using 3D Grid

Back Up Next

Grouping Toolbar





See Grouping Annotations Automatically



See Grouping and Ordering Annotations

Back Up Next

Capture Toolbar





See Creating an Annotation Capture:
This command open the Tolerancing & Annotation Captures workshop.

Back Up Next

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

Back Up Next

Deviations Toolbar (Compact)



Jump to Deviations Sub-Toolbar (Extended)

Deviations Sub-Toolbar (Extended)



See Creating a Deviation

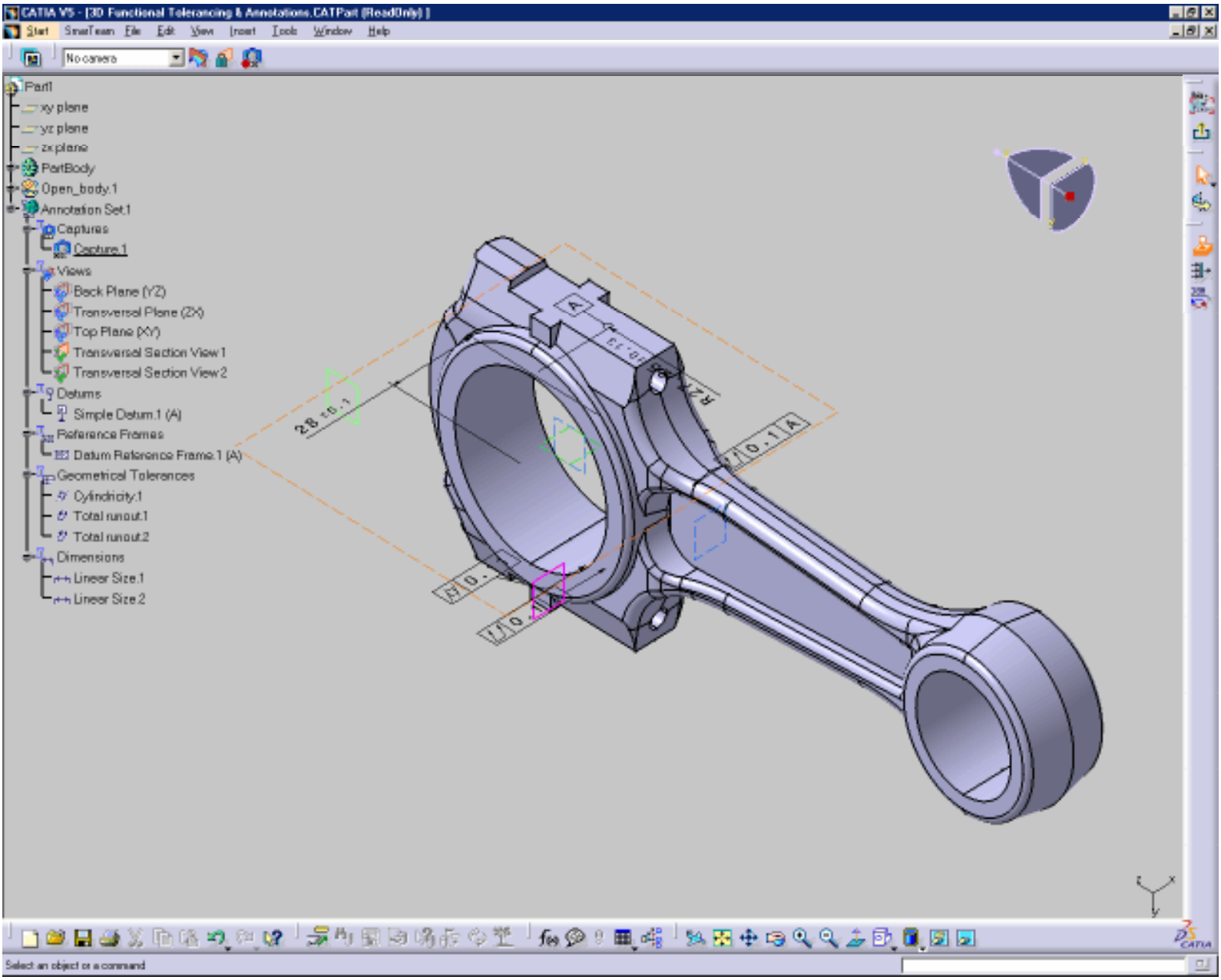
See Creating a Correlated Deviation

See Creating a Distance Between Two Points

Back Up Next

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
Back Up

Workbench Toolbar





See Creating an Annotation Capture

Up Next

Capture Visualization Toolbar




See Managing Associativity Between Elements


See Filtering Annotations


See Mirroring Annotations

Back Up Next

Capture Options Toolbar



	See Managing Capture Options
	See Managing Capture Options
	See Managing Capture Options
	See Managing Capture Options

Back Up Next

Camera Toolbar





See Creating a Camera

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Customizing for 3D Functional Tolerancing & Annotation

This section describes the different types of setting customization you can perform in 3D Functional Tolerancing & Annotations using the **Tools -> Options** command.

All tasks described here deal with permanent setting customization.



Tolerancing
Display
Constructed Geometry
Manipulators
Dimension
Annotation
Tolerance Values
View/Annotation Plane

Tolerancing

This task will show you how to customize 3D Functional Tolerancing & Annotation tolerancing settings.

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

   The **Options** dialog box appears.

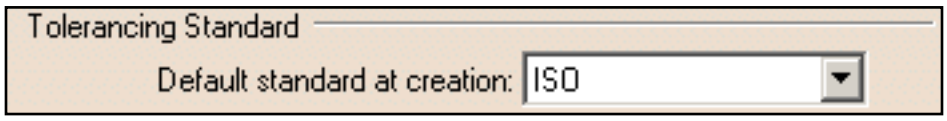
2. Click the **Mechanical Design** category.

3. Click the **3D Functional Tolerancing & Annotation** subcategory.

4. Click the **Tolerancing** tab.

   The **Tolerancing** tab appears, displaying five categories of options:

Tolerancing Standard



Default standard at creation

Defines the default standard at creation.

Provides four conventional standards:

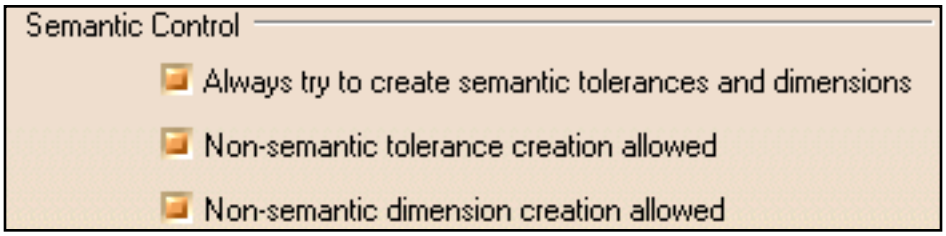
- ASME: American Society for Mechanical Engineers
- ANSI: American National Standards Institute
- ISO: International Organization for Standardization
- JIS: Japanese Industrial Standard

Also provides three CATIA-CADAM standards:

- CCDANSI: CATIA-CADAM American National Standards Institute
- CCDISO: CATIA-CADAM International Organization for Standardization

- CCDJIS: CATIA-CADAM Japanese Industrial Standard

Semantic Control



Always try to upgrade non-semantic tolerances and dimensions

Defines automatic upgrade of a non-semantic annotation during its creation to a semantic annotation. If it can be upgradeable to a semantic annotation only.

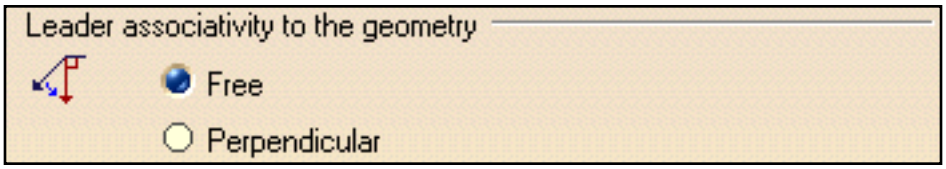
Non-semantic tolerance creation allowed

Defines whether non-semantic tolerances creation is allowed or not.

Non-semantic dimension creation allowed

Defines whether non-semantic dimensions creation is allowed or not.

Leader associativity to the geometry



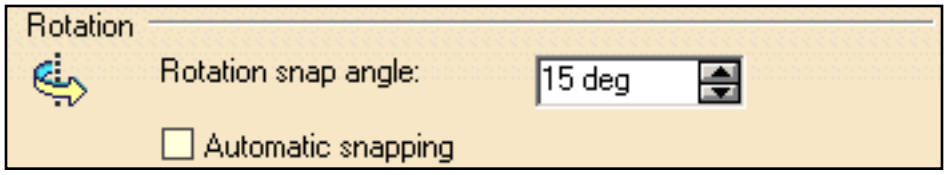
Free

Specifies that leader annotations are freely positioned relative to their geometrical elements.

Perpendicular

Specifies that leader annotations are positioned perpendicular to their geometrical elements.

Rotation



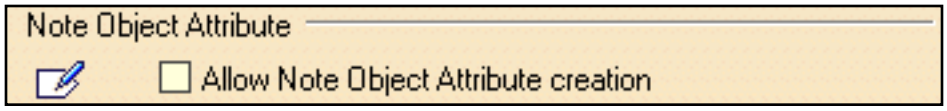
Rotation snap angle

Defines an angle value for rotating elements. This option is used to rotate text elements (text, frame, or leader).

Automatic snapping

Defines whether the rotation will be snapped to the angle value or not.

Note Object Attribute



Allow Note Object Attribute creation

Defines whether note object attribute may be created by user or not. Enable or disable the icon and menu item.

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Display

This task will show you how to customize 3D Functional Tolerancing & Annotation display settings.

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

   The **Options** dialog box appears.

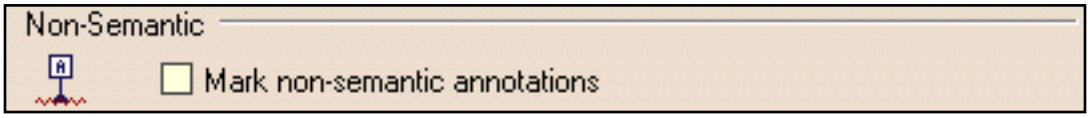
2. Click the **Mechanical Design** category.

3. Click the **3D Functional Tolerancing & Annotation** subcategory.

4. Click the **Display** tab.

   The **Display** tab appears, displaying the following categories of options:

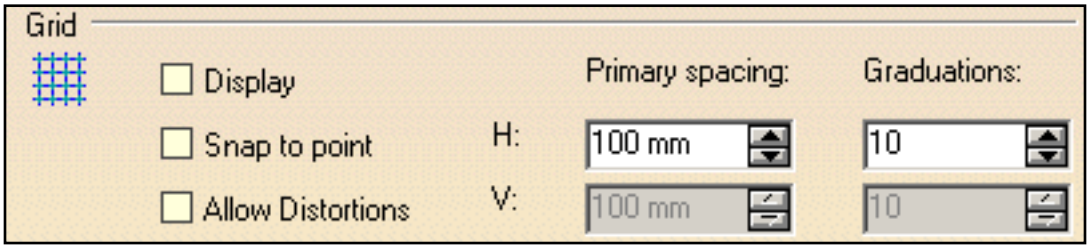
Non-Semantic



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.

Grid



Display

Defines whether the grid is displayed.

Snap to point

Defines whether annotations are snapped to the grid's point.

Allow Distortions

Defines whether grid spacing and graduations are the same horizontally and vertically.

H Primary spacing

Defines the grid's horizontal spacing.

H Graduations

Defines the grid's horizontal graduations.

V Primary spacing

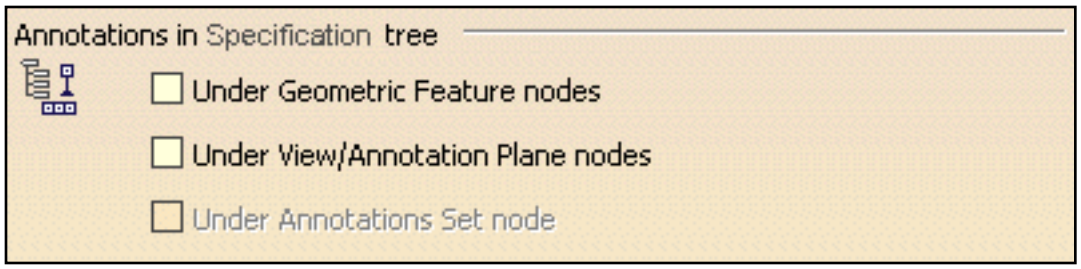
Defines the grid's vertical spacing (available only if Allow Distortions is checked).

V Graduations

Defines the grid's vertical graduations (available only if Allow Distortions is checked).



Annotations in Specification Tree



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 GSD feature nodes to which they are applied.

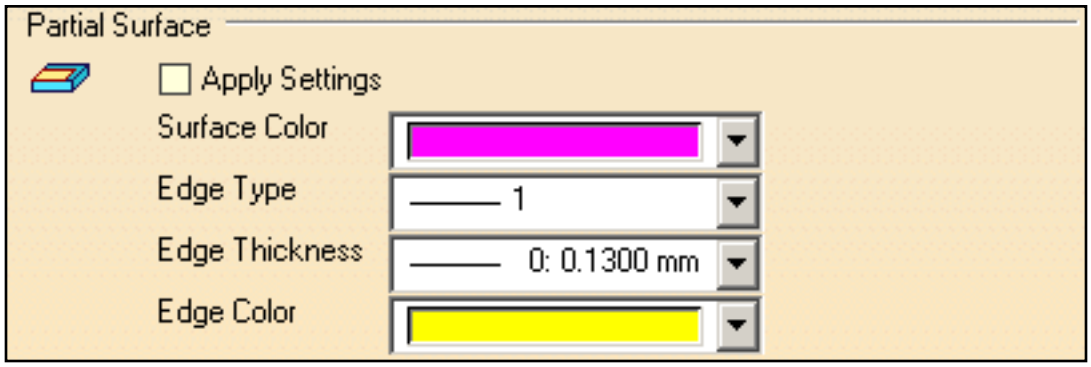
Under View / Annotation Plane nodes

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.

Under Annotations Set node

Defines that 3D annotations should be displayed under the annotation set node in the specification tree.

Partial Surface



Apply Settings

Defines whether the following settings are applied while creating a partial surface feature.

Surface Color

Defines the surface color of the partial surface.

Edge Type

Defines the edge type of the partial surface's border.

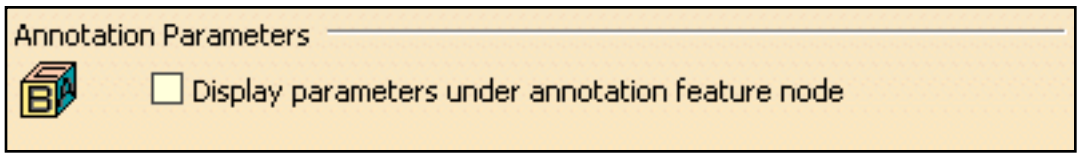
Edge Thickness

Defines the edge thickness of the partial surface's border.

Edge Color

Defines the edge color of the partial surface's border.

Annotation Parameters

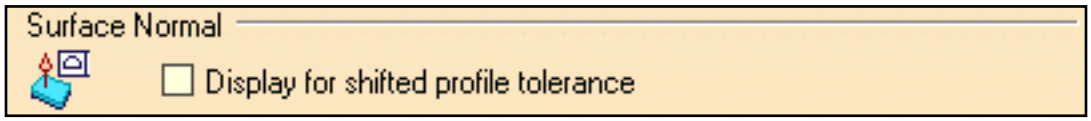


Display parameters under annotation feature node

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.

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.

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Constructed Geometry

This task will show you how to customize 3D Functional Tolerancing & Annotation tolerances settings.

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

   The **Options** dialog box appears.

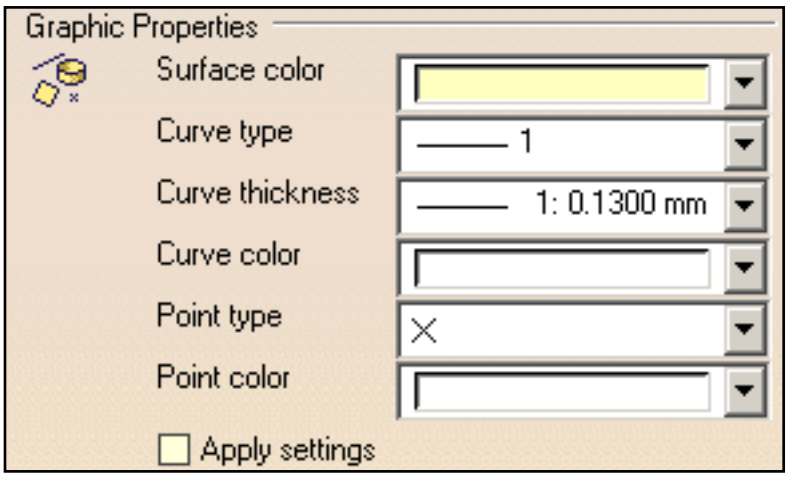
2. Click the **Mechanical Design** category.

3. Click the **3D Functional Tolerancing & Annotation** subcategory.

4. Click the **Constructed Geometry** tab.

   The **Constructed Geometry** tab appears, displaying three categories of options:

Graphic Properties



Surface color

Defines the constructed geometry surface color for plane and cylinder.

Curve type

Defines the constructed geometry curve type.

Curve thickness

Defines the constructed geometry curve thickness.

Curve color

Defines the constructed geometry curve color.

Point type

Defines the constructed geometry point type.

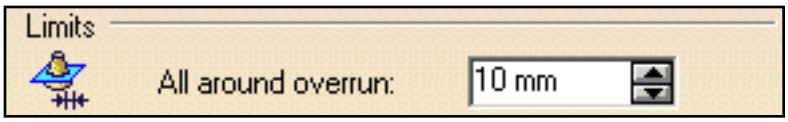
Point color

Defines the constructed geometry point color.

Apply settings

Defines whether all the graphic properties settings are applied or not.

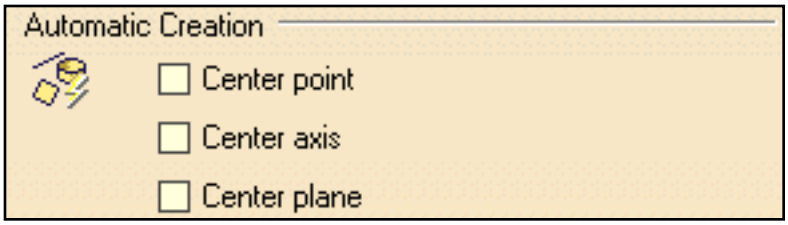
Limits



All around overrun

Defines the minimal limit between the constructed geometry and its related geometry.

Automatic Creation



Center point

Defines whether all the center point's constructed geometry is automatically created or not, for circle center, sphere center.

Center axis

Defines whether all the center axis's constructed geometry is automatically created or not, for cylinder, cone.

Center plane

Defines whether all the center plane's constructed geometry is automatically created or not, for slot.



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Manipulators

This task will show you how to customize 3D Functional Tolerancing & Annotation manipulators settings.

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

   The **Options** dialog box appears.

2. Click the **Mechanical Design** category.

3. Click the **3D Functional Tolerancing & Annotation** subcategory.

4. Click the **Manipulators** tab.

   The **Manipulators** tab appears, displaying two categories of options:

Manipulators



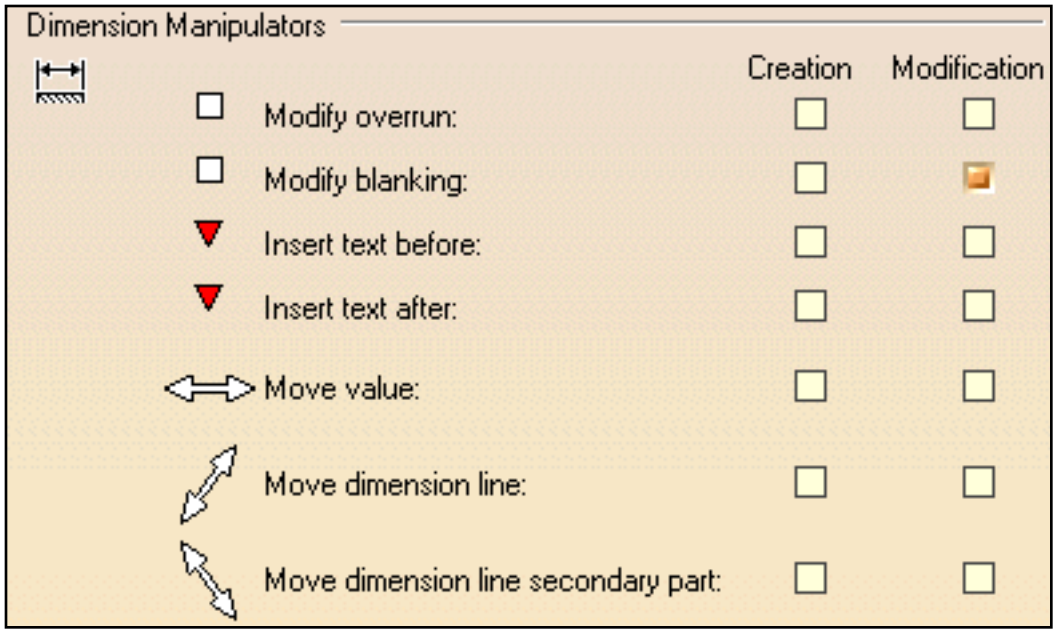
Reference size

Defines the annotation manipulator's size.

Zoomable

Defines whether the annotation manipulator is zoomable or not.

Dimension Manipulators



Modify overrun

Defines whether overrun extension lines can be modified during creation or modification, or not.

Modify blanking

Defines whether blanking can be modified during creation or modification, or not.

Insert text before

Defines whether a text before can be inserted during creation or modification, or not.

Insert text after

Defines whether a text after can be inserted during creation or modification, or not.

Move value

Defines whether only the value can be moved during creation or modification, or not.

Move dimension line

Defines whether only the dimension line can be moved during creation or modification, or not.

Move dimension line secondary part

Defines whether only the dimension line secondary part can be moved during creation or modification, or not.



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Dimension

This task will show you how to customize 3D Functional Tolerancing & Annotation dimension settings.

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

   The **Options** dialog box appears.

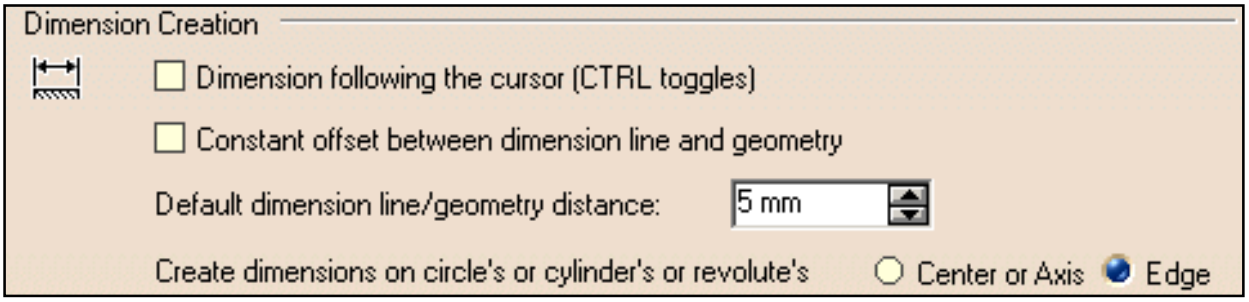
2. Click the **Mechanical Design** category.

3. Click the **3D Functional Tolerancing & Annotation** subcategory.

4. Click the **Dimension** tab.

   The **Dimension** tab appears, displaying two categories of options:

Dimension Creation



Dimension following the mouse

Defines whether the dimension line is positioned according to the cursor, following it dynamically during the creation process or not.

End dimension creation at line-up

Defines whether a dimension aligned to another automatically ends the command or not.

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.

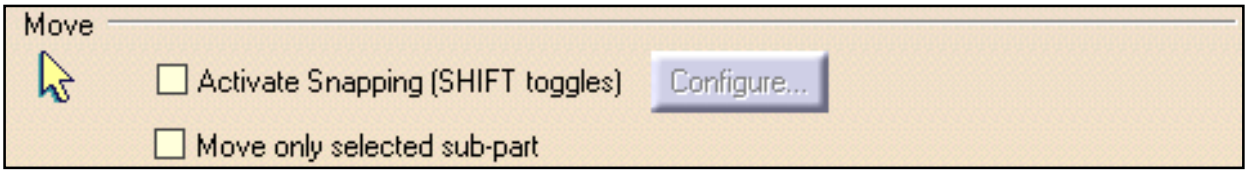
Default dimension line/geometry distance

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, dimension circles on their

Defines the dimension you will create between a circle and another element will be either on the circle center or on the circle edge.

Move



Activate Snapping

Defines whether the dimension will be snapped on the grid and/or the dimension value will be located at its default position between symbols (it will work only if the cursor is between the symbols) or not.

Move only selected sub-part

Defines whether only a dimension sub-part (text, line, etc...) will be moved or not.

Back Up Next

Annotation

This task will show you how to customize 3D Functional Tolerancing & Annotation annotation settings.

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

   The **Options** dialog box appears.

2. Click the **Mechanical Design** category.

3. Click the **3D Functional Tolerancing & Annotation** subcategory.

4. Click the **Annotation** tab.

   The **Annotation** tab appears, displaying two categories of options:

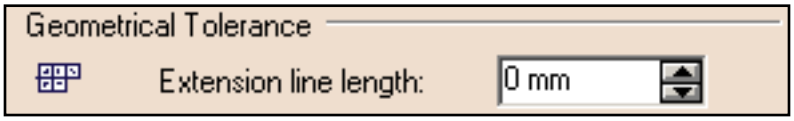
Annotation Creation



Annotation following the mouse (Ctrl toggles)

Defines whether the annotation is positioned according to the cursor, following it dynamically during the creation process or not.

Geometrical Tolerance



Extension line length

Defines the extension line length between the geometrical frame and the its leader.



Tolerances

This task will show you how to customize 3D Functional Tolerancing & Annotation tolerances settings.

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

   The **Options** dialog box appears.

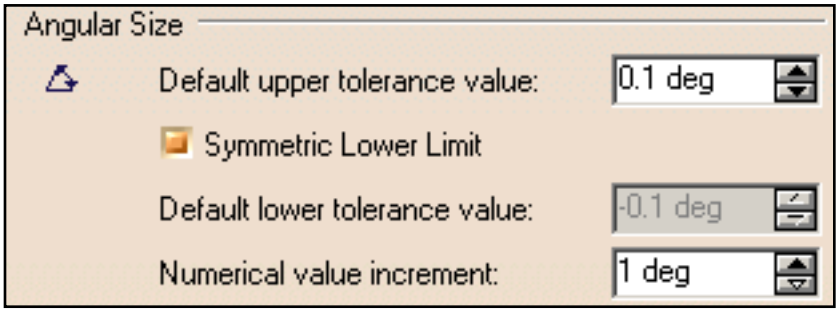
2. Click the **Mechanical Design** category.

3. Click the **3D Functional Tolerancing & Annotation** subcategory.

4. Click the **Tolerances** tab.

   The **Tolerances** tab appears, displaying three categories of options:

Angular Size



Default upper tolerance value

Defines the default upper tolerance value for angular size.

Symmetric lower limit

Defines whether the default lower tolerance value is symmetric in relation to the default upper tolerance value.

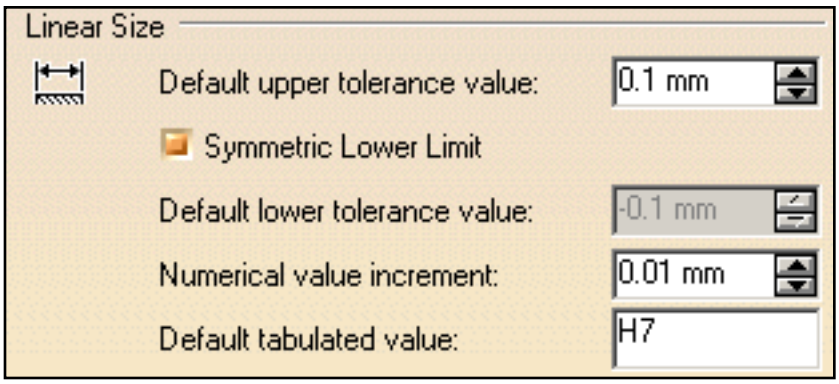
Default lower tolerance value

Defines the default lower tolerance value for angular size, disable when Symmetric lower limit is checked.

Numerical value increment

Defines the increment for angular size numerical value.

Linear Size



Default upper tolerance value

Defines the default upper tolerance value for linear size.

Symmetric lower limit

Defines whether the default lower tolerance value is symmetric in relation to the default upper tolerance value.

Default lower tolerance value

Defines the default lower tolerance value for linear size, disable when Symmetric lower limit is checked.

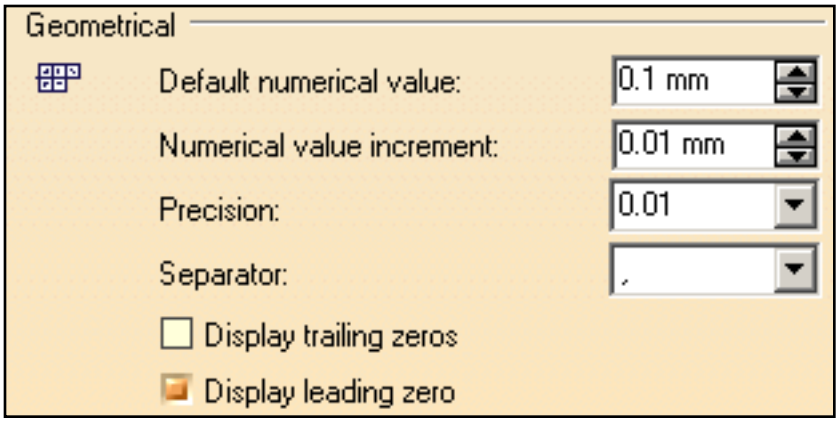
Numerical value increment

Defines the increment for linear size numerical value.

Default tabulated value

Defines the default tabulated for linear size.

Geometrical



Default numerical value

Defines the default numerical for geometrical tolerance.

Numerical value increment

Defines the increment for the to geometrical tolerance numerical value.

Precision

Defines the geometrical tolerance precision after the numerical separator.

Separator

Defines the geometrical tolerance symbol used as numerical separator.

Display trailing zeros

Defines whether "0"s complete the number of digit displayed after the separator, according to the precision, or not.

Display leading zeros

Defines whether the "0" before the numerical separator, when value is less than 1, is displayed or not.

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View/Annotation Plane

This task will show you how to customize 3D Functional Tolerancing & Annotation view/annotation plane settings.

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

   The **Options** dialog box appears.

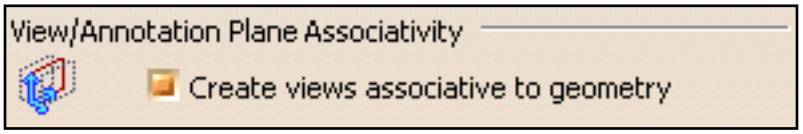
2. Click the **Mechanical Design** category.

3. Click the **3D Functional Tolerancing & Annotation** subcategory.

4. Click the **View / Annotation Plane** tab.

   The **View / Annotation Plane** tab appears, displaying the following categories of options:

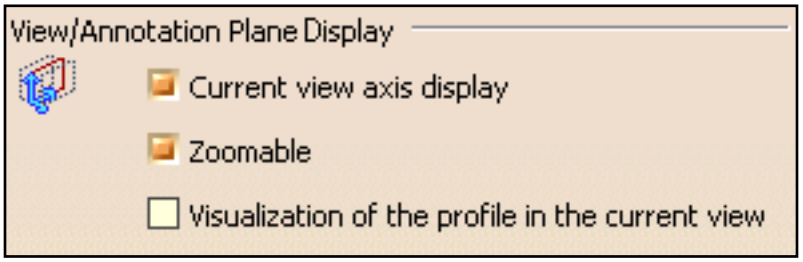
View/Annotation Plane Associativity



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.

View/Annotation Plane Display



Current view axis display

Defines whether the active annotation plane axis system is displayed.

Zoomable

Defines whether the annotation plane axis is zoomable.

Visualization of the profile in the current view

Defines whether the view/annotation plane profile on the part/product is displayed or not.



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Reference Information

This section contains reference information about 3D Functional Tolerancing & Annotations workbench.

Normative References
Principles and Fundamental Rules for Geometrical Tolerancing
Geometrical Tolerancing
Symbols for Geometrical Tolerances
Symbols for Modifiers
Datum Principles
Concepts
Standards
Properties
Dimension Units
Statistic Laws

Normative References

The table below provides a list of normative references.

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.
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.

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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).

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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 datums 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.



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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

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Symbols for Modifiers

The list of available symbols for modifiers.

Least Material Condition	Symbols
Maximum Material Condition (MMC)	Ⓜ
Least Material Condition (LMC)	Ⓛ
Tangent plane	Ⓣ
Regardless of Feature of Size (RFS)	Ⓢ
Free State	Ⓕ
Projected Tolerance zone	Ⓟ

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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.

When the identifiers are specified separately in the same frame of the tolerance frame, no hierarchy is established between the datum elements.



This notation only exists in ISO. However, the ISO recommends not using this specification any longer.

Two measurement computations have to be performed: the datum elements have to be fitted successively following their specification order and following the opposite specification order.

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.

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Concepts

3D Annotation and Annotation Plane

Non-semantic and Semantic Usage

Note Object Attribute

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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, DYN...).

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...

Roughness

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.



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Non-semantic and Semantic Usage

The goal of the Functional Tolerancing & Annotation product is to fully covered 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 Hole

Framed dimensions (basic dimensions) as defined in standards are not yet supported as semantic in Functional Tolerancing & Annotation.
In this case use non-semantic dimensions.

According to ISO and ASME/ANSI standards, framed dimensions (basic dimensions) are used to define:

- Position and orientation of the tolerance display.
- Position, size and orientation of partial surfaces and datum targets.
- Position and orientation of Datum Reference Frame.

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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.

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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.

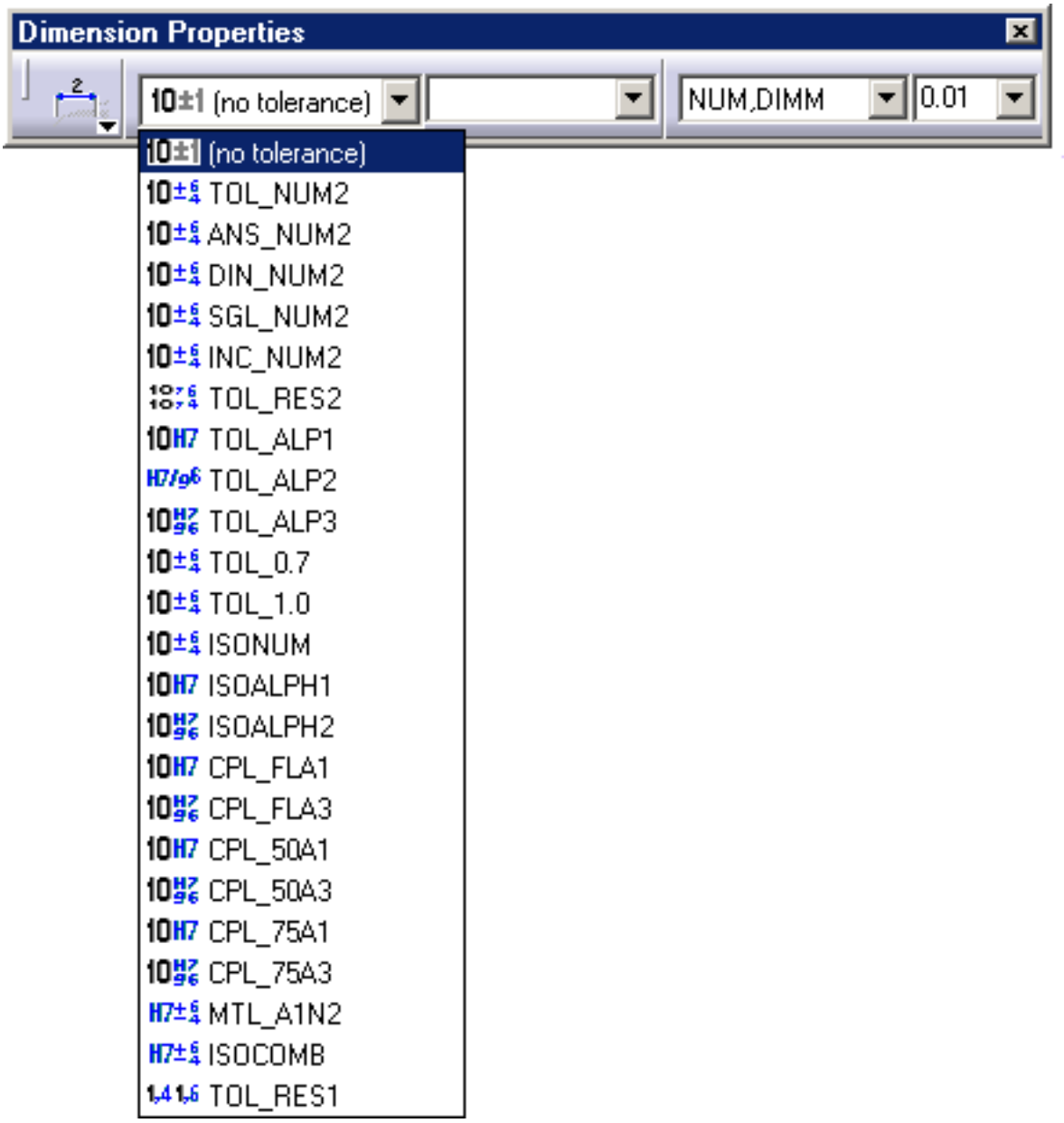
The values of the parameters in the file are taken into account when the first Functional Tolerancing and Annotation view is created, based on the current parent standard. Once the first view has been created, modifying the standard file will not affect this CATPart document.

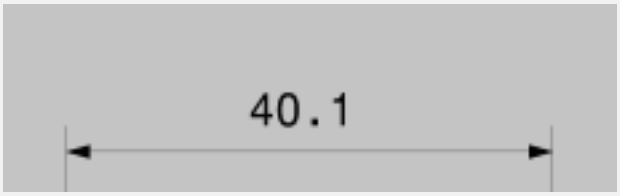
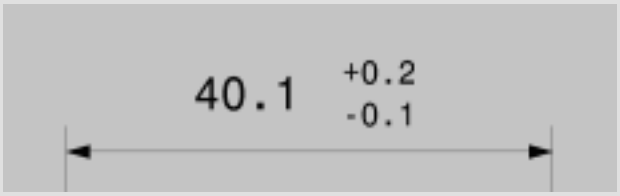
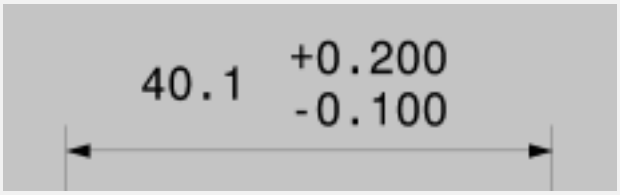
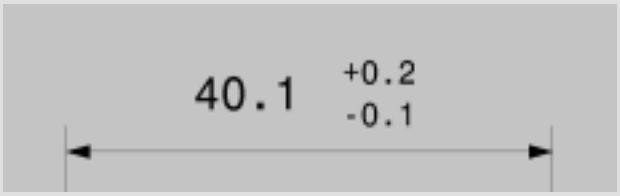
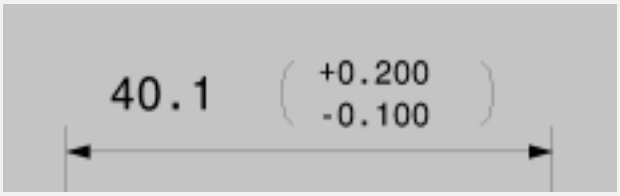
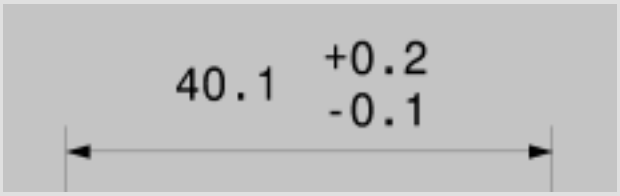
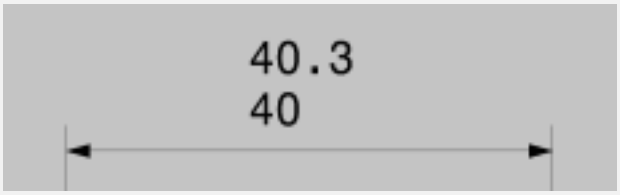
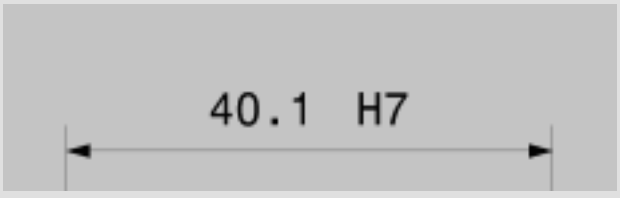
For more information on standards and their administration, refer to the User Tasks > Administration Tasks chapter in the *Interactive Drafting User's Guide.* Also refer to Dimension Tolerance Display and to Dimension Numerical Display in this guide.

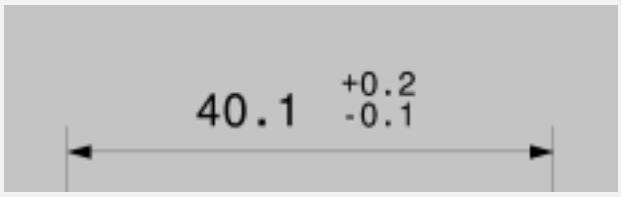
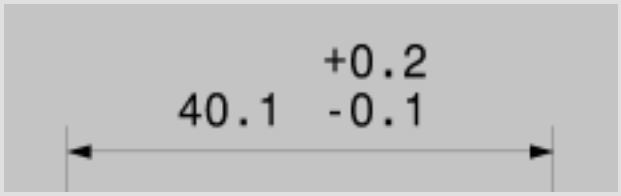
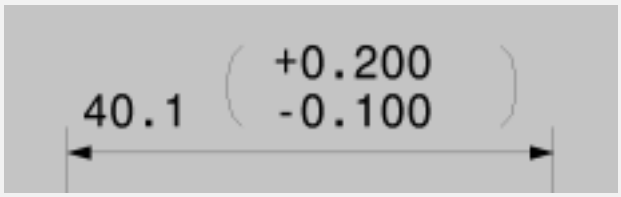
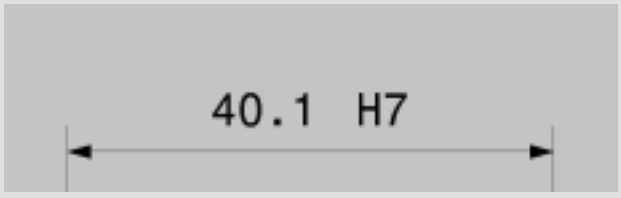
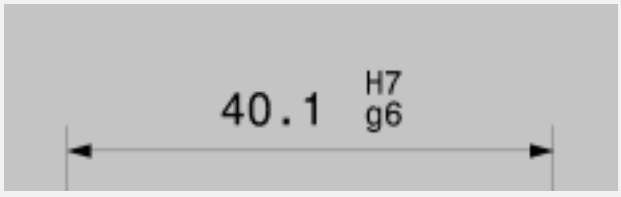
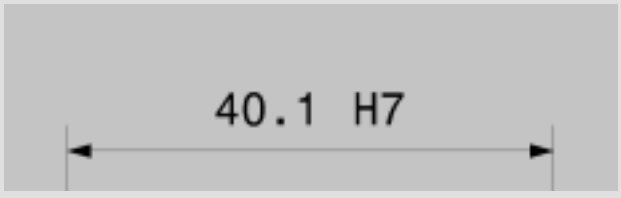
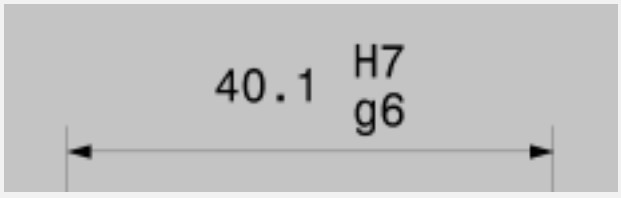
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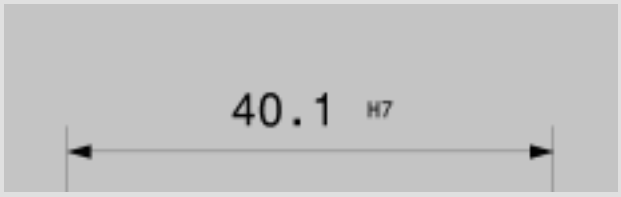
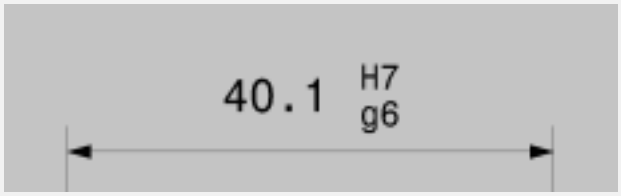
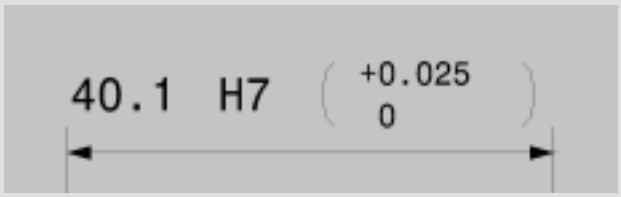
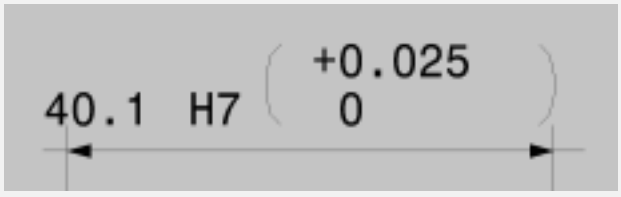
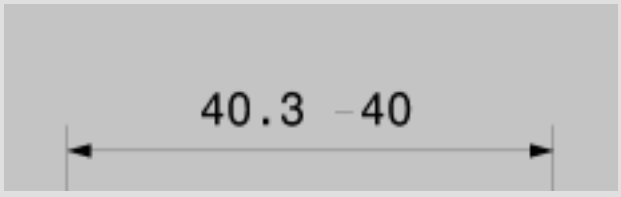
Dimension Tolerance Display

Dimension tolerance display formats are listed below. These formats are available in the **Properties** dialog box, **Tolerance** tab.



Name	Display	Description
10±1 (no tolerance)		No tolerance displayed.
10±6/4 TOL_NUM2		Numerical superimposed (small).
10±6/4 ANS_NUM2		Numerical superimposed with trailing zeros (large).
10±6/4 DIN_NUM2		Numerical superimposed (small).
10±6/4 SGL_NUM2		Numerical superimposed with trailing zeros between parentheses (small).
10±6/4 INC_NUM2		Numerical superimposed (large).
10 7.6/4 TOL_RES2		Numerical resolved, superimposed.
10H7 TOL_ALP1		Alphanumerical single value (large).

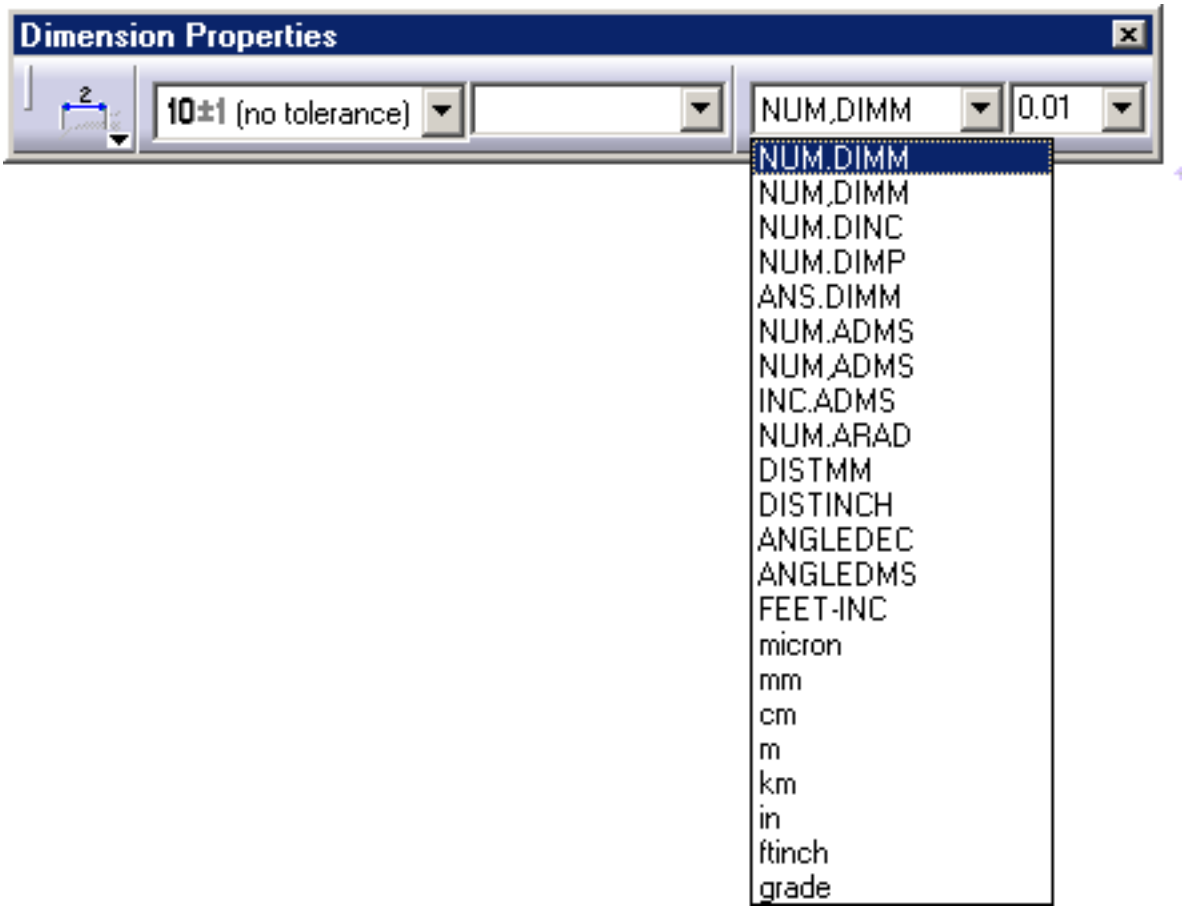
TOL_ALP2		Alphanumerical double value side-by-side (large).
TOL_ALP3		Alphanumerical double value superimposed (small).
TOL_0.7		Numerical superimposed (small).
TOL_1.0		Numerical superimposed (large).
ISONUM		Numerical superimposed with trailing zeros between parentheses (large).
ISOALPH1		Alphanumerical single value (large).
ISOALPH2		Alphanumerical double value superimposed (small).
CPL_FLA1		Alphanumerical single value (large).
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±6/4 MTL_A1N2		Alphanumerical single value (large) and numerical superimposed between parentheses (small).
H7±6/4 ISOCOMB		Alphanumerical single value (large) and numerical superimposed between parentheses (large).
1,4 1,6 TOL_RES1		Numerical resolved, side-by-side.

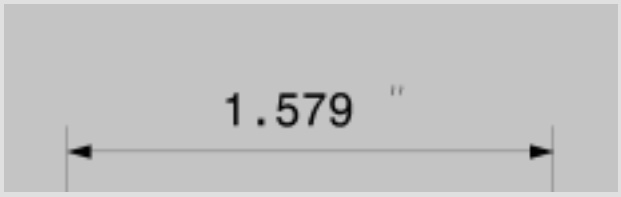
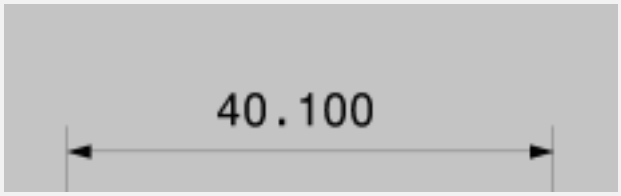
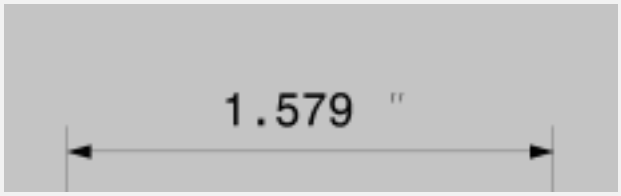
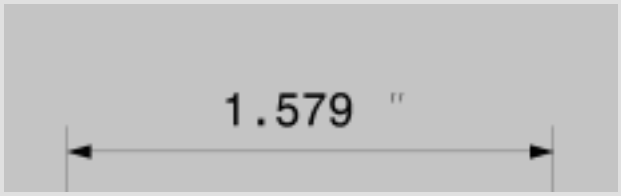
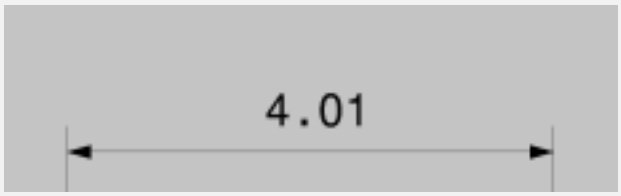
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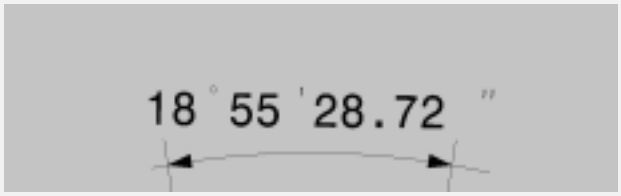
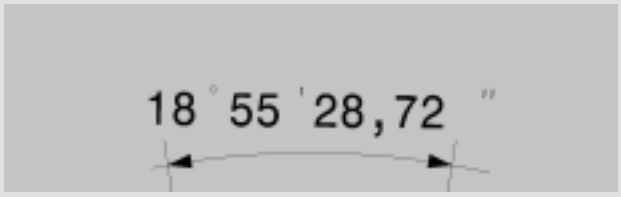
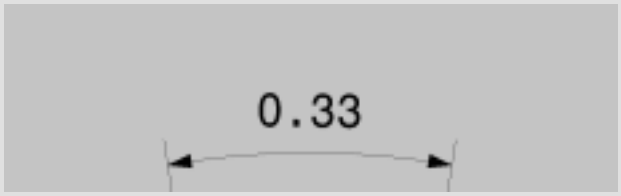
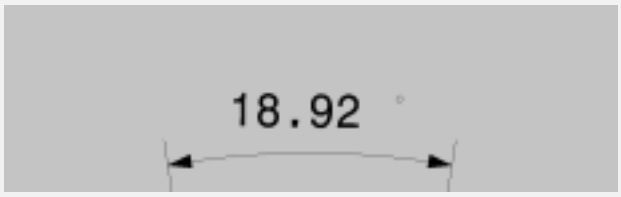
Dimension Numerical Display

Dimension numerical display formats are listed below. These formats are available in the **Properties** dialog box, **Value** tab.



Name	Display	Description
NUM.DIMM	40.1	Millimeters with dot.
NUM,DIMM	40,1	Millimeters with comma.
NUM.DINC	1.579	Inches with trailing zeros.

NUM.DIMP		Inches with unit display.
ANS.DIMM		Millimeters with trailing zeros.
DISTMM		Millimeters with dot.
DISTINCH		Inches with unit display
FEET-INC		Feet and Inches with unit display
micron		Microns.
mm		Millimeters.
cm		Centimeters.
m		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.

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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.

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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:

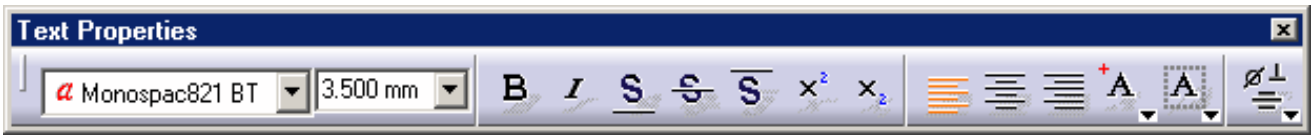
Along the top of the text: left, center, or right.

Along the vertical center of text: left, center or right.

Along the bottom of the text: left, center, or right.

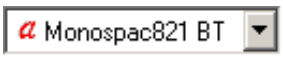
Frame: draws a single-line frame around the text. The frame size can be variable, or fixed.

Insert Symbol: inserts several symbol types including geometrical tolerancing ones especially in the text editor.



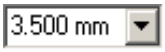
Font Name

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.



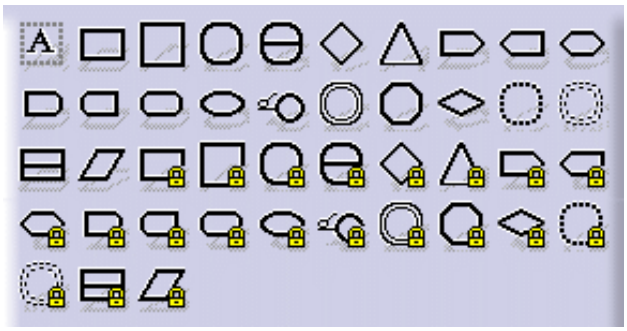
Anchor Point

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.



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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 **Set as Default** command.
See Managing Graphical Properties.

Properties			Application Default Properties	User Default Properties	Only User Default Properties
Font	Font	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Style	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Size	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	UnderLine	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Color	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Ratio	Yes	Application Default	Set as Default	Set as Default
	Strikethrough	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Superscript	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Subscript	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Overline	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
Text	Frame	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Color	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Thickness	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Line Type	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Anchor point	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Anchor Line	Yes	Application Default	Application Default	Set as Default

	Line Spacing	Yes	Application Default	Set as Default	Set as Default
	Line Spacing Mode	Yes	Application Default	Set as Default	Set as Default
	Justification	Yes	Text Properties Toolbar	Set as Default	Set as Default
	Word wrap	Yes	Application Default	Set as Default	Set as Default
	Reference	Yes	Application Default	Set as Default	Set as Default
	Orientation	Yes	Application Default	Set as Default	Set as Default
	Angle	Yes	Application Default	Set as Default	Set as Default
	Mirroring	Yes	Application Default	Set as Default	Set as Default
	Auto flip	Yes	Application Default	Set as Default	Set as Default
Graphic	Color	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Linetype	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Thickness	Yes	Text Properties Toolbar	Text Properties Toolbar	Set as Default
	Pickable	No	Application Default	Application Default	Application Default

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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:

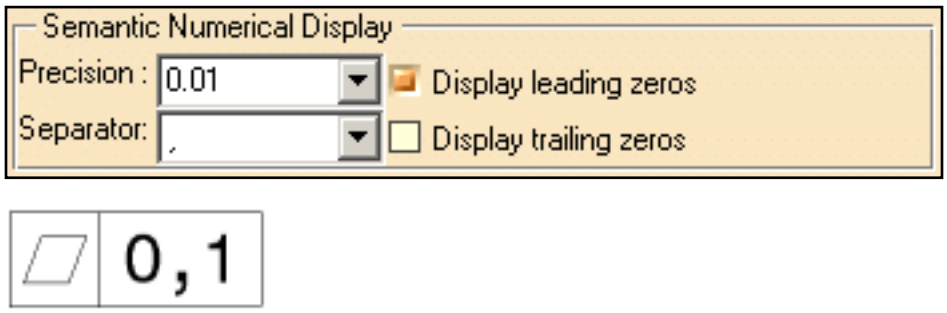
Precision: defines the number of digit displayed after the separator.

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.

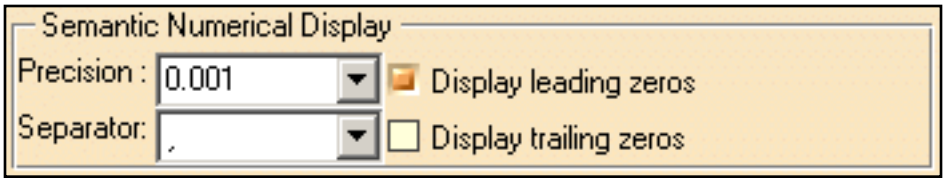
Default settings according to the ISO standard

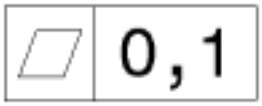


In the following examples we start from the ISO default settings.

Precision

Set the precision to **0.001**

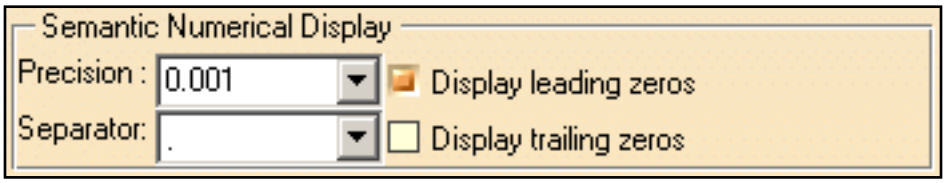


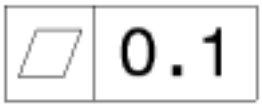


According to the unchecked **Display trailing zeros** setting only the effective digit after the separator are displayed.

Separator

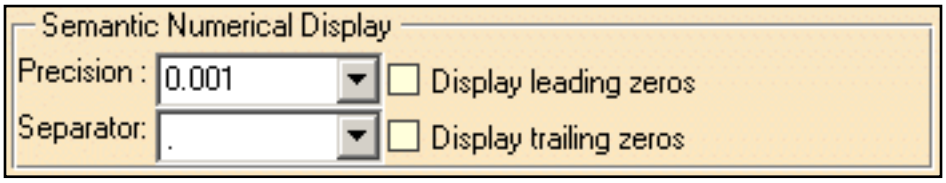
Change the separator symbol for the **.** (dot)

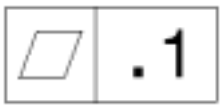




Display leading zeros

Uncheck the **Display leading zeros** setting.

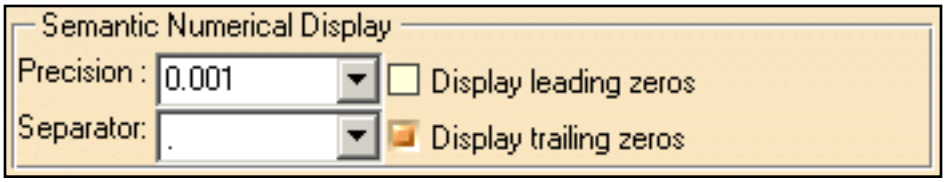


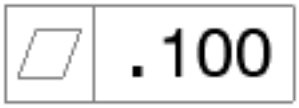


According to the unchecked setting the "0" is removed before the separator.

Display trailing zeros

Check the **Display trailing zeros** setting.



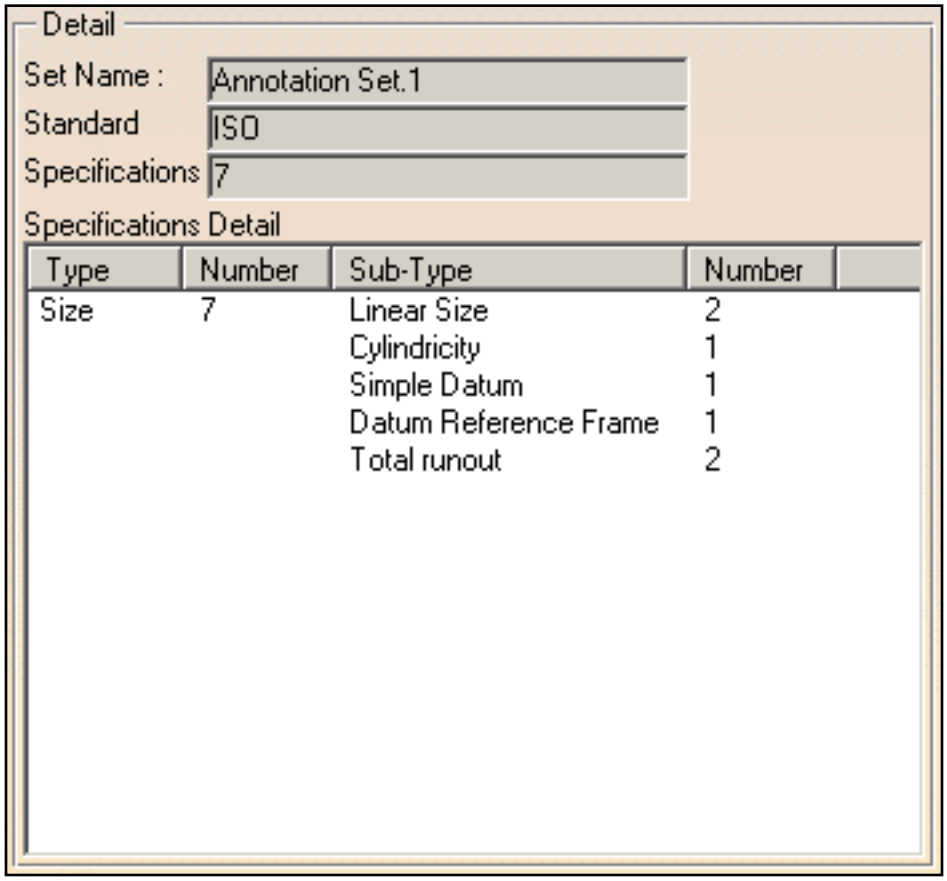


According to the setting "0" are added to complete the precision format.

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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	

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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.



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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.

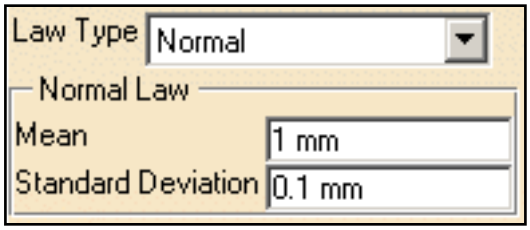
Pearson Law: describe the Pearson probability law equations.

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:

$$SD(X) = \sigma$$

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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:

$$E(X) = \frac{(b-a)}{2}$$

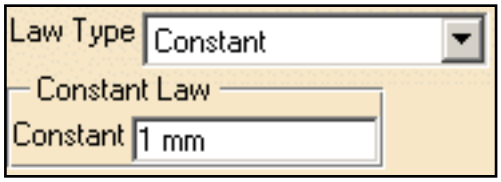
With variance:

With standard deviation:

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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:

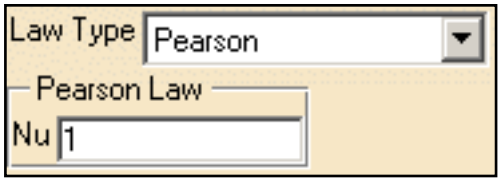
$$SD(X) = 0$$

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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 $Y_i \ (i = 1,...,\nu)$ 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:

$$Var(\chi^2) = 2\nu$$

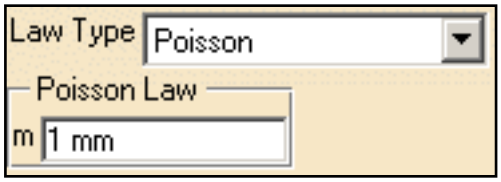
With standard deviation:

$$SD(\chi^2) = \sqrt{2\nu}$$

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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:

$$Var(X) = m$$

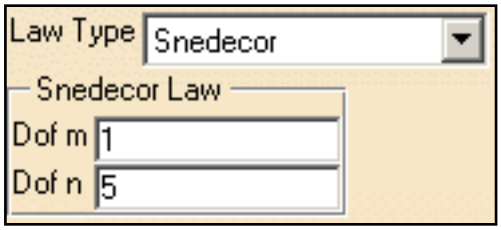
With standard deviation:

$$SD(X) = \sqrt{m}$$

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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:

$$Var\ (F) = \frac{2n^2(m+n-2)}{m(n-2)^2(n-4)}$$

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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))



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