Generative Shape Design & Optimizer

Overview

Welcome to the *Generative Shape Design User's Guide* !
This guide is intended for users who need to become quickly familiar with the product.

This overview provides the following information:

- Generative Shape Design in a Nutshell
- Before Reading this Guide
- Getting the Most Out of this Guide
- Accessing Sample Documents
- Conventions Used in this Guide

Generative Shape Design in a Nutshell

The Generative Shape Design workbench allows you to quickly model both simple and complex shapes using wireframe and surface features. It provides a large set of tools for creating and editing shape designs and, when combined with other products such as Part Design, it meets the requirements of solid-based hybrid modeling.

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

This new application is intended for both the expert and the casual user. Its intuitive interface offers the possibility to produce precision shape designs with very few interactions. The dialog boxes are self explanatory and require practically no methodology, all defining steps being commutative.

As a scalable product, Generative Shape Design can be used with other Version 5 products such as Part Design and FreeStyle Shaper and Optimizer. 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.

This User's Guide has been designed to show you how to create and edit a surface design part. There are numerous techniques to reach the final result. This book aims at illustrating these various possibilities.

Before Reading this Guide

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

You may also like to read the following complementary product guides:

- *Part Design User's Guide*

Getting the Most Out of this Guide

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

Once you have finished, you should move on to the Basic Tasks and Advanced Tasks sections, which deal with handling all the product functions.

The Workbench Description section, which describes the Generative Shape Design workbench, and the Customizing section, which explains how to set up the options, will also certainly prove useful.

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

Accessing Sample Documents

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

Conventions

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

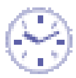
Graphic Conventions

The three categories of graphic conventions used are as follows:

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

Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted as follows:

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

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

Graphic Conventions Indicating the Configuration Required

Graphic conventions indicating the configuration required are denoted as follows:

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

Graphic Conventions Used in the Table of Contents

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

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

Index

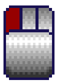
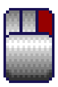
Text Conventions

The following text conventions are used:

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

How to Use the Mouse

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

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

What's New?

New Functionalities

Managing Multi-Result Operations
Creating Variable Offset Surfaces
Creating Rough Offset Surfaces
Inserting a Body into an Ordered Geometrical Set

Creating Volumes

Creating a Multi-Sections Volume
Creating a Swept Volume
Creating a Draft
Creating a Variable Angle Draft
Creating a Draft from Reflect Lines
Creating a Shell
Creating a Sew Surface
Trimming Volumes

Working With the BiW Templates

Creating a Bead

Enhanced Functionalities

Creating Wireframe Geometry

Creating Points
- New Circle/Sphere center sub-type

Creating Lines
- New Up to option to create a line up to an element (point, curve, or surface)

Creating Circles
- New Center and axis sub-type with or without projection
- New Dia/Radius button to switch to a Diameter or Radius value
- New Axis computation button to create axis features while creating or modifying a circle

Creating Parallel Curves
- The second parallel curve created using the "Both Sides" option is now aggregated under the first one

Creating Surfaces

Creating Offset Surfaces
- New Automatic smoothing to clean the geometry of a surface
- New flag notes to obtain accurate diagnosis for each erroneous sub-element

New Temporary Analysis mode to check connections between surfaces or curves

Performing Operations on Shape Geometry

Smoothing Curves

The maximum deviation is now displayed on the curve

You can specify a continuity mode when smoothing the curve: point, tangent or curvature continuity

Splitting Geometry

You can now select several elements to cut

Extracting Geometry

You can now select a volume as the extracted element

Performing a Symmetry on Geometry

You can now select an axis system as the element to be transformed by symmetry

Transforming Elements from an Axis to Another

You can now select an axis system as the element to be transformed into a new axis system

Creating Shape Fillets

You can now create a bitangent fillet with relimiters

New Law button to select an external law

Extrapolating Curves

New Up to option to extrapolate a curve up to another curve

Using Tools

Defining an Axis System

You can now choose the destination of the axis system

Working With a Support

You can now create an infinite plane from a limited planar surface

Working With a 3D Support

You can now featurize the grid lines as lines or planes

Analyzing Using Parameterization

New Bodies filter

Selecting Using Multi-Output

You can now select a geometrical set or a multi-output feature as input

Managing Geometrical Sets and Ordered Geometrical Sets

Managing Ordered Geometrical Sets

You can now insert a body into an ordered geometrical set.

Patterning

Creating Rectangular Patterns

You can now duplicate volumes

Creating Circular Patterns

You can now assign distinct angle values between each instance

You can now duplicate volumes

Working with the Generative Shape Optimizer Workbench

Deforming Surfaces According to Curve Wrapping

You can now keep the tangency constraint on the first, and/or last, pair of curves

Deforming Surfaces According to Surface Wrapping

Automatic extrapolation of the reference and target surfaces when necessary

Working with the Developed Shapes Workbench

Unfolding a Surface

A new tab lets you transfer curves or points

Working With the BiW Templates

Creating a Mating Flange

New Local thickness option

New Both sides option

Customizing

General Settings

Tolerant laydown is now available with the Fill and Extrapol commands

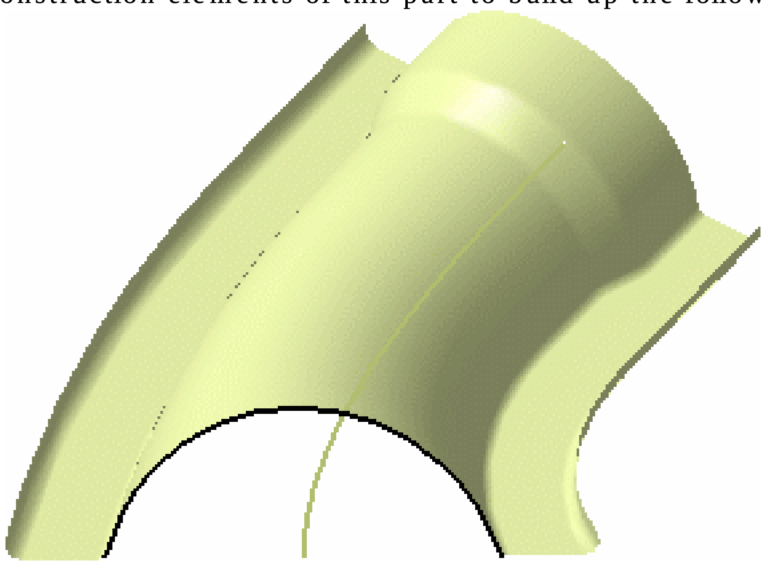
New Stacked analysis option to set the analysis as temporary

Getting Started

Before getting into the detailed instructions for using Generative Shape Design, the following tutorial aims at giving you a feel of 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:

This tutorial should take about 20 minutes to complete.

You will use the construction elements of this part to build up the following shape design.



Entering the Workbench

This first task shows you how to enter the Shape Design workbench and open a wireframe design part.

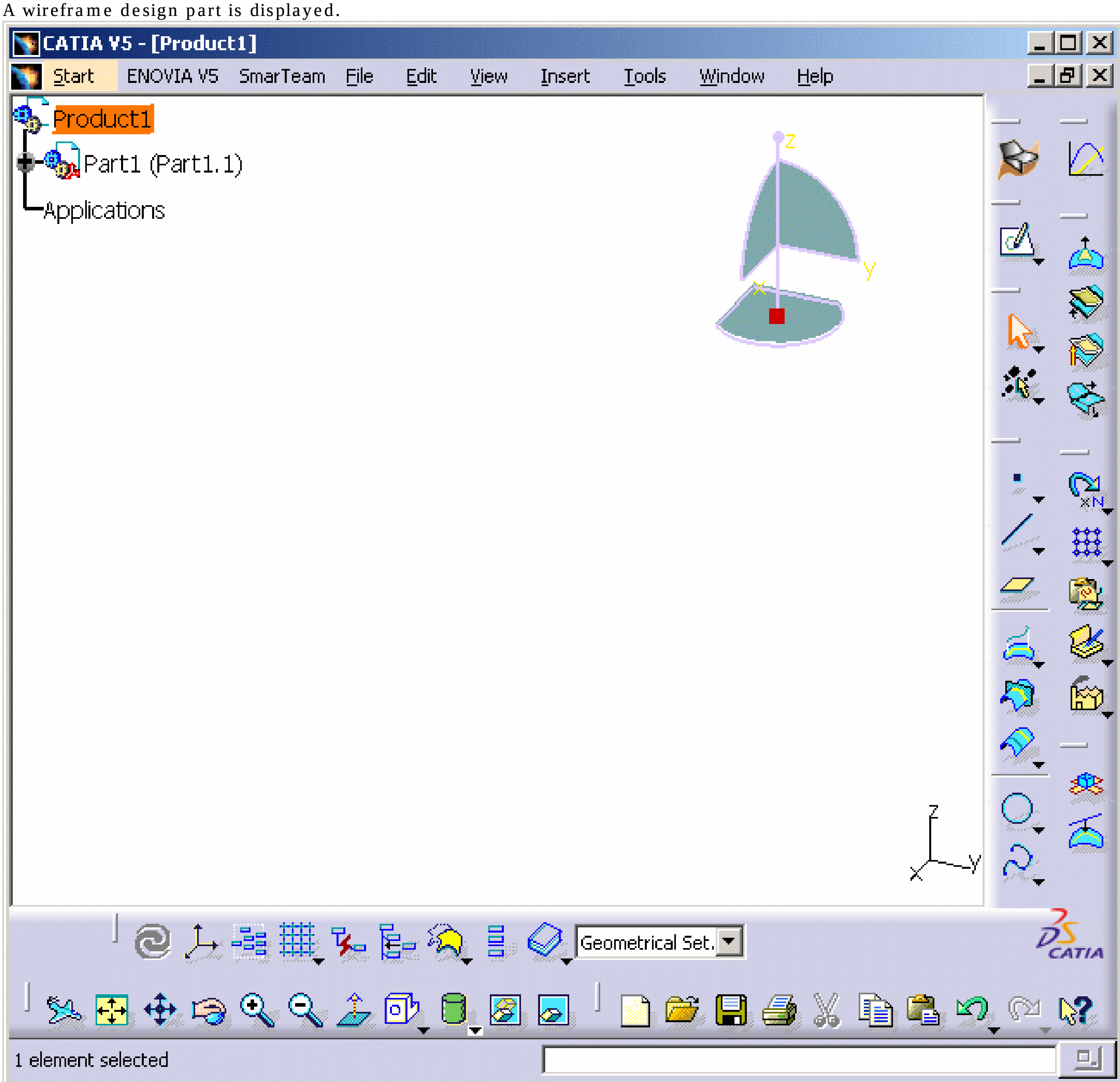
Before starting this scenario, you should be familiar with the basic commands common to all workbenches. These are described in the *Infrastructure User's Guide*.

1. Select **Shape -> Generative Shape Design** from the **Start** menu.

The Shape Design workbench is displayed.

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

A wireframe design part is displayed.



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.

Lofting, Offsetting and Intersecting

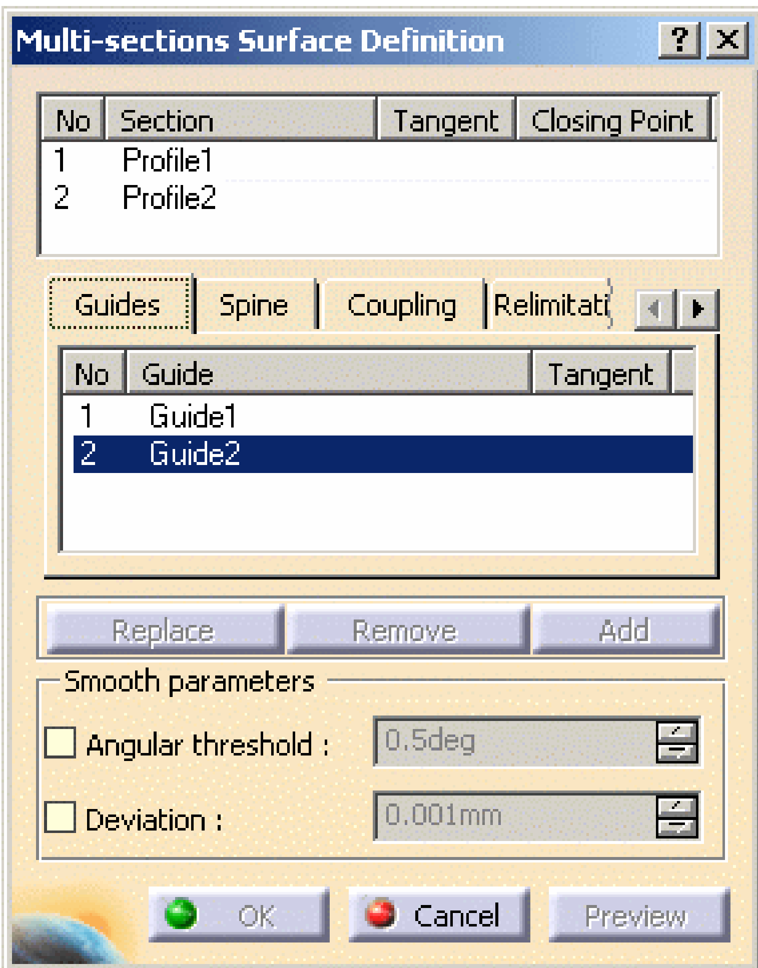
This task shows you how to create a multi-sections and an offset surfaces as well as an intersection.

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

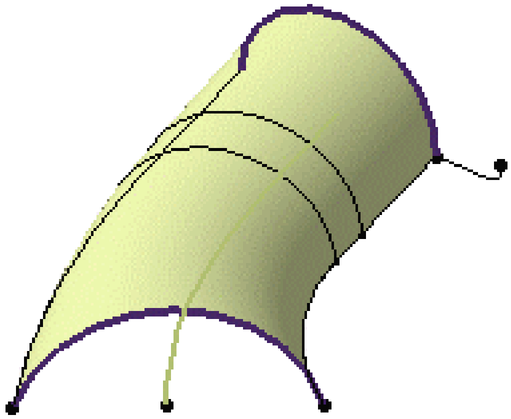
The Multi-sections Surface Definition dialog box appears.

2. Select the two section curves.

3. Click within the Guides window then select the two guide curves.



4. Click **OK** to create the multi-sections surface.

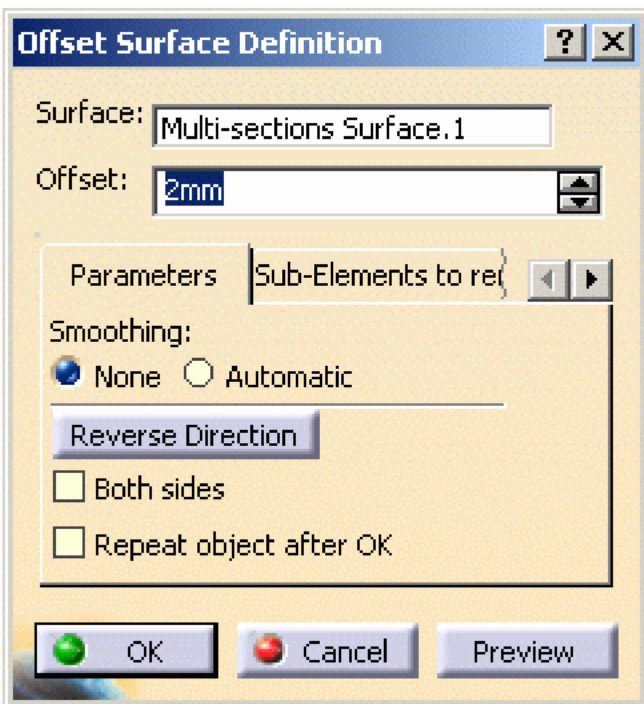


5. Click the **Offset** icon .

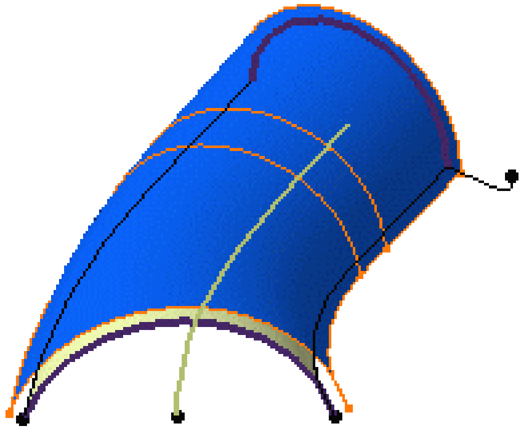
6. Select the multi-sections surface.

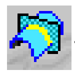
7. Enter an offset value of 2mm.

The offset surface is displayed normal to the multi-sections surface.



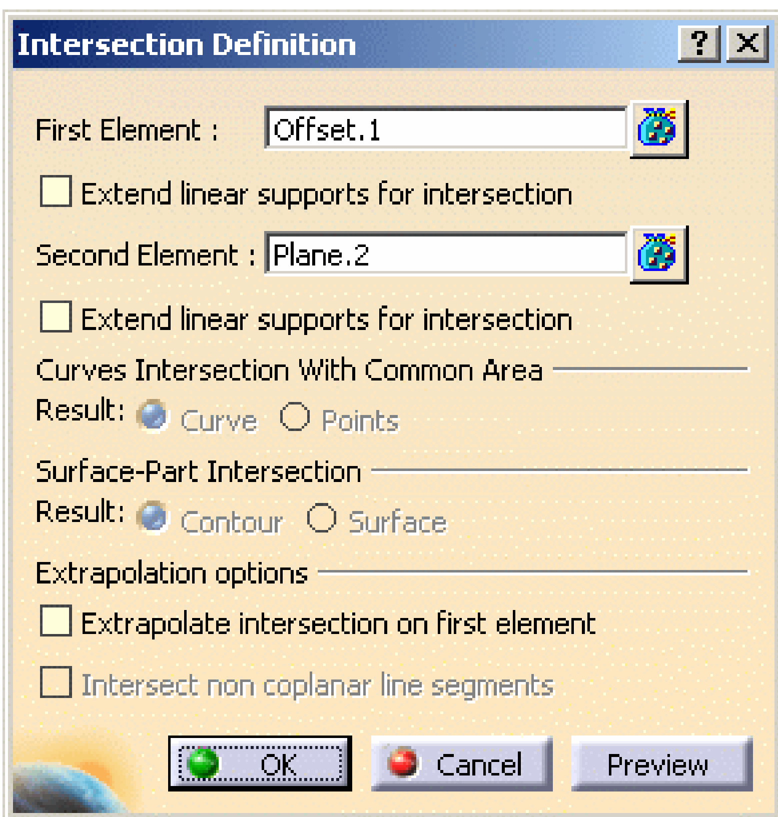
8. Click OK to create the offset surface.



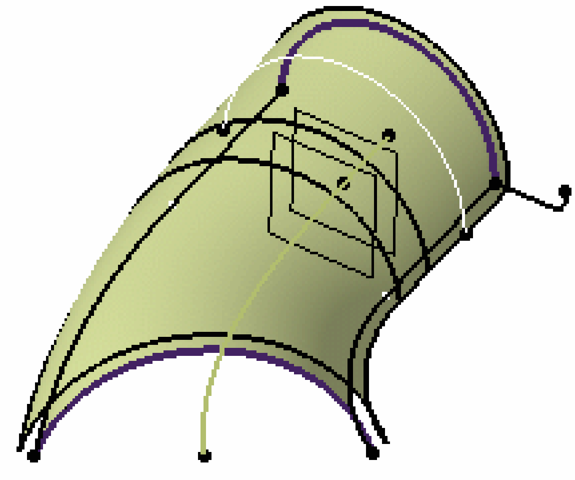
9. Click the **Intersection** icon .

The Intersection dialog box appears.

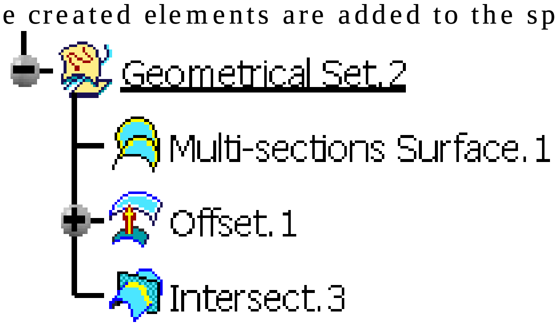
10. Select the offset surface then the first plane (Plane.2) to create the intersection between these two elements.



11. Click OK in the dialog box.



The created elements are added to the specification tree:



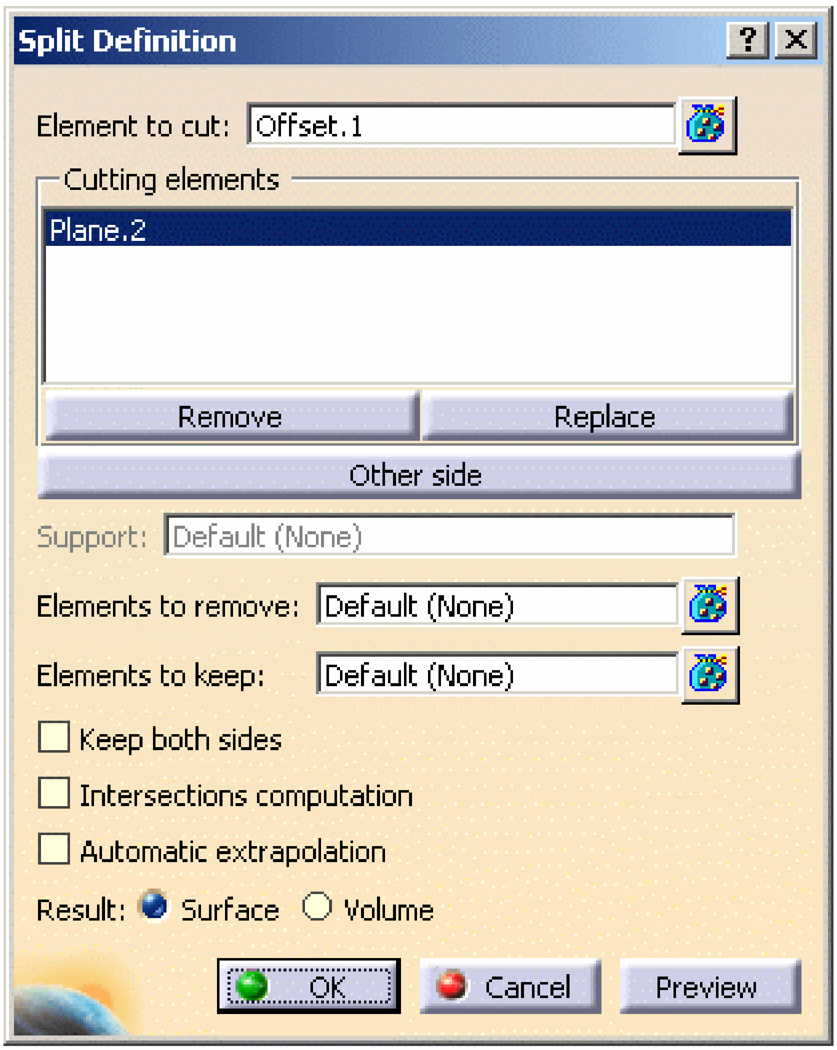
Splitting, Lofting and Filleting

This task shows how to split surfaces then create a multi-sections surface and two fillets.

1. Click the **Split** icon .

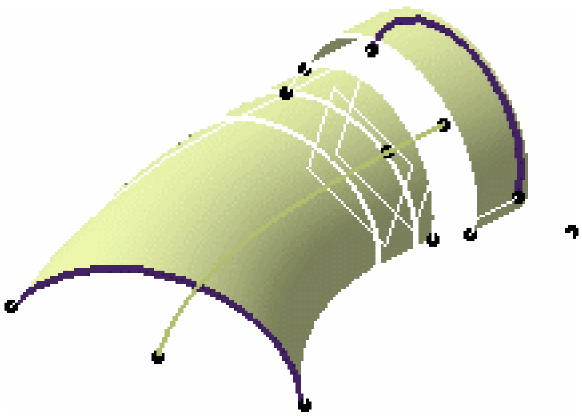
The Split Definition dialog box appears.

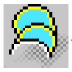
2. Select the offset surface by clicking on the portion that you want to keep after the split.
3. Select the first plane (Plane.2) as cutting element.
4. Click OK to split the surface.



5. Repeat the previous operations by selecting the multi-sections surface then the second plane (Plane.3) to define the intersection first, then to cut the surface.

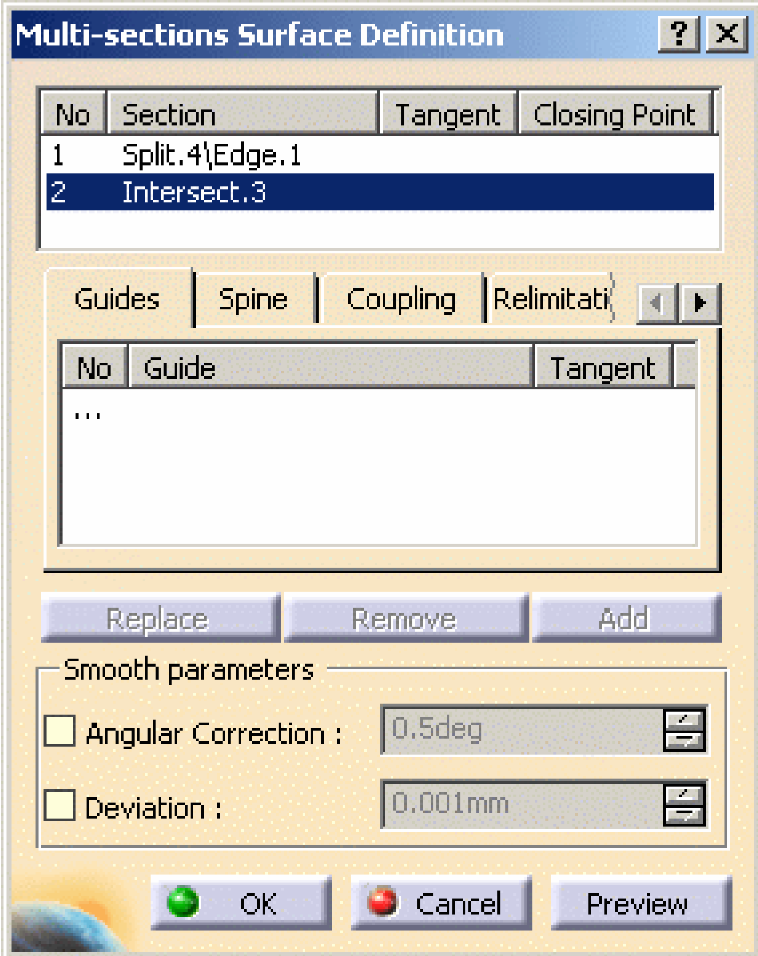
6. Click OK to split the surface.



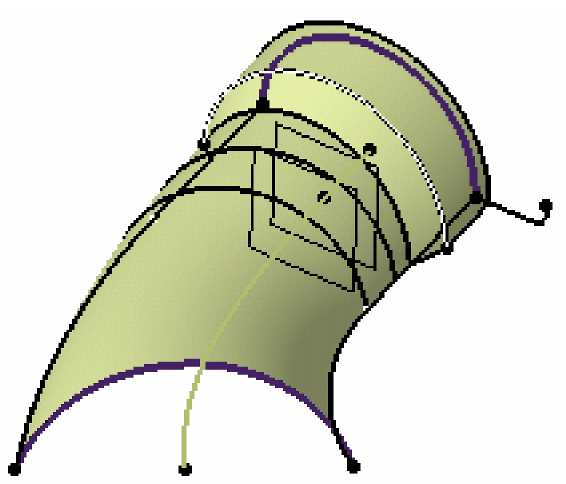
7. Click the **Multi-sections Surface** icon .

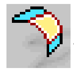
The Multi-sections Surface Definition dialog box appears.

8. Select the intersection edges of the two split surfaces as sections.



10. Click OK to create the multi-sections surface between the two split surfaces.



11. Click the **Shape Fillet** icon .

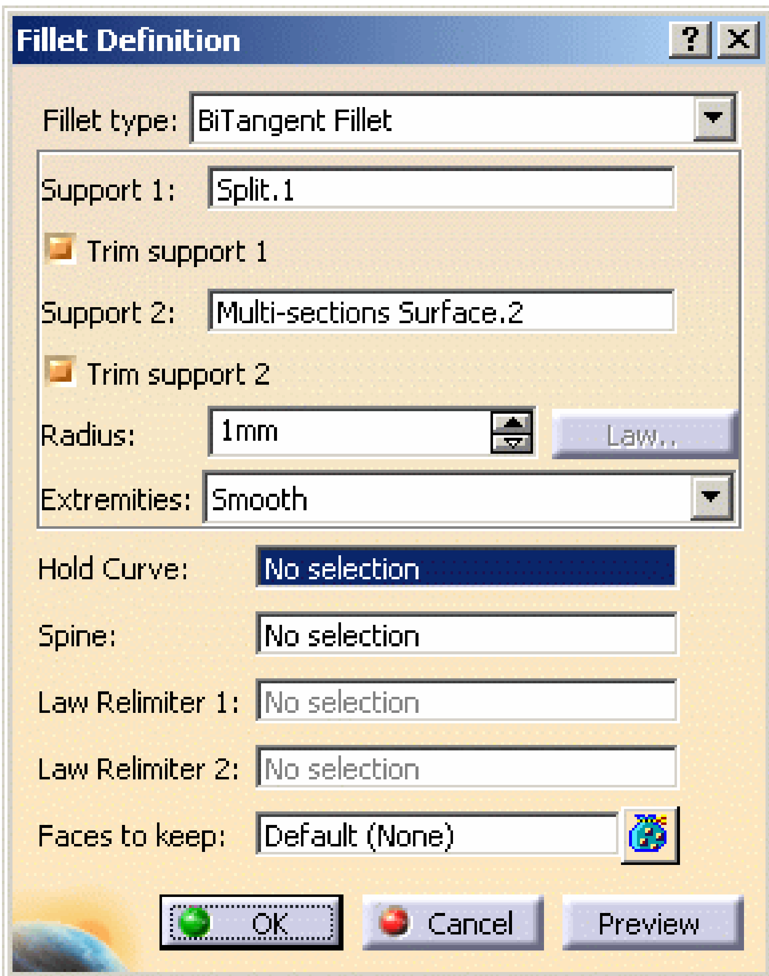
The Fillet Definition dialog box appears.

12. Select the first split surface as the first support element.

13. Select the multi-sections surface you just created as the second support element.

14. Enter a fillet radius of 3mm.

The orientations of the surfaces are shown by means of arrows.



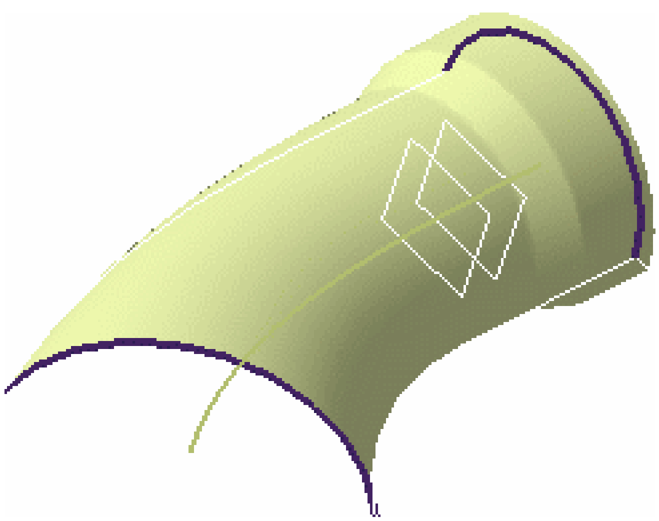
15. Make sure that the surface orientations are correct (arrows pointing down) then click OK to create the first fillet surface.

16. Repeat the filleting operation, clicking the icon, then selecting the second split surface as the first support element.

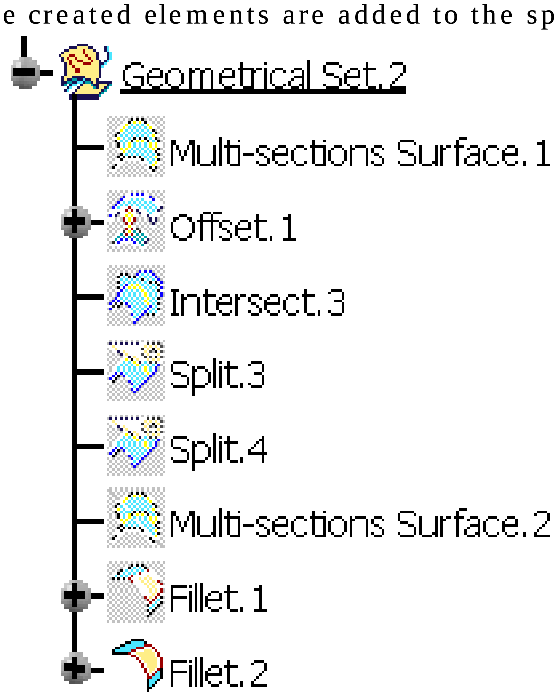
17. Select the previously created filleted surface as the second support element.

18. Enter a fillet radius of 3mm.

19. Make sure that the surface orientations are correct (arrows pointing up) then click OK to create the second filleted surface.



The created elements are added to the specification tree:



Sweeping and Filleting

This task shows how to create swept surfaces and fillets on both sides of the part.

You will use the profile element on the side of the part for this. In this task you will also create a symmetrical profile element on the opposite side of the part.

1. Click the **Sweep** icon .

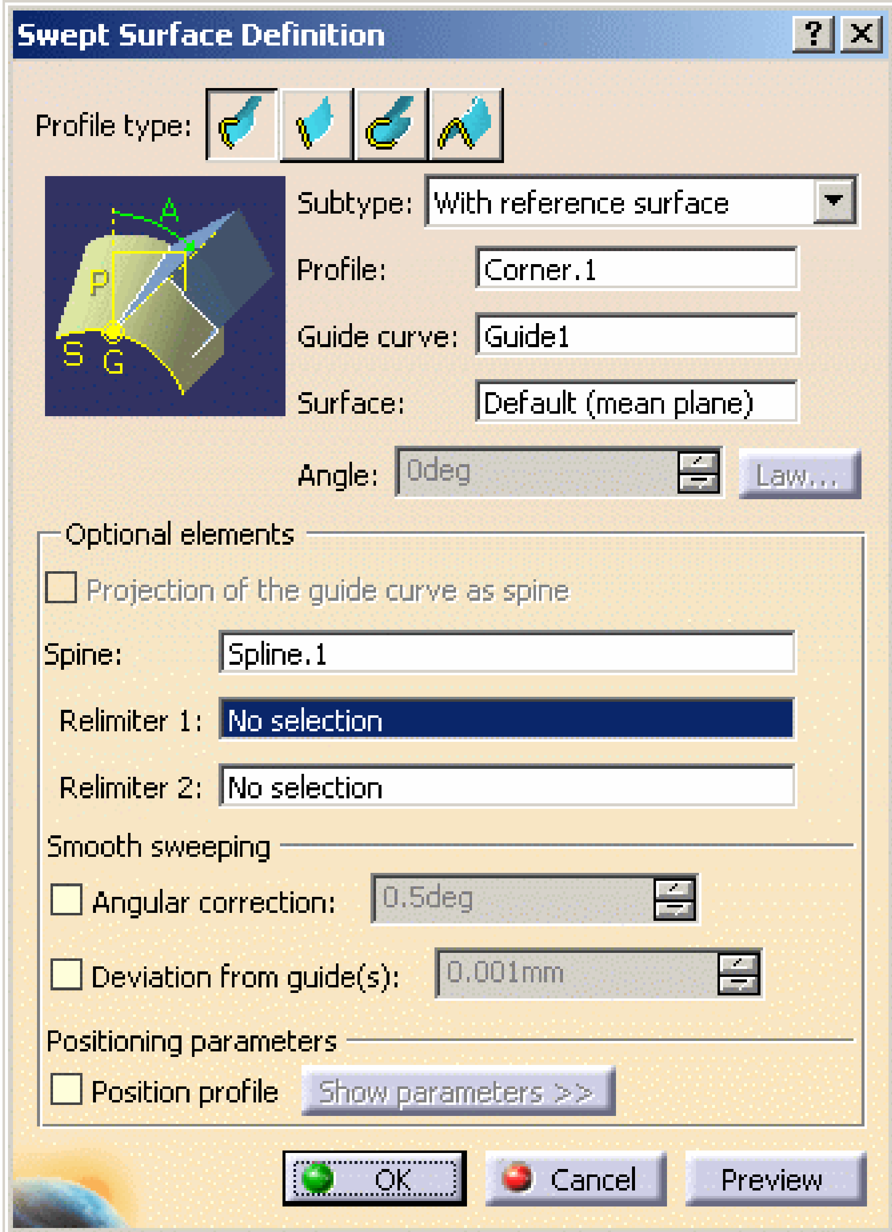
The Swept Surface Definition dialog box appears.

2. Click the **Explicit** sweep icon.

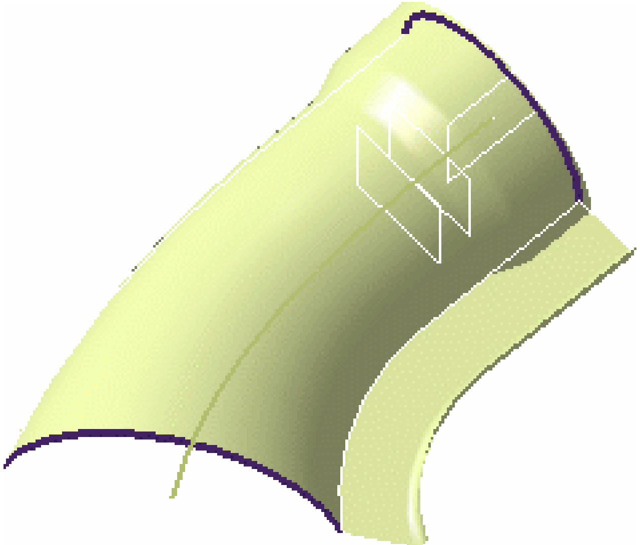
3. Select the profile element (Corner.1).

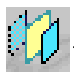
4. Select the guide curve (Guide.1).

5. Select the central curve (Spline.1) as the spine.



6. Click OK to create the swept surface.

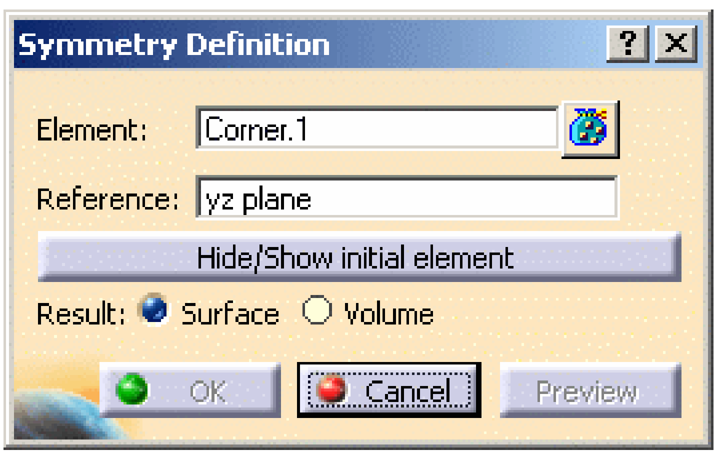


7. Click the **Symmetry** icon .

The Symmetry Definition dialog box appears.

8. Select the profile element to be transformed by symmetry.

9. Select the YZ plane as reference element.



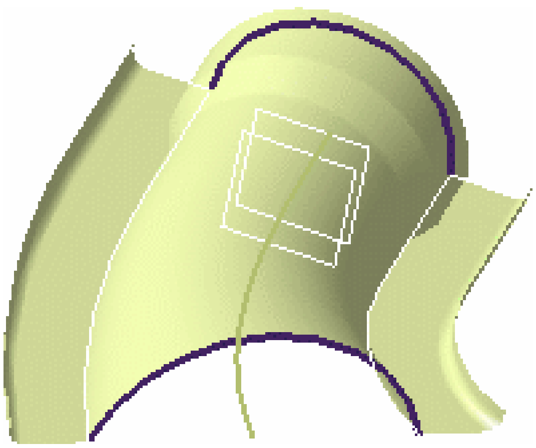
10. Click OK to create the symmetrical profile element.

11. Click the **Sweep** icon  again.

12. Select the profile (Symmetry.3) and the guide curve (Guide.2).

13. Select the central curve (Spline.1) as the spine.

14. Click OK to create the swept surface.



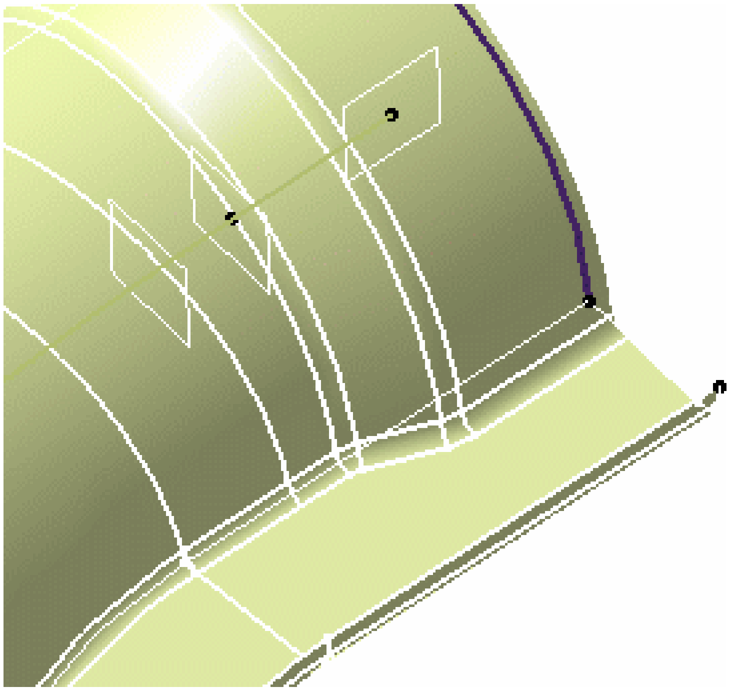
15. To create a fillet between the side portion and the central part click the **Shape Fillet** icon .

16. Select the side sweep element and the central portion of the part, then enter a fillet radius of 1mm (make sure the arrows are pointing up).

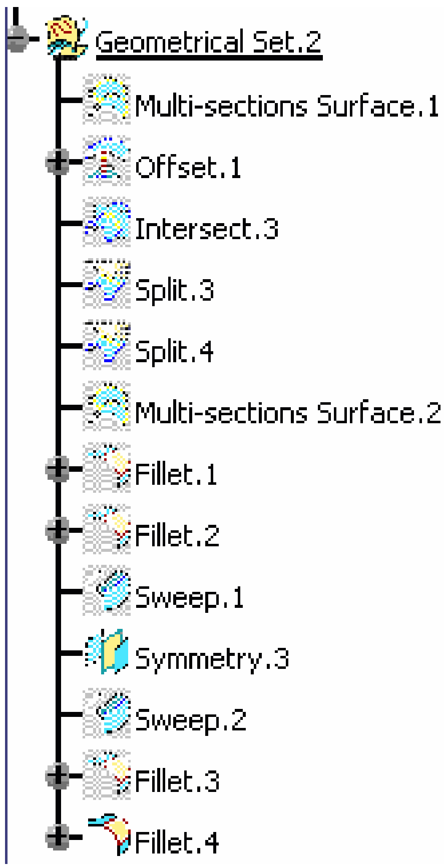
17. Click Preview to preview the fillet, then OK to create it.

18. Repeat the filleting operation between the other sweep element and the central portion of the part, and entering a fillet radius of 1mm (make sure the arrows are pointing up).

19. Click OK to create the fillet.



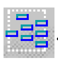
The created elements are added to the specification tree:

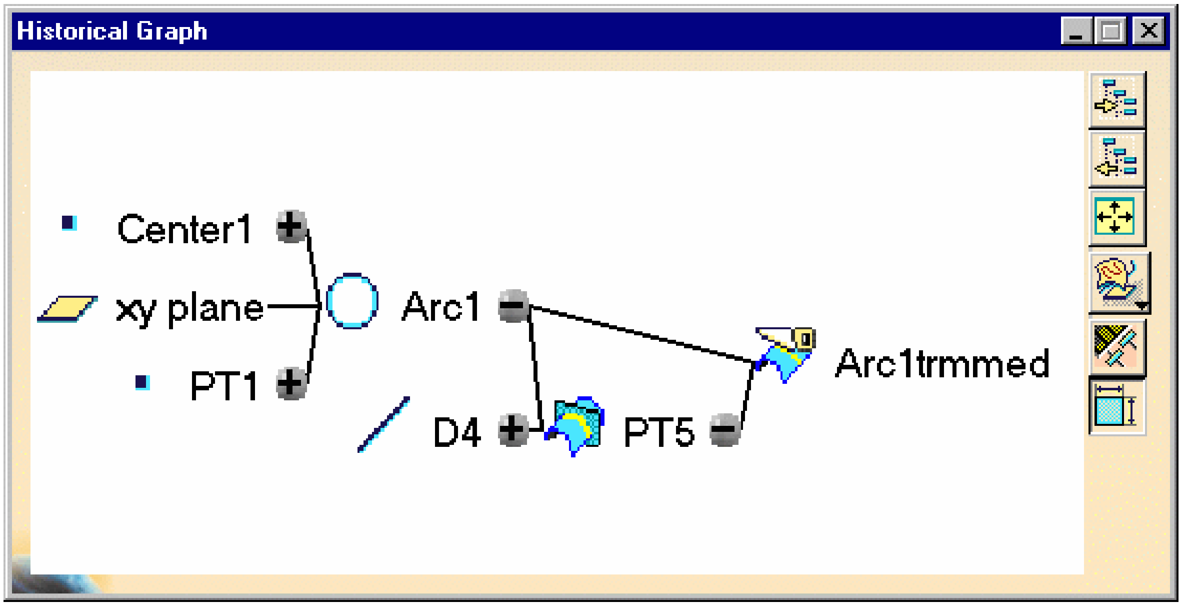


Using the Historical Graph

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

This task shows how to use the historical graph.

1. Select the element for which you want to display the historical graph.
2. Click the **Show Historical Graph** icon .



The Historical Graph dialog box appears.
In this case, you can examine the history of events that led to the construction of the Multi-sections surface.1 element. Each branch of the graph can be expanded or collapsed depending on the level of detail required.

The following icon commands are available.

- Add graph
- Remove graph
- Reframe graph
- Surface or Part representation
- Parameters filter
- Constraints filter

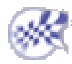
Right-clicking anywhere in the historical graph enables the user to:

- hide or show an element
- display the properties of an element
- reframe the graph
- display the whole graph of the part (as well as the roots and their first parents)
- clean the graph
- refresh the graph

- display the parents of the elements in the graph (but not the roots)

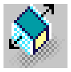
Selecting and right-clicking an element enables the user to add the children to the selected element.

3. Just click the Close icon to exit this mode.

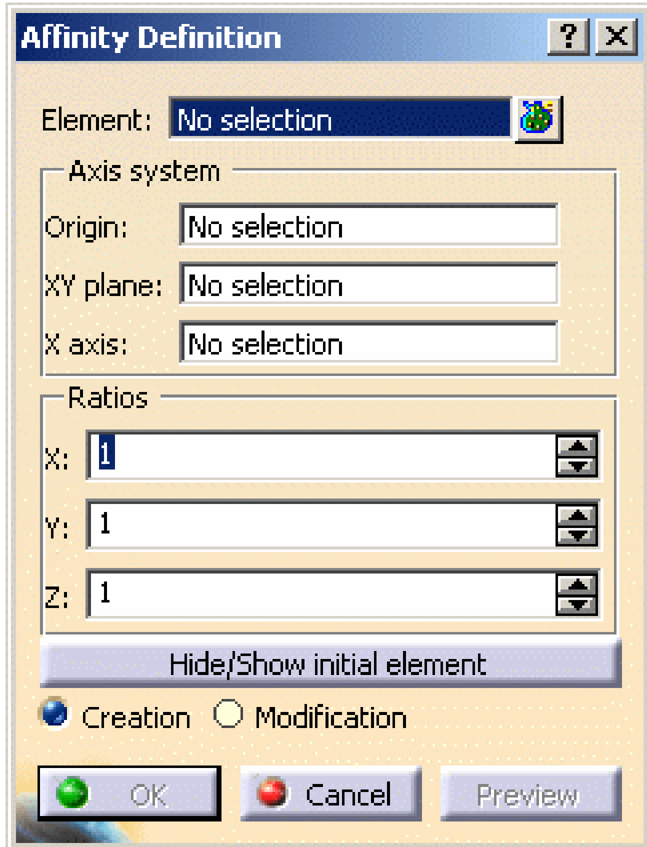


Transforming the Part

This task shows you how to modify the part by applying an affinity operation.

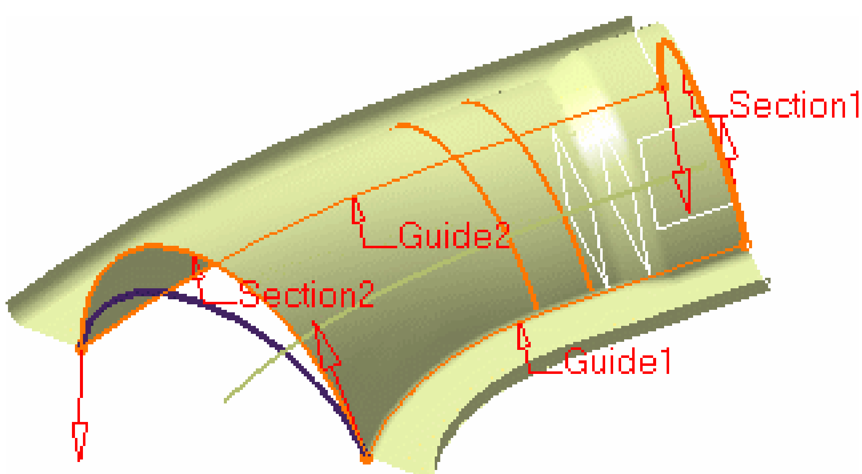
1. Click the **Affinity** icon .

   The Affinity Definition dialog box appears.



2. Select the end section profile to be transformed by the affinity.
3. Specify the characteristics of the axis system to be used for the affinity operation:

- point PT0 as the origin
- plane XY as reference plane
- horizontal edge of the corner profile as x-axis.



4. Specify the affinity ratios:

   X=1, Y=1 and Z=1.5.

5. Click OK to create the new profile.

6. Edit the definition of the lofted surface (Multi-sections Surface.1), by double-clicking it, then select the second section, click the Replace button and select the new profile.

7. Click OK in the dialog box.

8. If needed, click the **Update** icon to update your design.

Multi-selection is available. Refer to Selecting Using Multi-Mutput to find out how to display and manage the list of selected elements.

Basic Tasks

The basic tasks you will perform in the Generative Shape Design workbench will involve creating and modifying wireframe and surface geometry that you will use in your part.

The table below lists the information you will find in this section.

Creating Wireframe Geometry
Creating Surfaces
Performing Operations on Shape Geometry
Editing Surfaces and Wireframe Geometry
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.

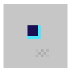
You should avoid selecting self-intersecting sketches as well as sketches containing heterogeneous elements such as a curve and a point for example.

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 geometry
- Trim geometry
- All transformations: translation, rotation, symmetry, scaling, affinity and axis to axis
- Developed wires (Developed Shapes)

Creating Wireframe Geometry

Generative Shape Design allows you to create wireframe geometry such as points, lines, planes and curves. You can make use of this elementary geometry when you create more complex surfaces later on.

 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.

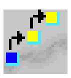
Create points on a plane: select a plane and possibly a reference point, then click the plane.

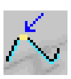
Create points on a surface: select a surface and possibly a reference point, an element to set the projection orientation, and a length.

Create points as a circle center: select a circle.

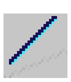
Create points at tangents: select a curve and a line.

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 extrema: select a curve and a direction into which the extremum point is detected.

 Create polar extrema: select a contour and its support, a computation mode, and a reference axis-system (origin and direction) .

 Create lines between two points: select two points.

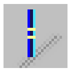
Create lines based on a point and a direction: select a point and a line, then specify the start and end points of the line.

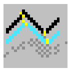
Create lines at an angle or normal to a curve: select a curve and its support, a point on the curve, then specify the angle value, the start and end points of the line.

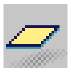
Create lines tangent to a curve: select a curve and a reference point, then specify the start and end points of the line.

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

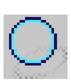
Create a plane normal to a curve: select any curve and a point

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 conics: select a support plane, start and end points, and any other three constraints (intermediate points or tangents).

Create spirals: select a support plane, center point, and reference direction, then set the radius, angle, and pitch as needed.

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 point and a direction, and specify the helix pitch, height, orientation and taper angle.

Create a spine: select several planes or planar curves to which the spine is normal

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 parallel curves: select the reference curve, a support plane or surface, and specify the offset value from the reference.

Create a 3D Curve Offset: select the reference curve, a direction and specify the offset value from the reference.

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

Create combined curves: select the curves, possibly directions, and specify the combine type.

Create reflect lines: select the support and direction, and specify an angle.

Create intersections: select the two elements to be intersected.

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/sphere center
- tangent point on a curve
- between

Open the Points3D1.CATPart document.

1. Click the **Point** icon .

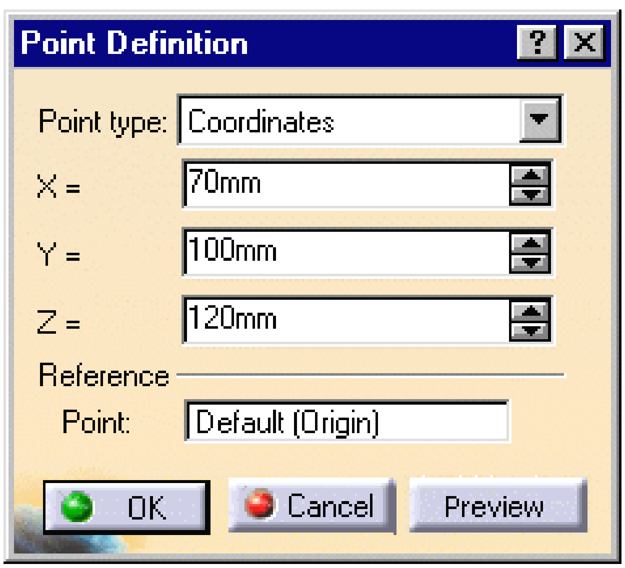
The Point Definition dialog box appears.

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.

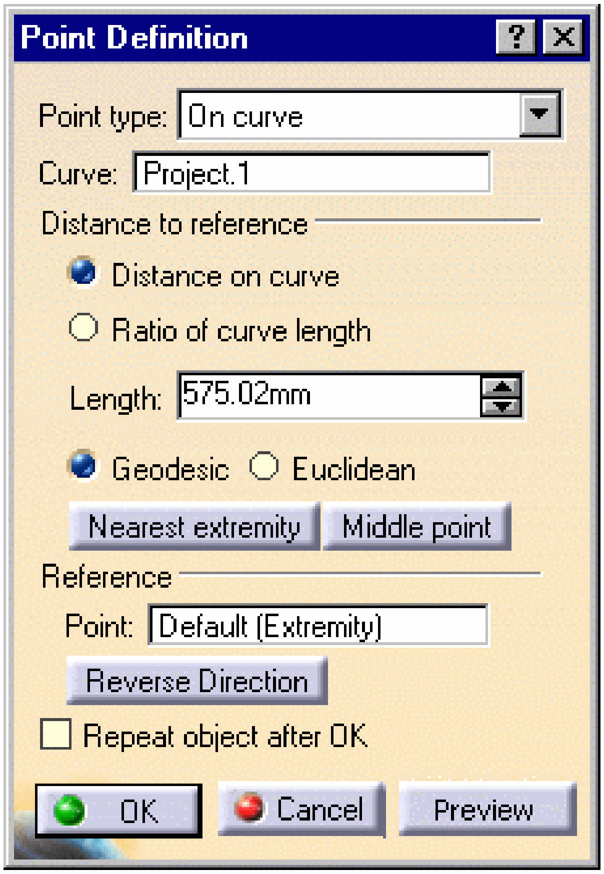


When creating a point within a user-defined axis-system, note that the **Coordinates in absolute axis-system** check button is added to the dialog box, allowing you to be define, or simply find out, the point's coordinates within the document's default axis-system.
If you create a point using the coordinates 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.

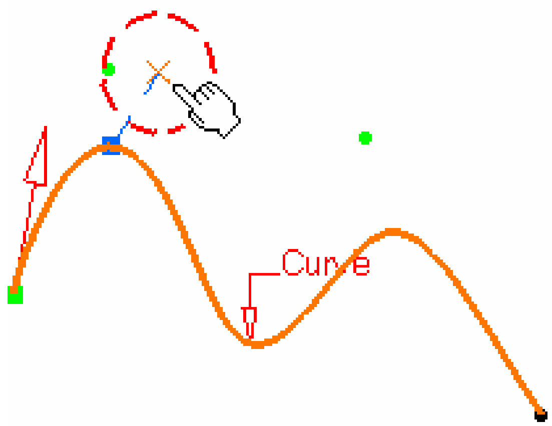
On curve

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

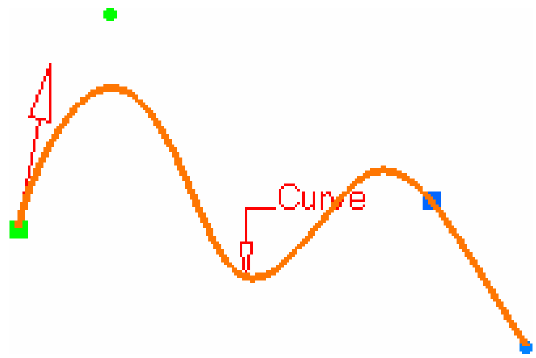


If this point is not on the curve, it is projected onto the curve.
If no point is selected, the curve's extremity is used as reference.

- 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.

If the reference point is located at the curve's extremity, even if a ratio value is defined, the created point is always located at the end point of the curve.

You can also:

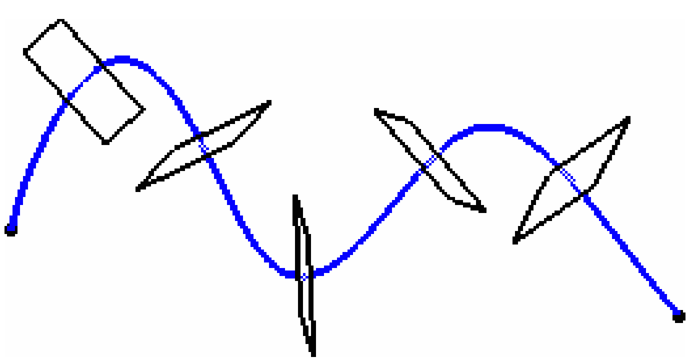
- click the **Nearest extremity** button to display the point at the nearest extremity of the curve.
- click the **Middle Point** button to display the mid-point of the curve.

Be careful that the arrow is orientated towards the inside of the curve (providing the curve is not closed) when using the **Middle Point** option.

- 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.

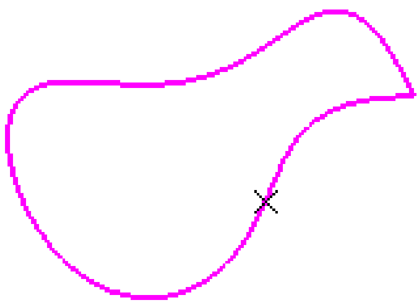
You will also be able to create planes normal to the curve at these points, by checking the **Create normal planes also** button, and to create all instances in a new geometrical set by checking the

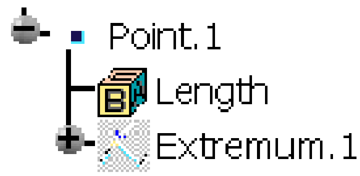


Create in a new geometrical set button. If the button is not checked the instances are created in the current geometrical set .

- 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.

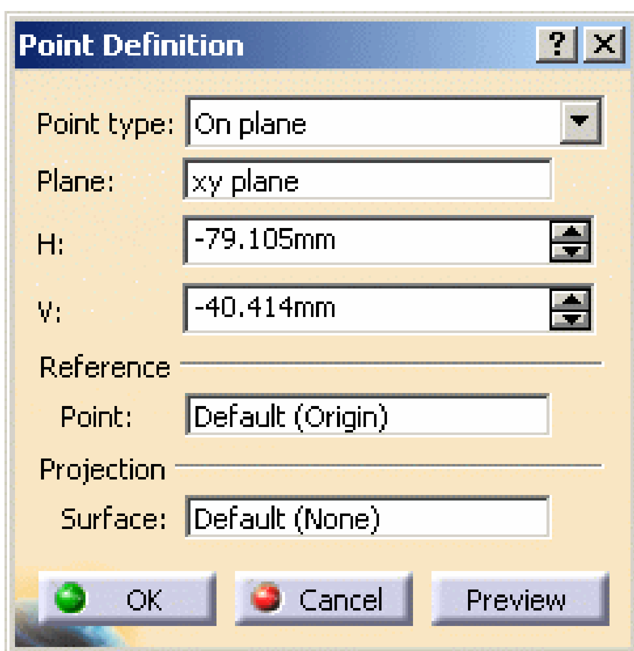




On plane

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

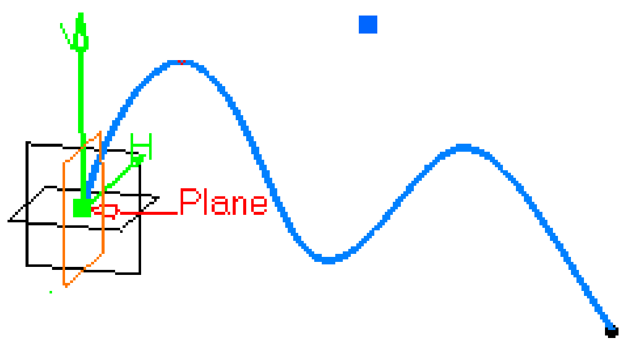
If no point is selected, the projection of the model's origin on the plane is taken as reference.



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

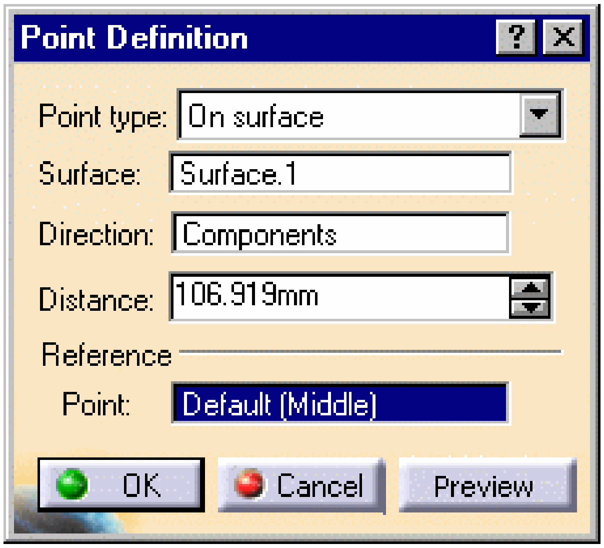
If no surface is selected, the behavior is the same.

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.



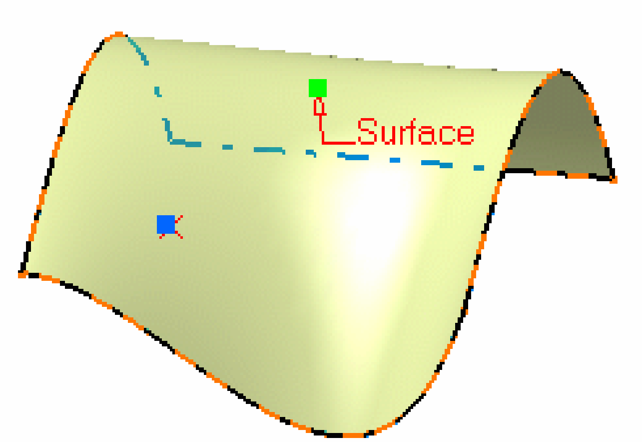
Would the plane move, during an update for example, the reference direction would then be projected on the plane.

- Click in the plane to display a point.



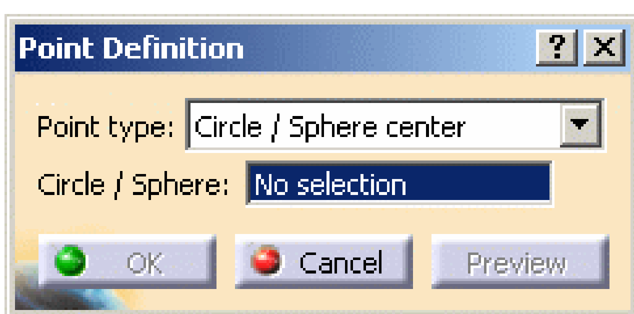
On surface

- 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.



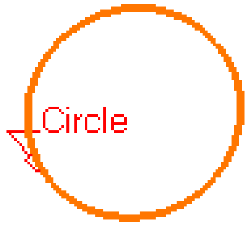
- Enter a distance along the reference direction to display a point.

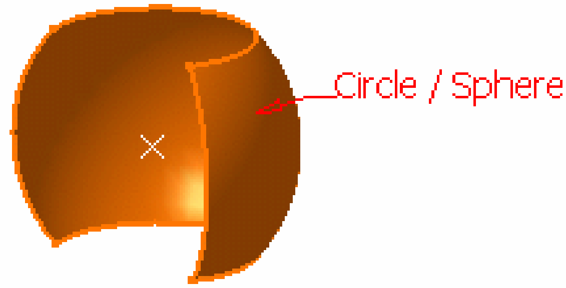
Circle/Sphere center



- Select a circle, circular arc, or ellipse, or
- Select a sphere or a portion of sphere.

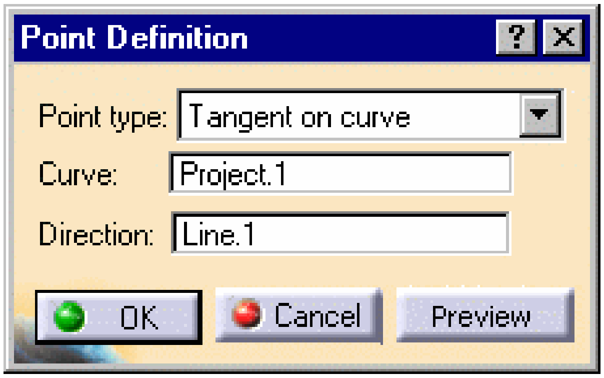
A point is displayed at the center of the selected element.





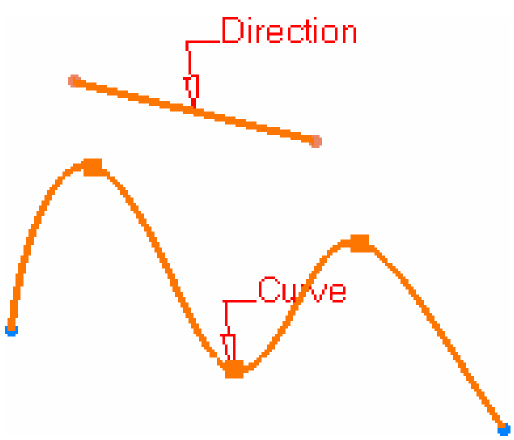
Tangent on curve

- Select a planar curve and a direction line.



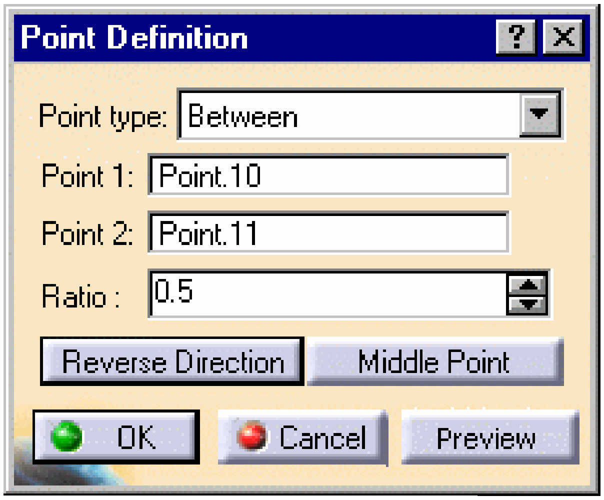
A point is displayed at each tangent.

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



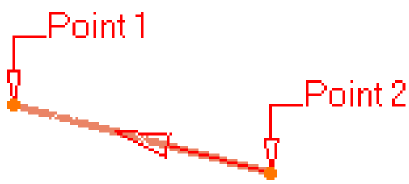
- Click **YES**: you can then select a reference element, to which only the closest point is created.
- Click **NO**: all the points are created.

For further information, refer to the Managing Multi-Result Operations chapter.



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).

Be careful that the arrow is orientated towards the inside of the curve (providing the curve is not closed) when using the **Middle Point** option.

- 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.

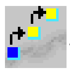
The point (identified as Point.xxx) is added to the specification tree.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a point in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

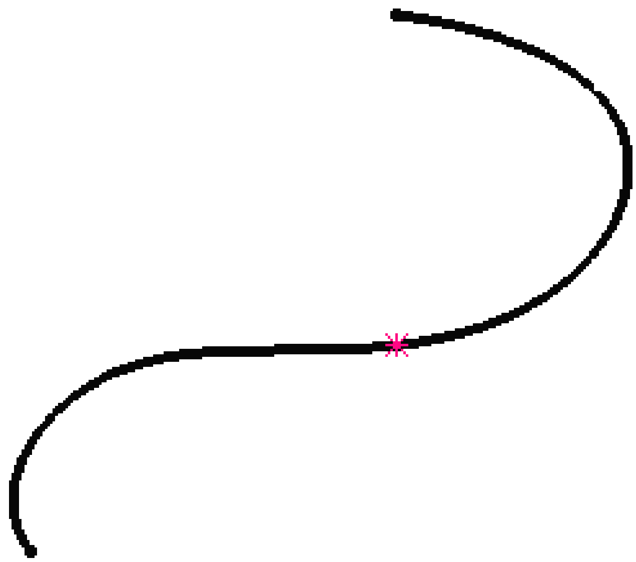
Creating Multiple Points and Planes

This task shows how to create several points, and planes, at a time:

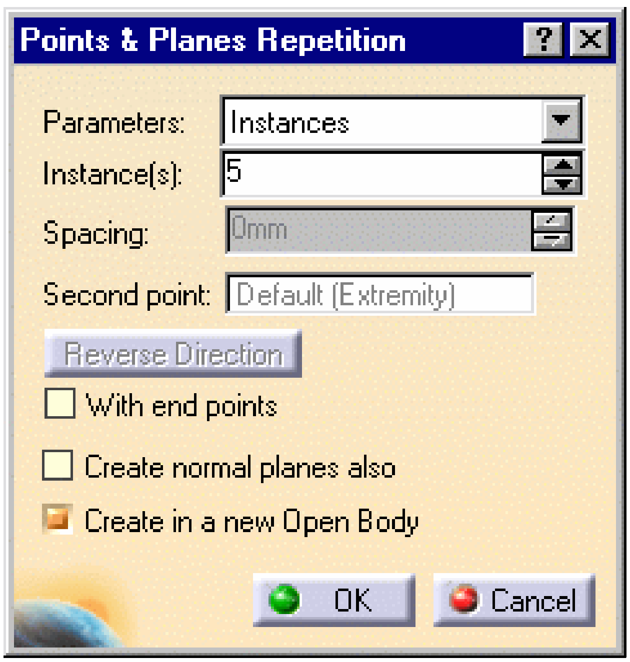
Open the MultiplePoints1.CATPart document.
Display the Points toolbar by clicking and holding the arrow from the **Point** icon.

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

2. Select a curve or a Point on curve.



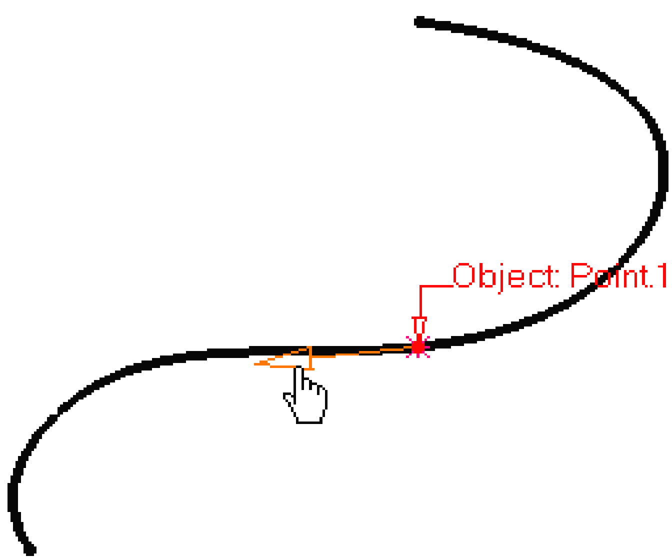
The Points Creation Repetition dialog box appears.



3. Define the number or points to be created (instances field).

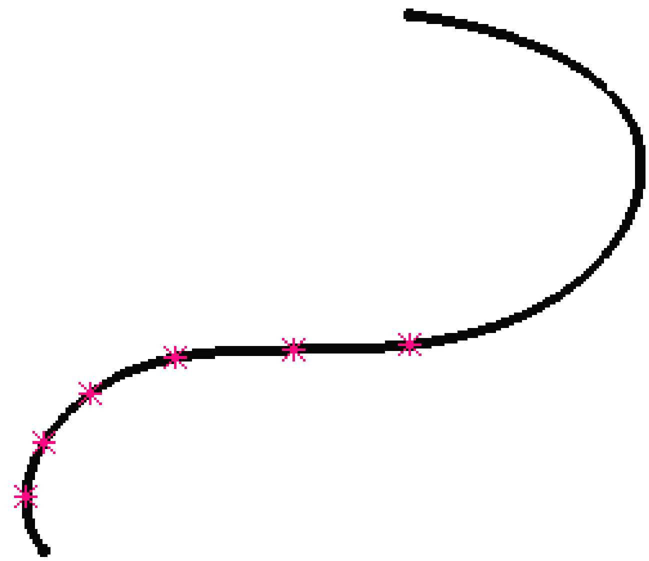
   Here we chose 5 instances.

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.



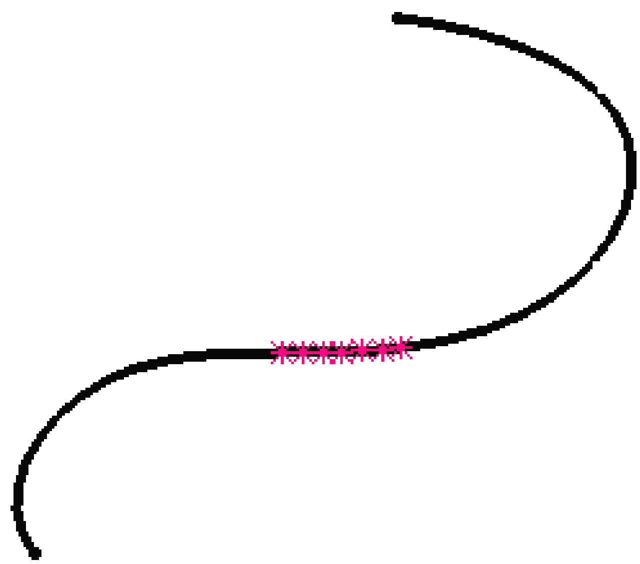
If you check the **With end points** option, the last and first instances are the curve end points.

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

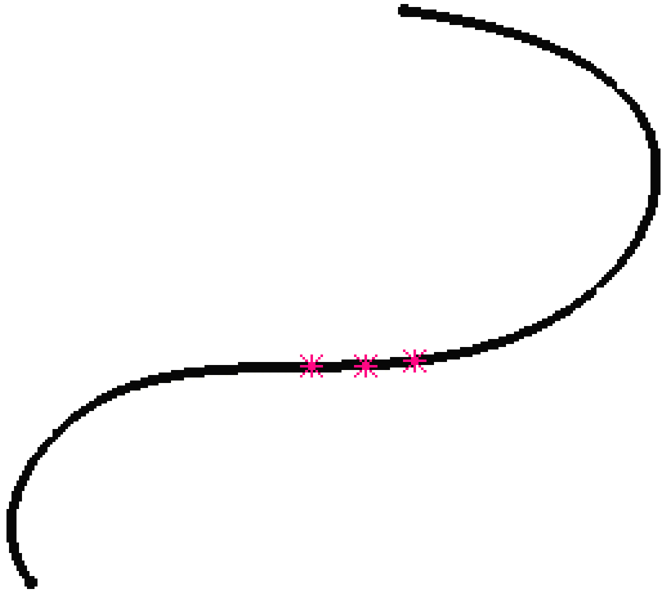


The points (identified as Point.xxx as for any other type of point) are added to the specification tree.

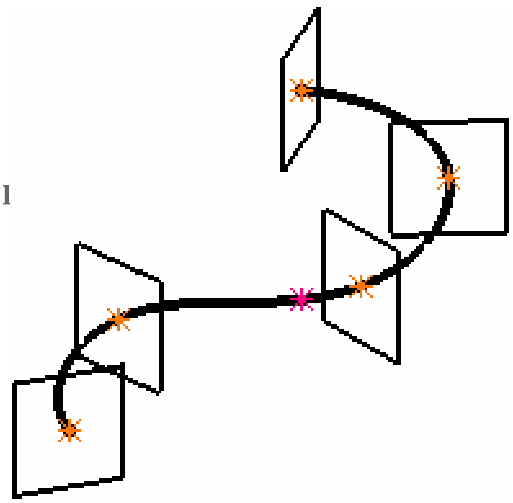
- If you selected a point on a curve, you can select a second point, thus defining the area of the curve where points should be created.
  Simply click the **Second point**field in the Multiple Points Creation dialog box, then select the limiting point.
  If you selected the Point2 created above as the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



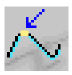
- Check the **Create normal planes also** to automatically generate planes at the point instances.
- Check the **Create in a new geometrical set** if you want all object instances in a separate Geometrical Set.
  A new Geometrical Set will be created automatically.
  If the option is not checked the instances are created in the current Geometrical Set.

Creating Extremum Elements

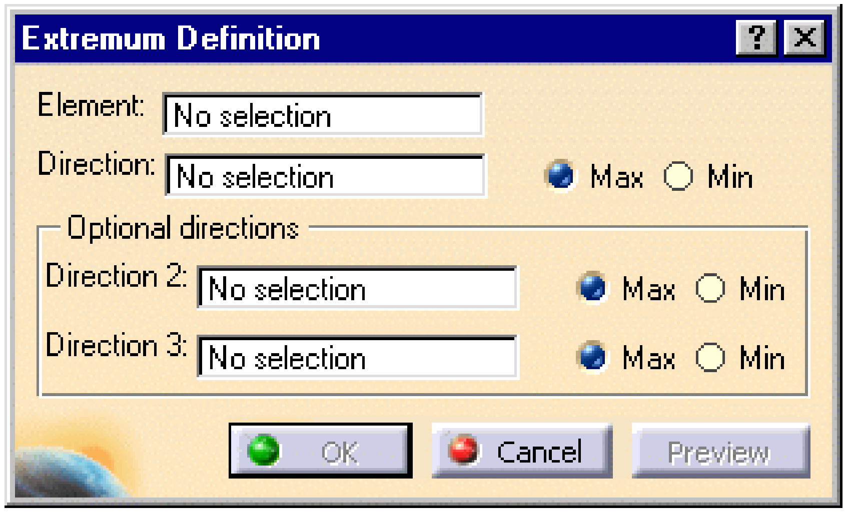
This command is only available with the Generative Shape Design 2 product.

This task shows you to create extremum elements (points, edges, or faces), that is elements at the minimum or maximum distance on a curve, a surface, or a pad, according to given directions.

Open the Extremum1.CATPart document.
Display the **Points** toolbar by clicking and holding the arrow from the **Point** icon.

1. Click the **Extremum** icon .

   The Extremum Definition dialog box is displayed.

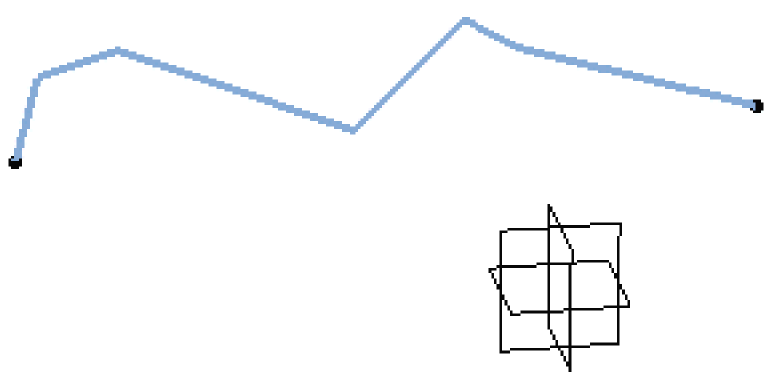


2. Set the correct options:

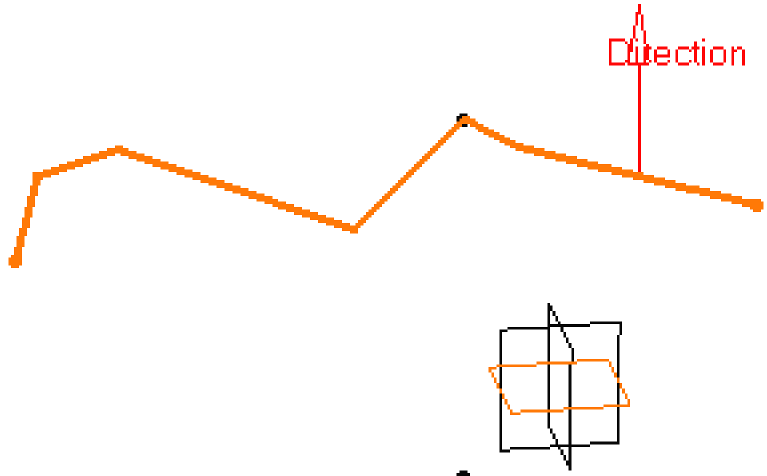
- **Max**: according to a given direction the highest point on the curve is created
- **Min**: according to the same direction the lowest point on the curve is created

Extremum Points on a curve:

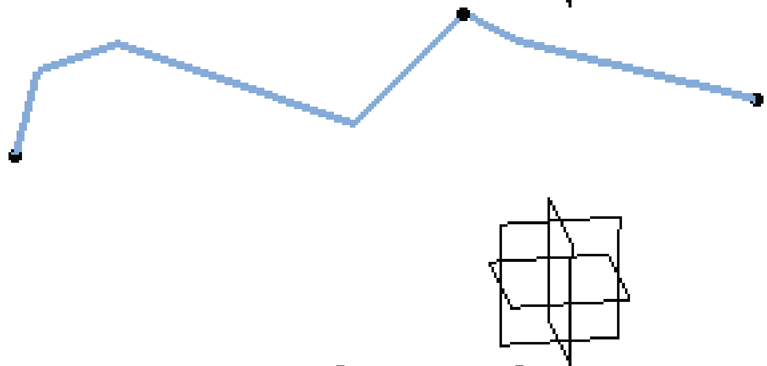
3. Select a curve.



4. Select the direction into which the extremum point must be identified.



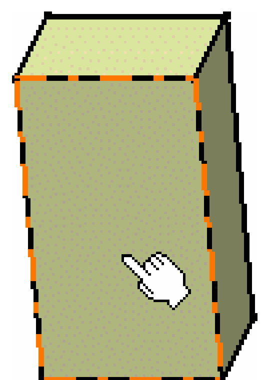
5. Click OK.



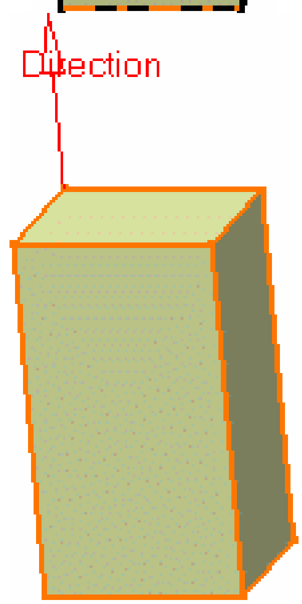
The point (identified as Extremum.xxx) is added to the specification tree.

Extremum on a surface:

3. Select a surface.



4. Select the direction into which the extremum must be identified.

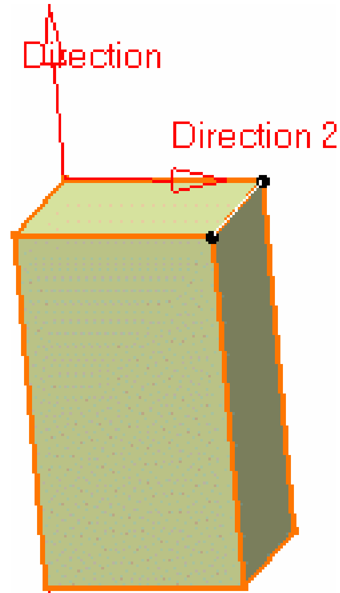


If you click OK, the extremum face is created.

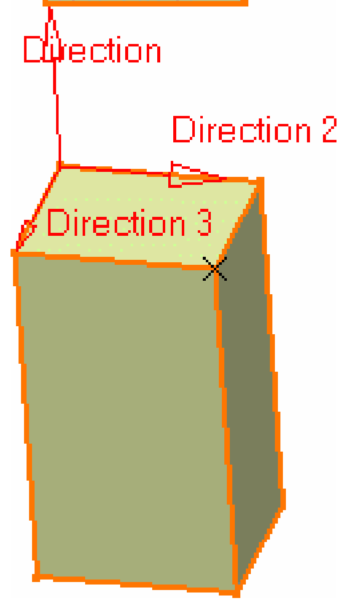
Giving only one direction is not always enough. You need to give a second, and possibly a third direction depending on the expected result (face, edge or point) to indicate to the system in which direction you want to create the extremum element. These directions must not be identical.

3. Select a second direction.

If you click OK, the extremum edge is created.



4. Select a third direction.



5. Click OK.

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

Creating Polar Extremum Elements

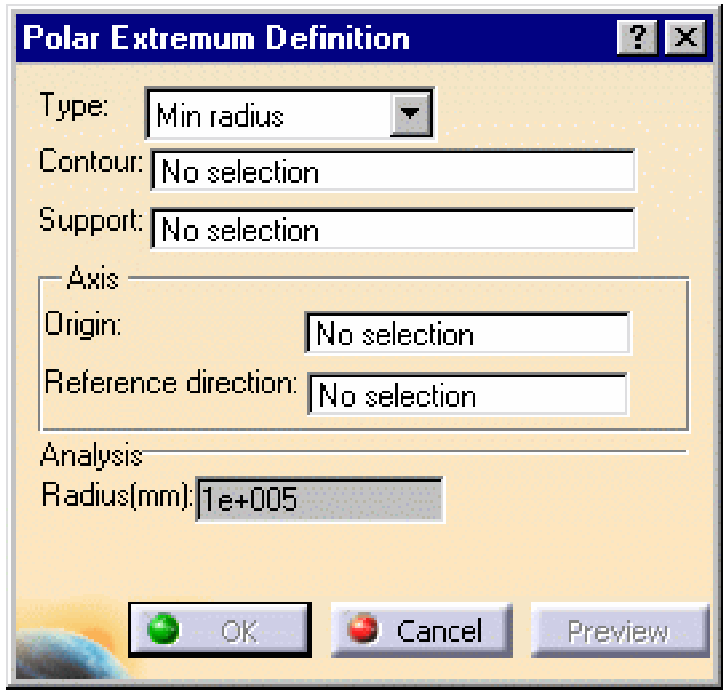
This command is only available with the Generative Shape Design 2 product.

This task shows how to create an element of extremum radius or angle, on a planar contour.

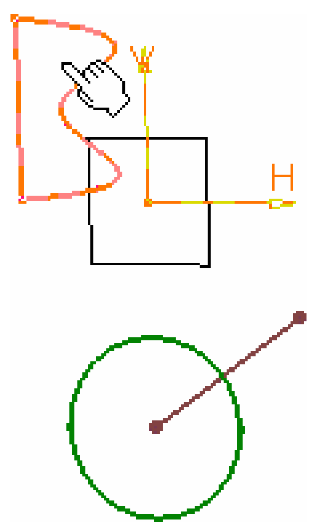
Open the Extremum2.CATPart document.

1. Click the **Polar Extremum** icon .

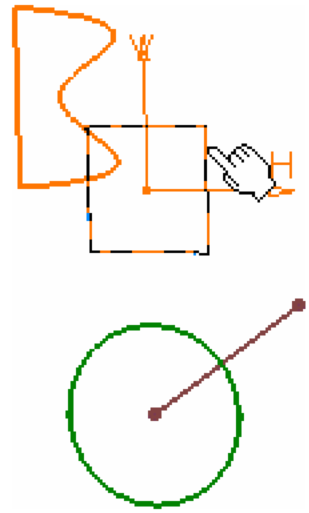
The Polar Extremum Definition dialog box appears.



2. Select the contour, that is a connex planar sketch or curve on which the extremum element is to be created.



Non connex elements, such as the letter A in the sample, are not allowed.



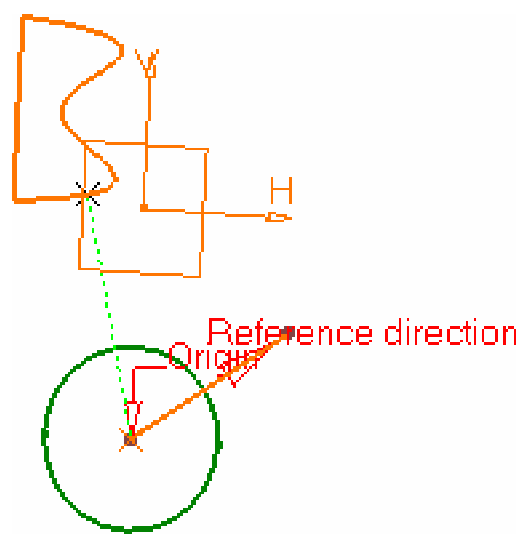
3. Select the supporting surface of the contour.

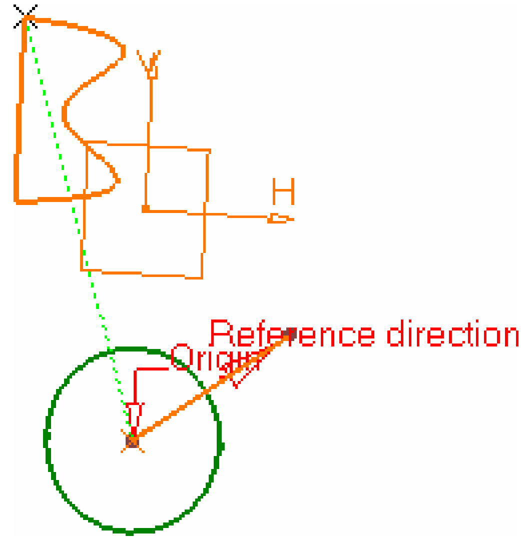
4. Specify the axis origin and a reference direction, in order to determine the axis system in which the extremum element is to be created.

5. Click Preview:

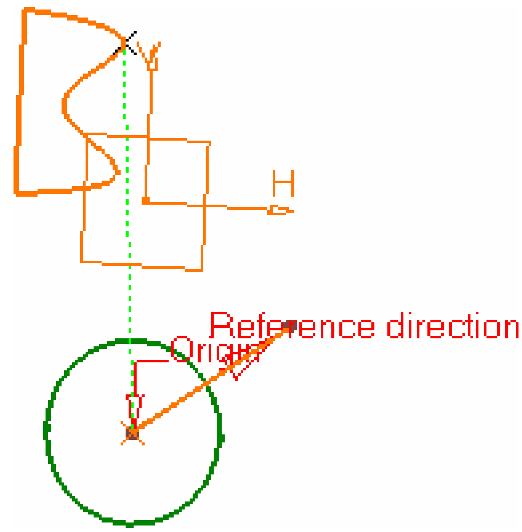
Depending on the selected computation type, the results can be:

- **Min radius**: the extremum element is detected based on the shortest distance from the axis-system origin

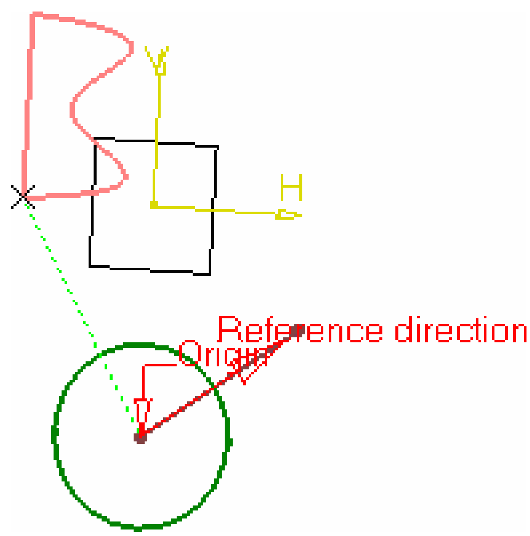




- **Max radius**: the extremum element is detected based on the longest distance from the axis-system origin



- **Min angle**: the extremum element is detected based on the smallest angle from the selected direction within the axis-system



- **Max angle**: the extremum element is detected based on the greatest angle from the selected direction within the axis-system

The radius or angle value is displayed in the Polar Extremum Definition dialog box for information.

6. Click **OK** to create the extremum point.

The element (identified as Polar extremum.xxx), a point in this case, is added to the specification tree.

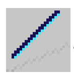
Creating Lines

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

It also shows you how to create a line up to an element, define the length type and automatically reselect the second point.

Open the Lines1.CATPart document.

1. Click the **Line** icon .

   The Line Definition dialog box is displayed.

2. Use the drop-down list to choose the desired line type.

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

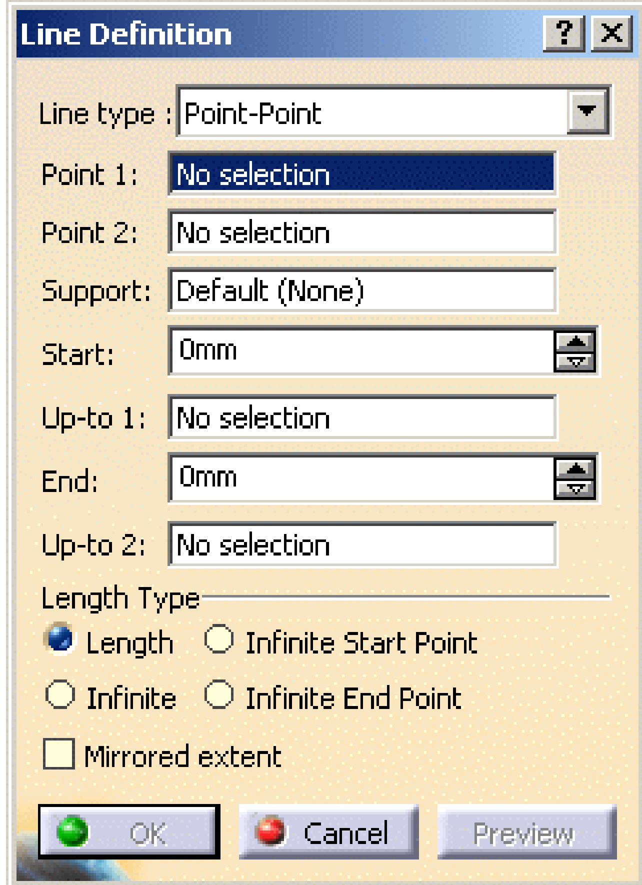
Defining the line type

Point - Point

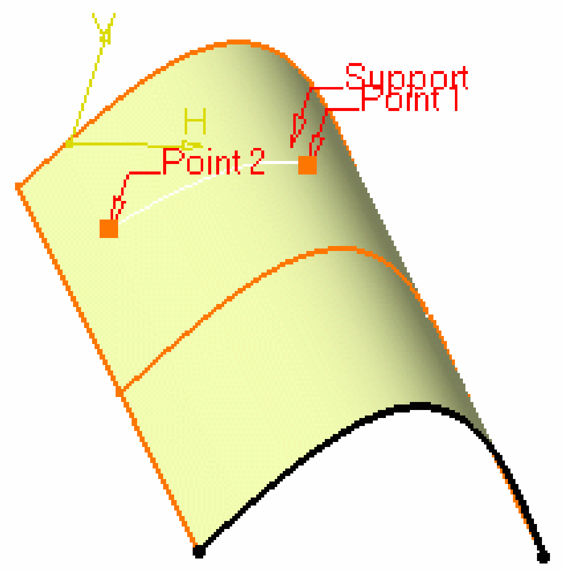
This command is only available with the Generative Shape Design 2 product.

- Select two points.

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

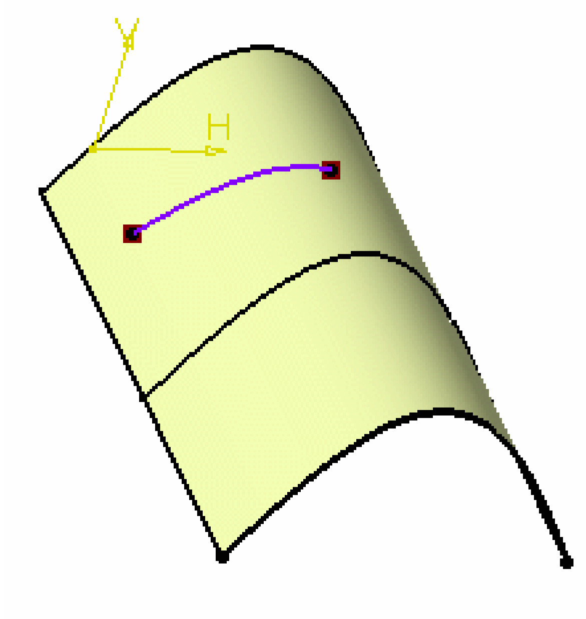


- If needed, select a support surface.
  In this case a geodesic line is created, i.e. going from one point to the other according to the shortest distance along the surface geometry (blue line in the illustration below).
  If no surface is selected, the line is created between the two points based on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



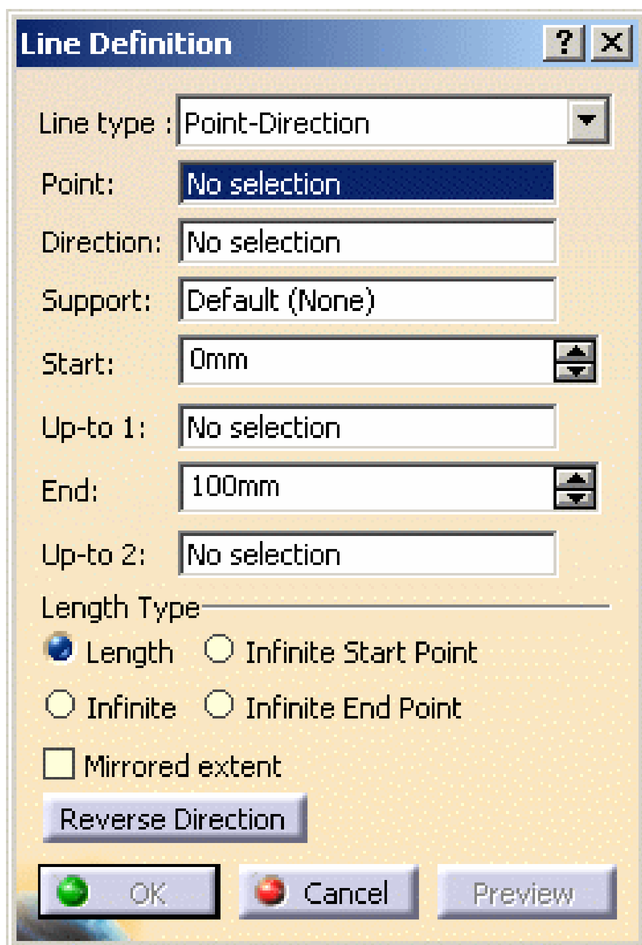
If you select two points on closed surface (a cylinder for example), the result may be unstable. Therefore, it is advised to split the surface and only keep the part on which the geodesic line will lie.

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.

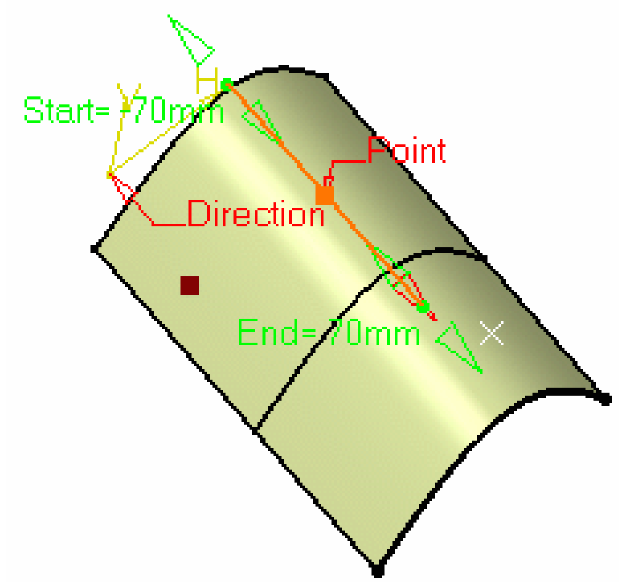
The projections of the 3D point(s) must already exist on the selected support.

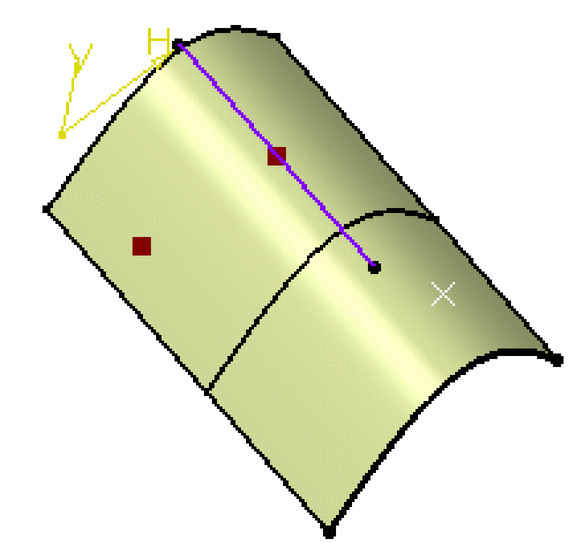


Point - Direction

- 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.





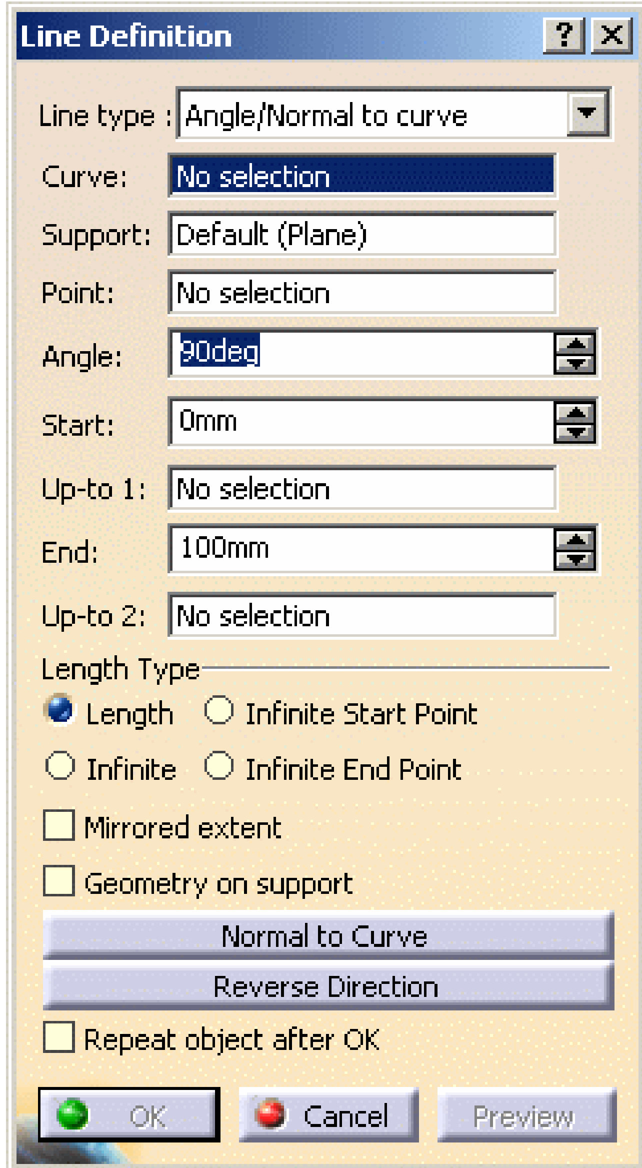
The projections of the 3D point(s) must already exist on the selected support.

Angle or Normal to curve

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

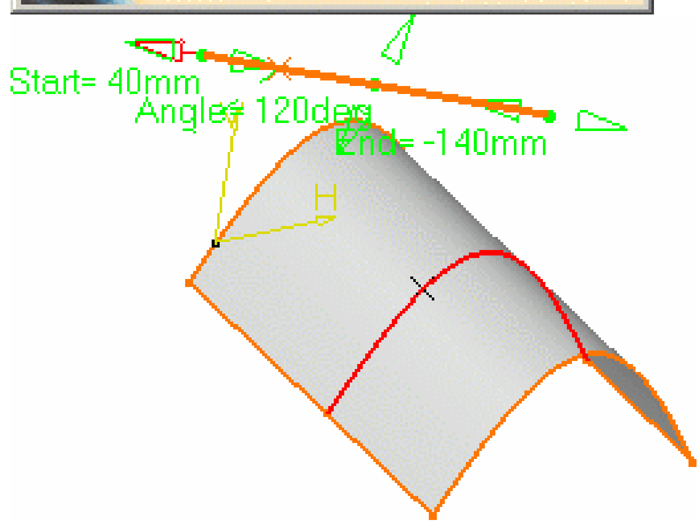
- If the selected curve is planar, then the **Support** is set to Default (Plane).

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



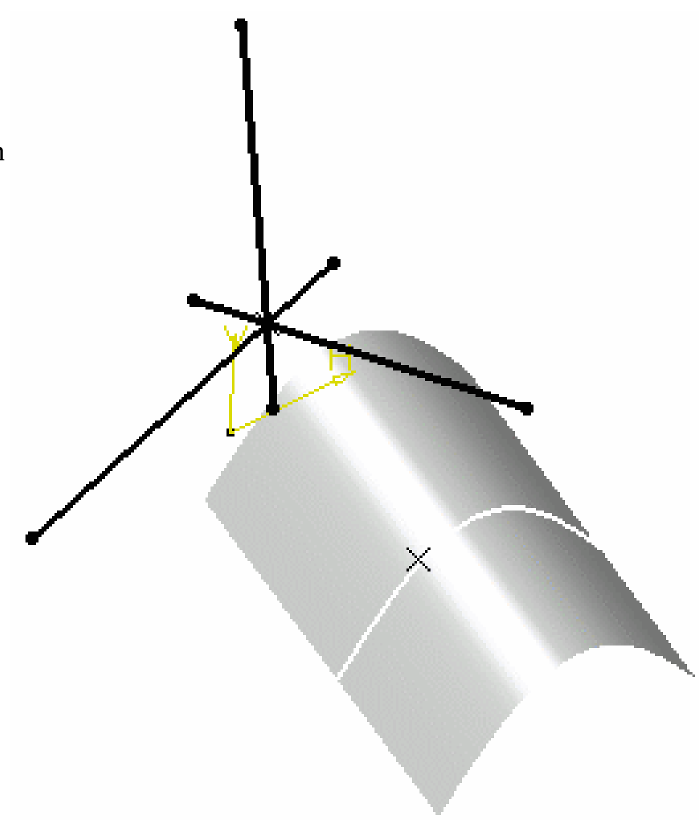
- Select a **Point** on the curve.
- Enter an **Angle** value.

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.

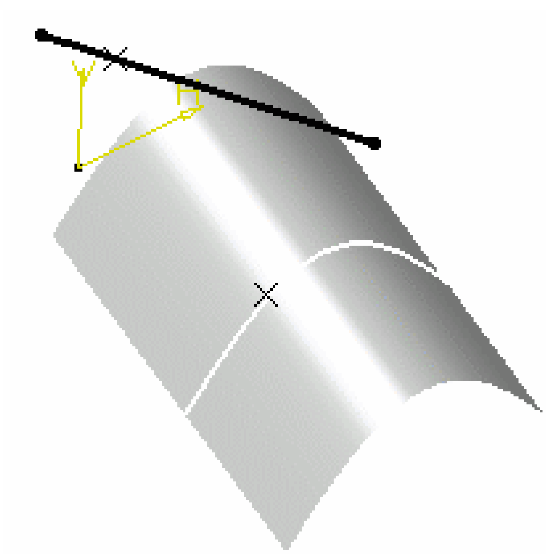
- Click the **Repeat object after OK** if you wish to create more lines with the same definition as the currently created line.
  In this case, the Object Repetition dialog box is displayed, and you key in the number of instances to be created before pressing OK.

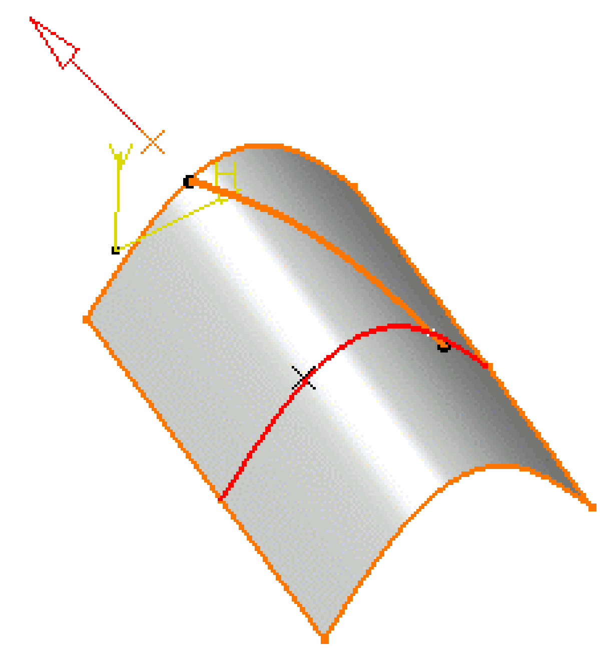


As many lines as indicated in the dialog box are created, each separated from the initial line by a multiple of the **angle** value.

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.

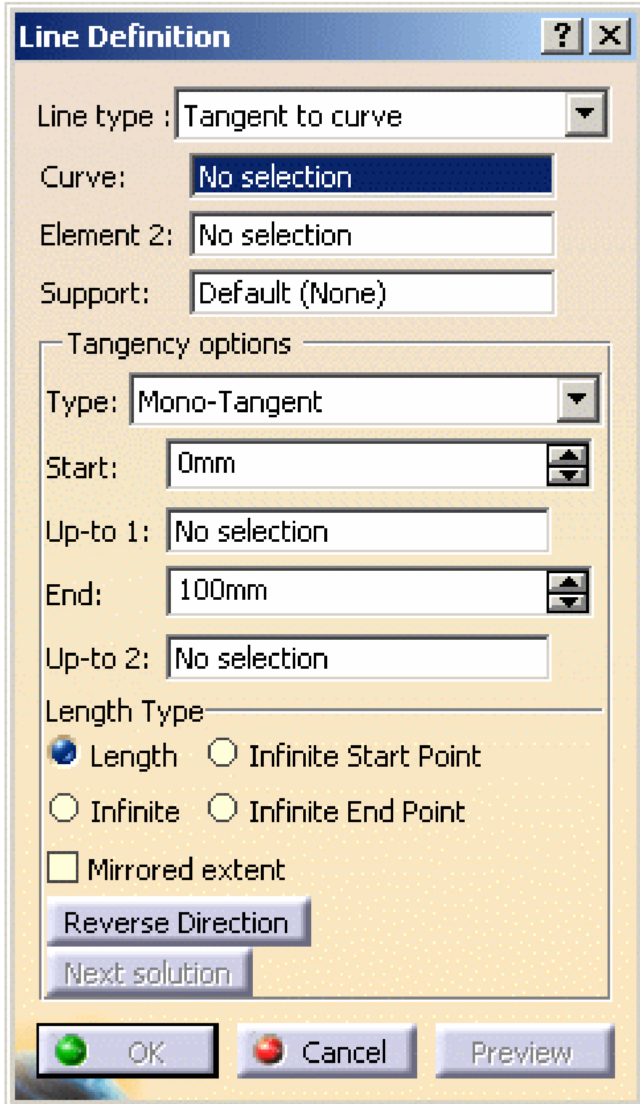
Tangent to curve

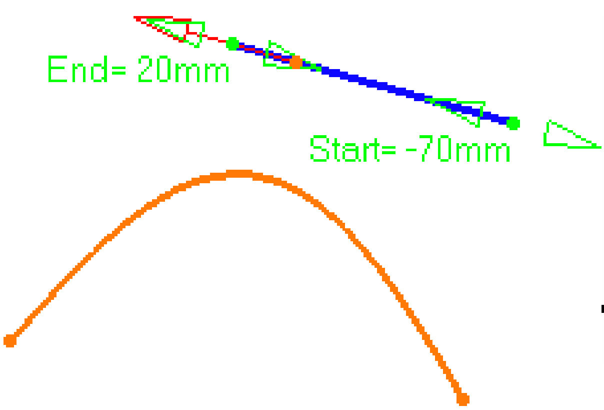
- Select a reference **Curve** and a **point** or another **Curve** to define the tangency.
    - if a point is selected (mono-tangent mode): a vector tangent to the curve is displayed at the selected point.
    - If a second curve is selected (or a point in bi-tangent mode), you need to select a support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

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

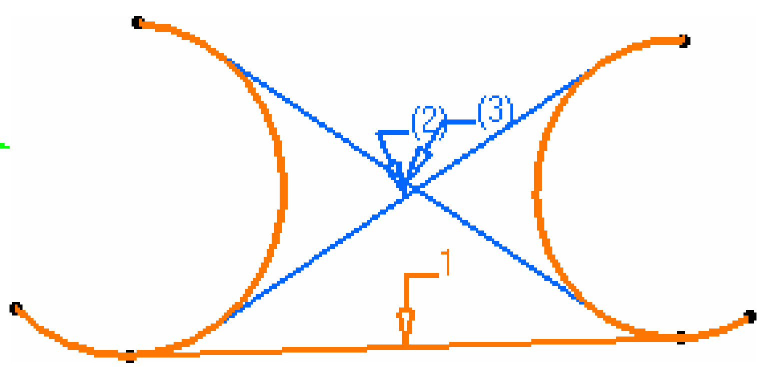
- If an explicit **Support** has been defined, a contextual menu is available to clear the selection.

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



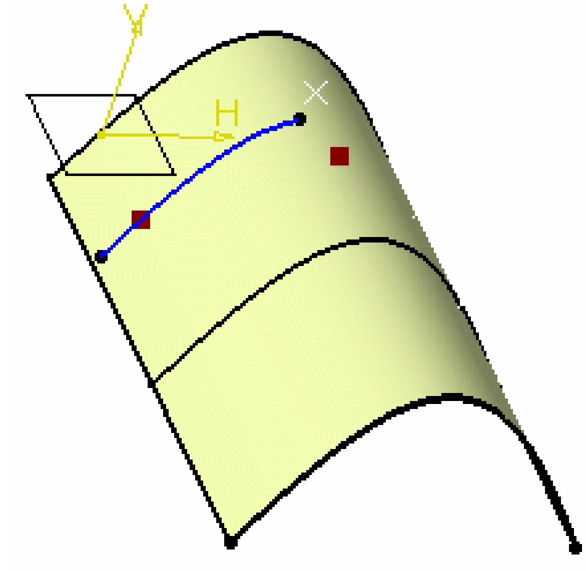


Line tangent to curve at a given point



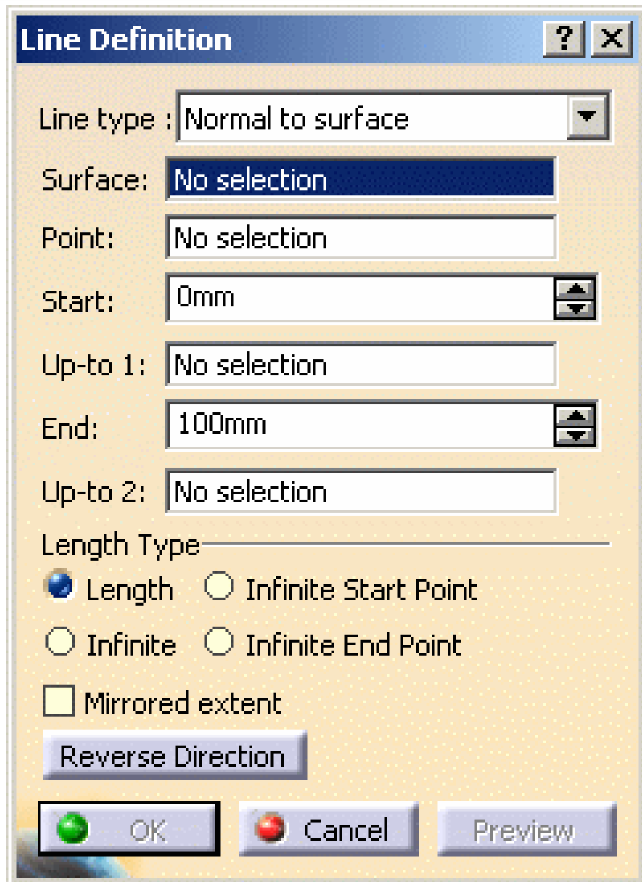
Line tangent to two curves

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

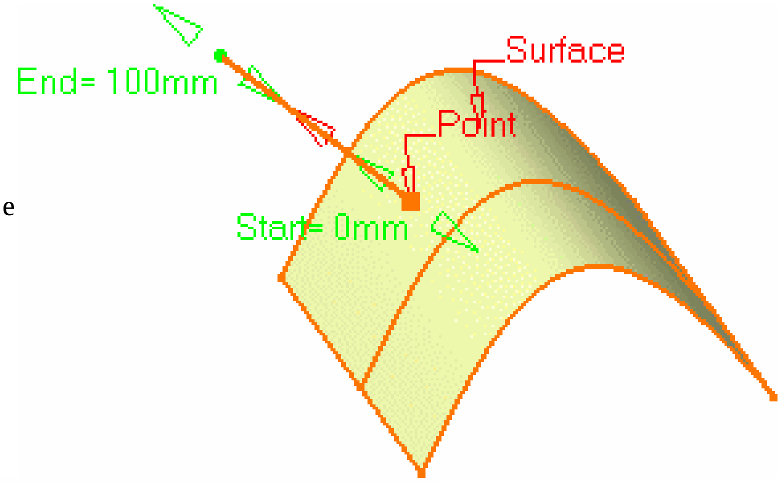


Normal to surface

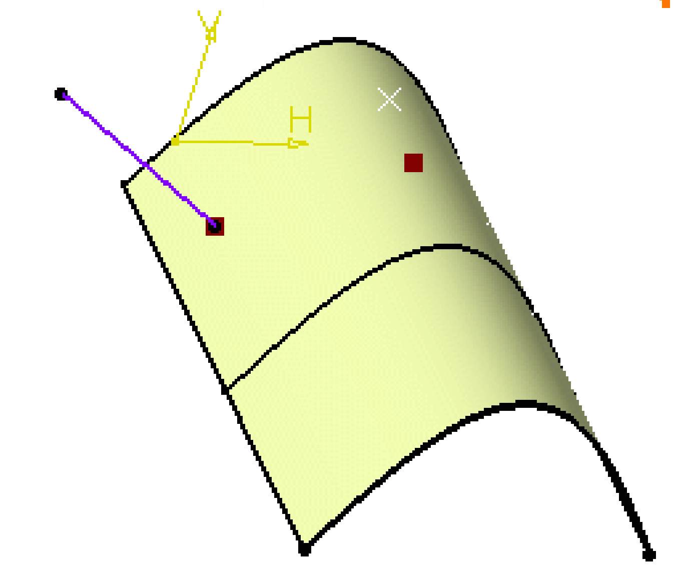
- Select a reference **Surface** and a **Point**.
  A vector normal to the surface is displayed at the reference point.
  Proposed **Start** and **End** points of the new line are shown.



If the point does not lie on the support surface, the minimum distance between the point and the surface is computed, and the vector normal to the surface is displayed at the resulted reference point.

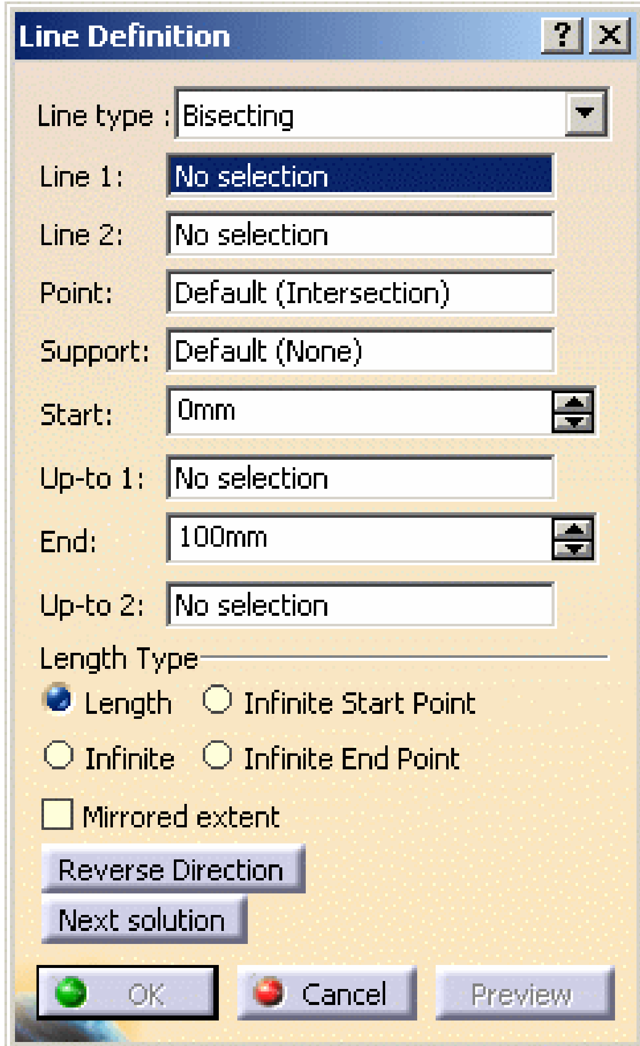


- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.

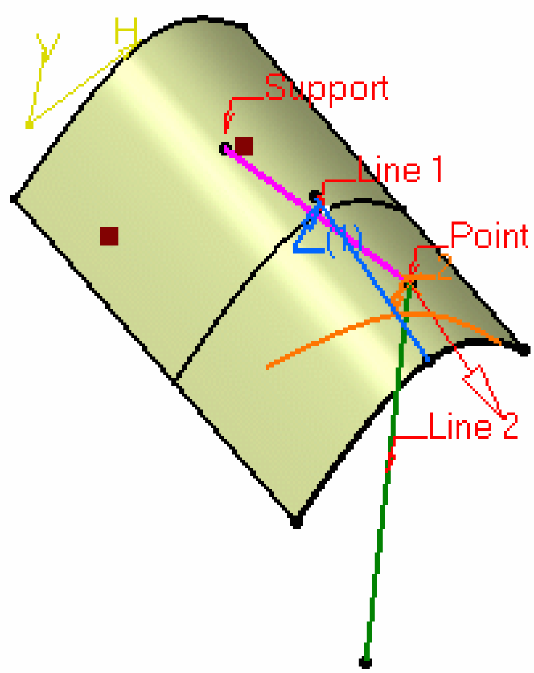


Bisecting

- 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.
- Check the **Mirrored extent** option to create a line symmetrically in relation to the selected **Start** point.
  It is only available with the **Length** Length type.
- 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 a line up to an element

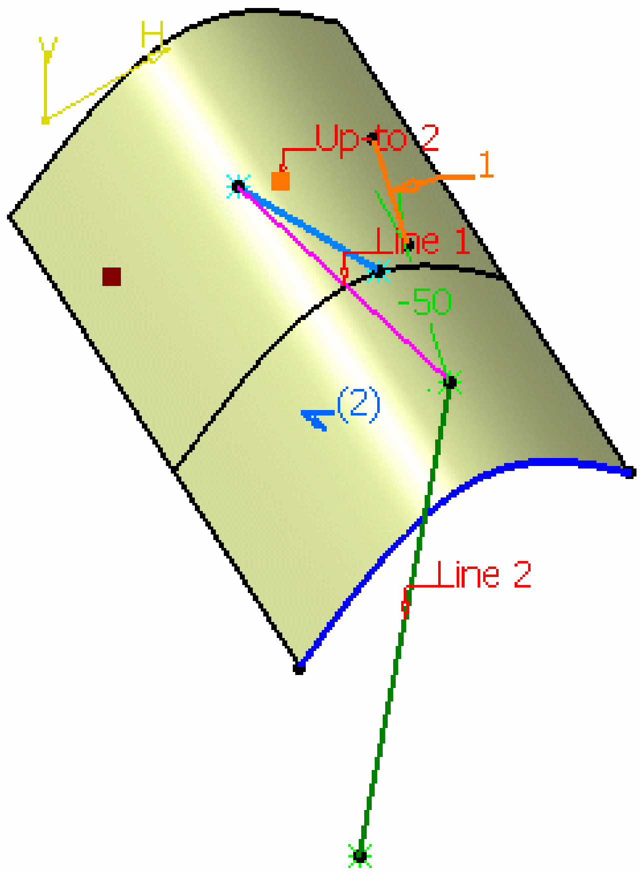
This capability allows you to create a line up to a point, a curve, or a surface.

- It is available with all line types, but the Tangent to curve type.

Up to a point

- Select a point in the **Up-to 1** and/or **Up-to 2** fields.

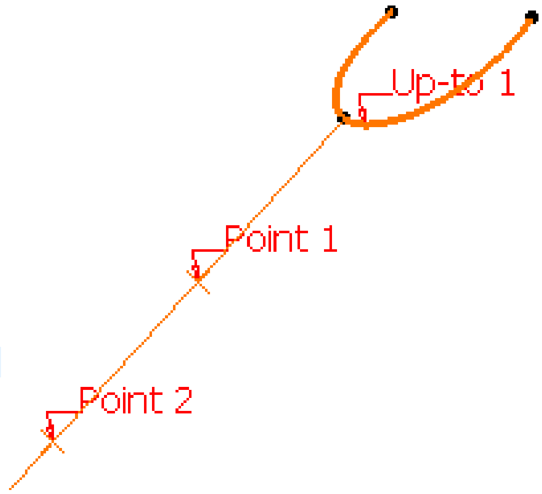
  Here is an example with the Bisecting line type, the **Length** Length type, and a point as **Up-to 2** element.



Up to a curve

- Select a curve in the **Up-to 1** and/or **Up-to 2** fields.

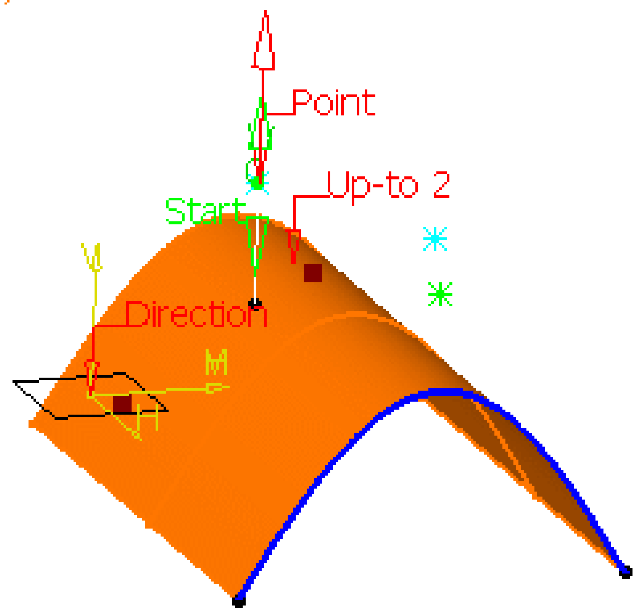
  Here is an example with the Point-Point line type, the **Infinite End** Length type, and a curve as the **Up-to 1** element.



Up to a surface

- Select a surface in the **Up-to 1** and/or **Up-to 2** fields.

  Here is an example with the Point-Direction line type, the **Length** Length type, and the surface as the **Up-to 2** element.



- If the selected Up-to element does not intersect with the line being created, then an extrapolation is performed. It is only possible if the element is linear and lies on the same plane as the line being created.
  However, no extrapolation is performed if the Up-to element is a curve or a surface.
- The **Up-to 1** and **Up-to 2** fields are grayed out with the **Infinite** Length type, the **Up-to 1** field is grayed out with the **Infinite Start** Length type, the Up-to 2 field is grayed out with the **Infinite End** Length type.
- The **Up-to 1** field is grayed out if the **Mirrored extent** option is checked.
- In the case of the Point-Point line type, **Start** and **End** values cannot be negative.

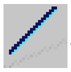
Defining the length type

- 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 out when one of the **Infinite** options is chosen.

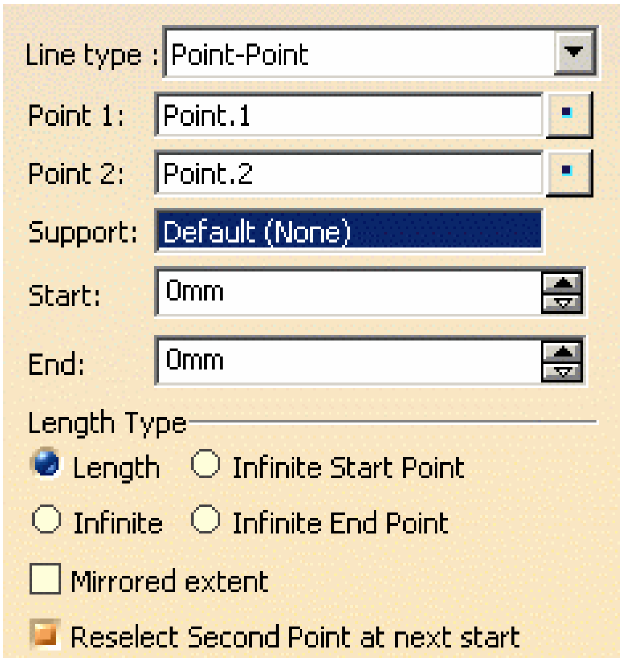
Reselecting automatically a second point

This capability is only available with the Point-Point line method.

1. Double-click the **Line** icon .

   The Line dialog box is displayed.

2. Create the first point.



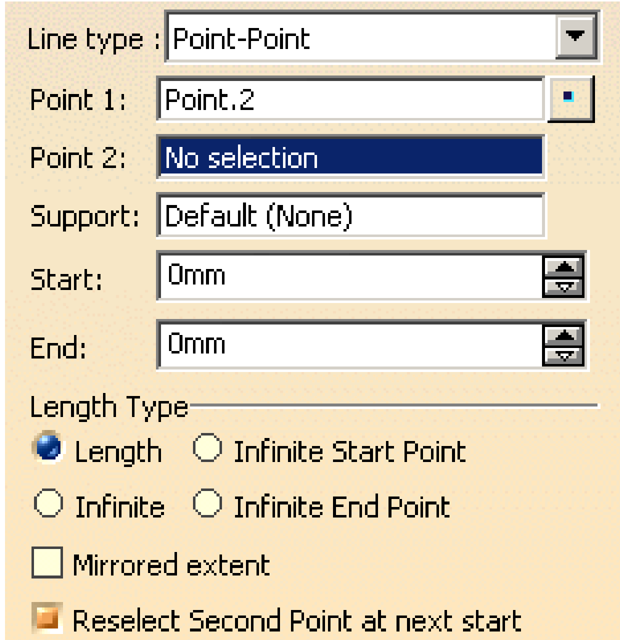
The **Reselect Second Point at next start** option appears in the Line dialog box.

3. Check it to be able to later reuse the second point.

4. Create the second point.

5. Click OK to create the first line.

The Line dialog box opens again with the first point initialized with the second point of the first line.



6. Click OK to create the second line.

To stop the repeat action, simply uncheck the option or click Cancel in the Line dialog box.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a line in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

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.

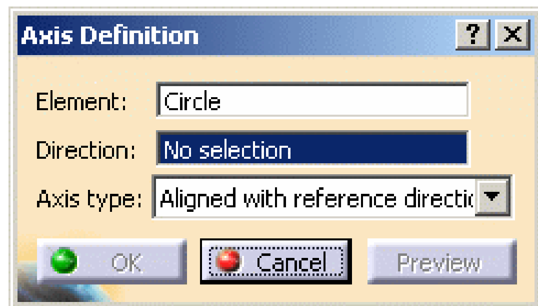
2. Select an **Element** where to create the axis.

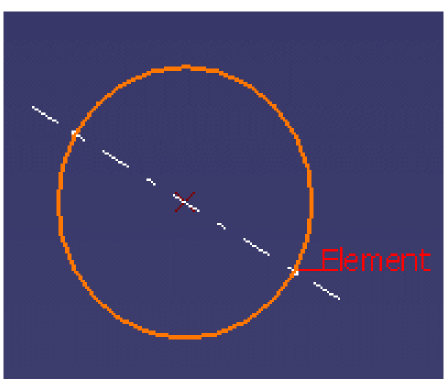
This element can be:

- 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

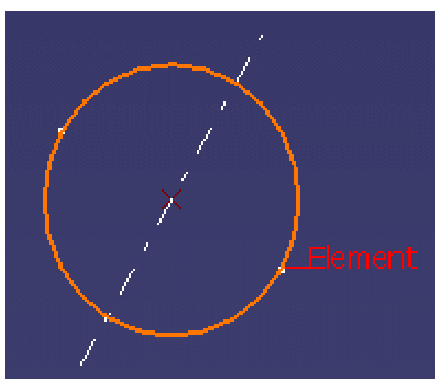
Circle

- 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

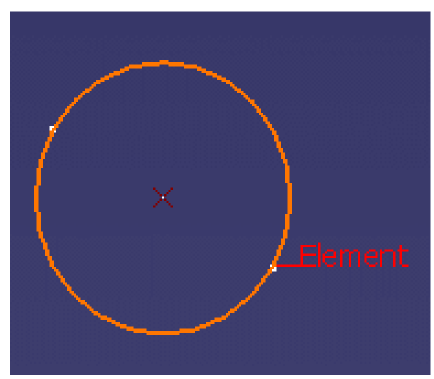




Aligned with reference direction



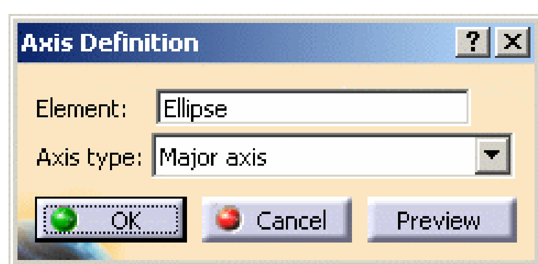
Normal to reference direction

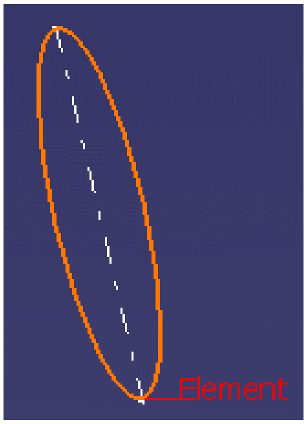


Normal to circle

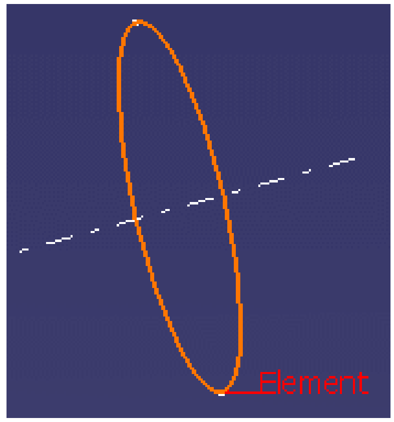
Ellipse

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

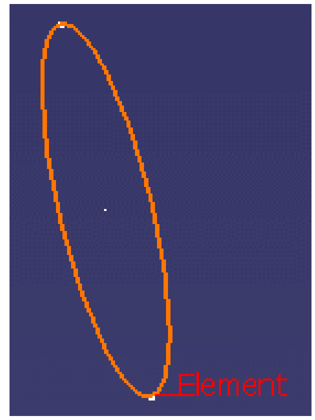




Major axis



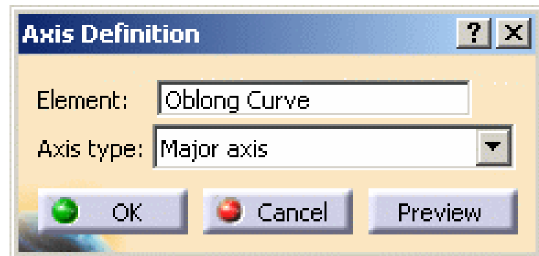
Minor axis

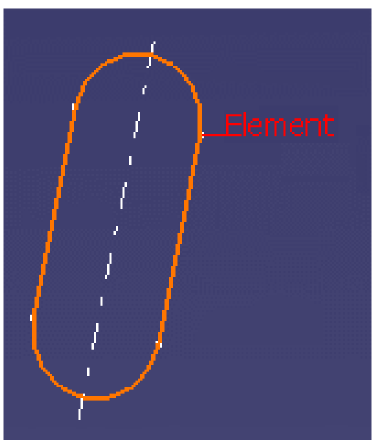


Normal to ellipse

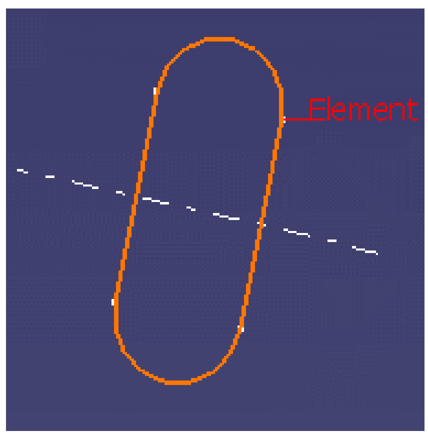
Oblong Curve

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

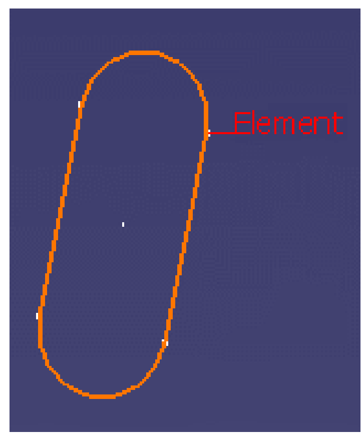




Major axis



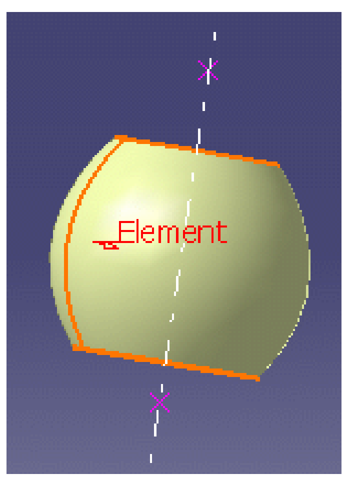
Minor axis



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.

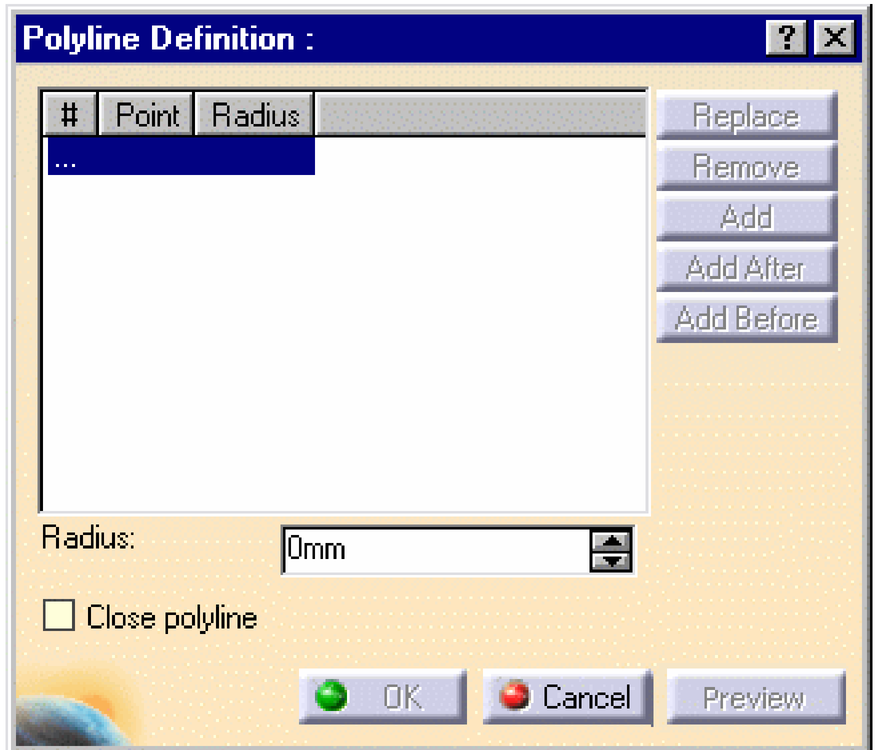
Creating Polylines

This task shows you how to create a polyline, that is a broken line made of several connected segments.
These linear segments may be connected by a 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.

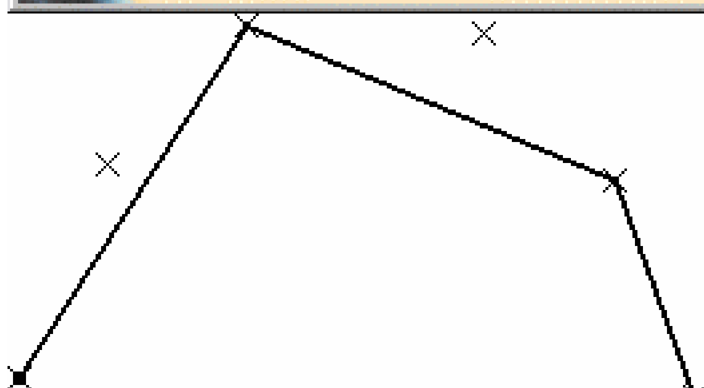
The Polyline Definition dialog box appears.



2. Select several points in a row.

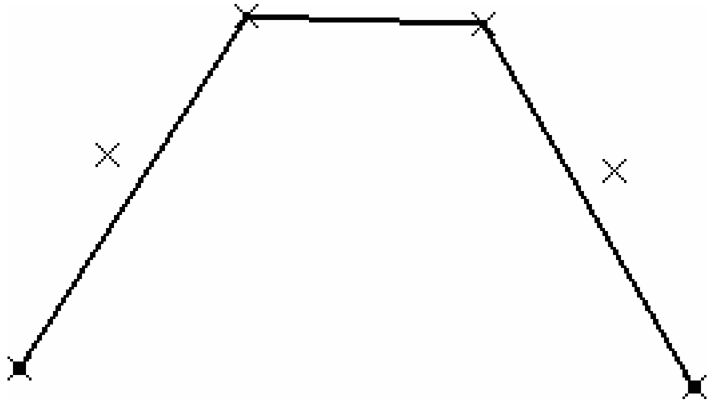
   Here we selected Point.1, Point.5, Point.3 and Point.2 in this order.

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.

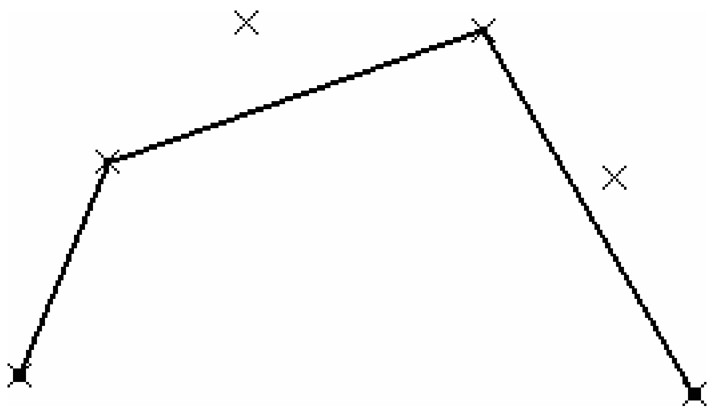


The resulting polyline now looks like this:

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.

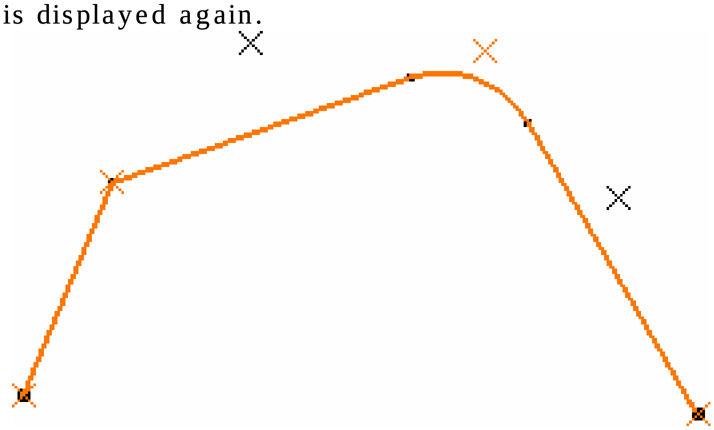


The element (identified as Polyline.xxx) is added to the specification tree.

7. Double-click the polyline from the specification tree.

The Polyline Definition dialog box is displayed again.

8. Select Point.6 within the dialog box, enter a value in the Radius field, and click Preview.

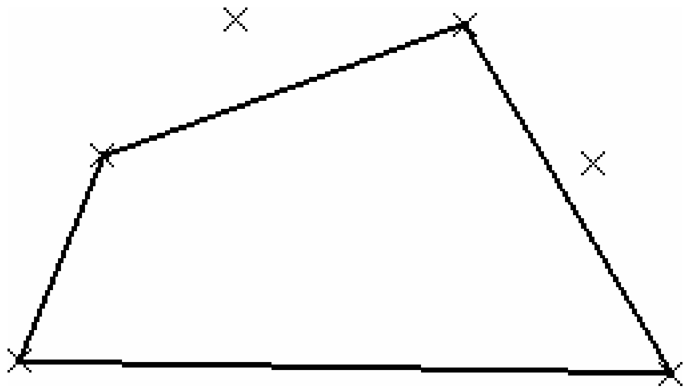


A curve, centered on Point.6, and which radius is the entered value (R=30 here) is created.

- You can define a radius for each point, except end points.
- You can also define radii at creation time.
- The blending curve's center is located on the side of the smallest angle between the two connected line segments.

9. Click OK to accept the new definition of the polyline.

- 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.



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.

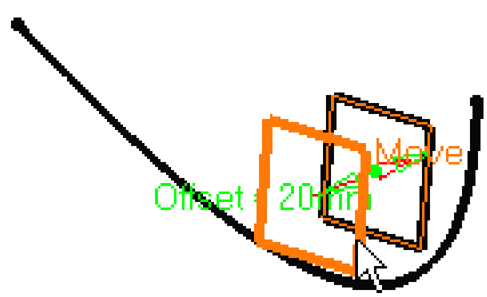
2. Use the combo to choose the desired **Plane type**.

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

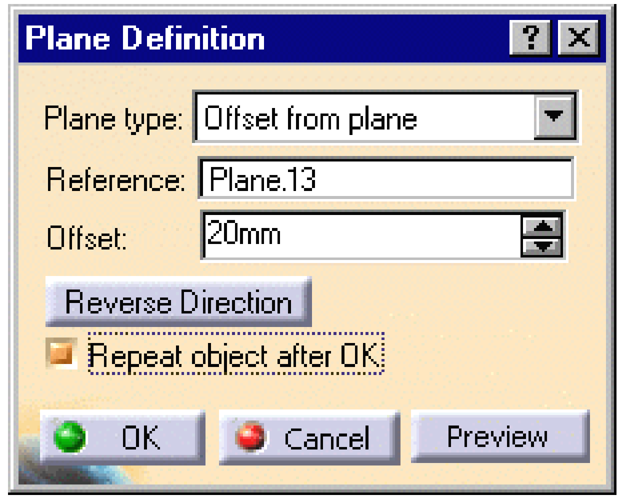
Offset from plane

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

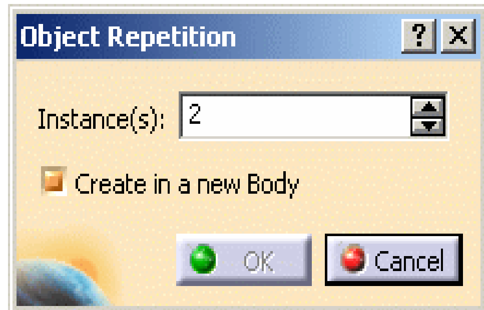
A plane is displayed offset from the reference plane.



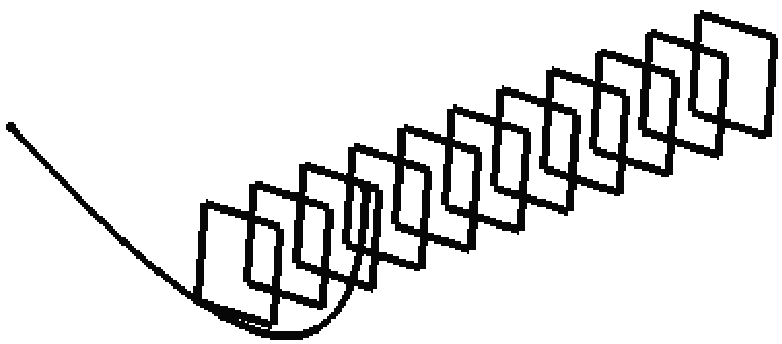
Use the **Reverse Direction** button to reverse the change the offset direction, or simply click on the arrow in the geometry.



- 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.

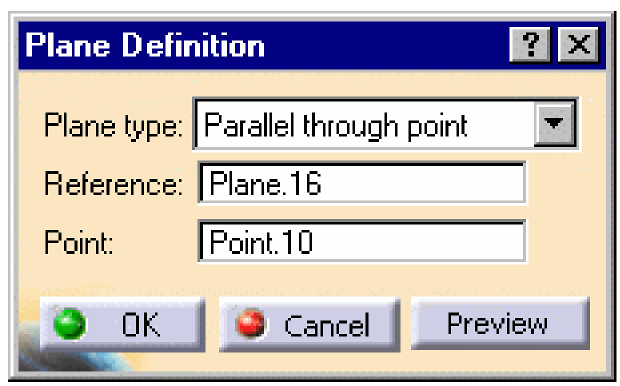


As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Offset** value.

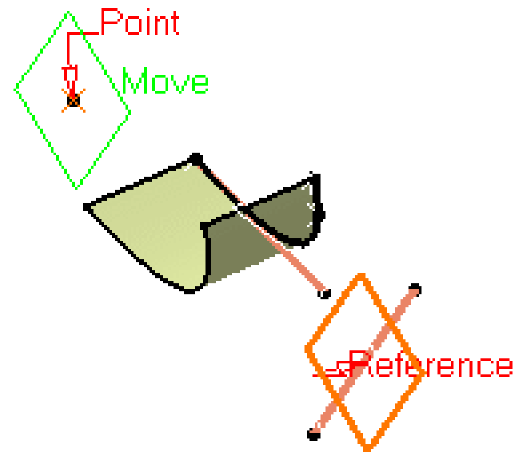


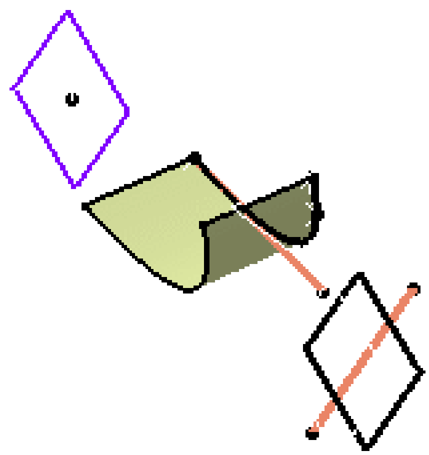
Parallel through point

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



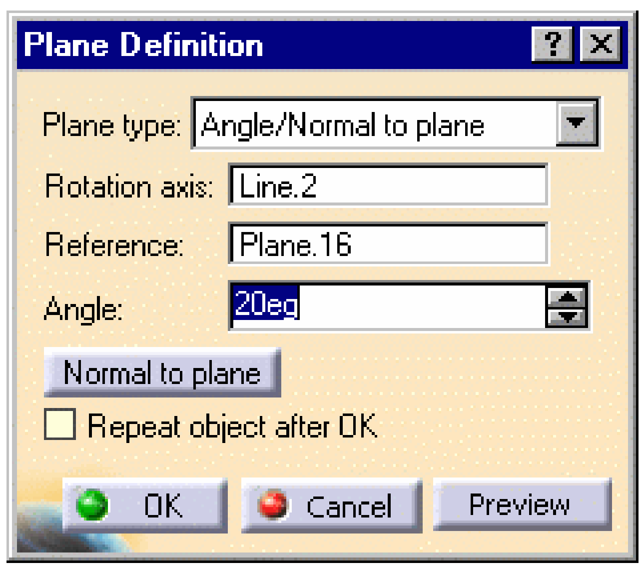
A plane is displayed parallel to the reference plane and passing through the selected point.



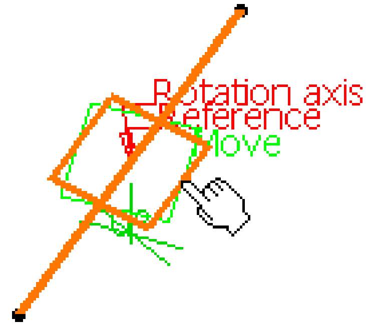


Angle or normal to plane

- 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.



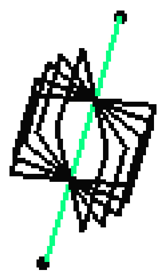
A plane is displayed passing through the rotation axis. It is oriented at the specified angle to the reference plane.



- 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.

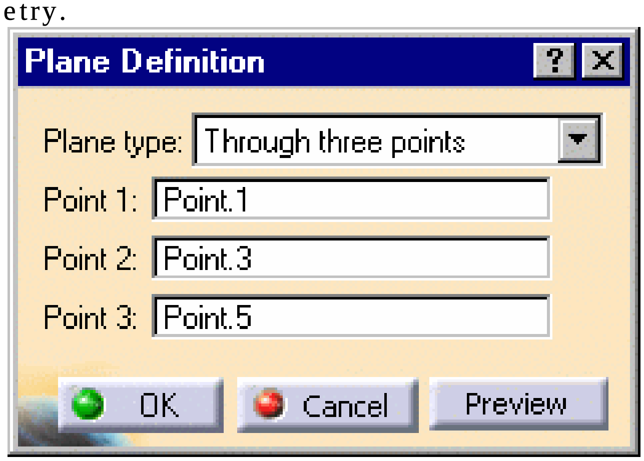
As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Angle** value.

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



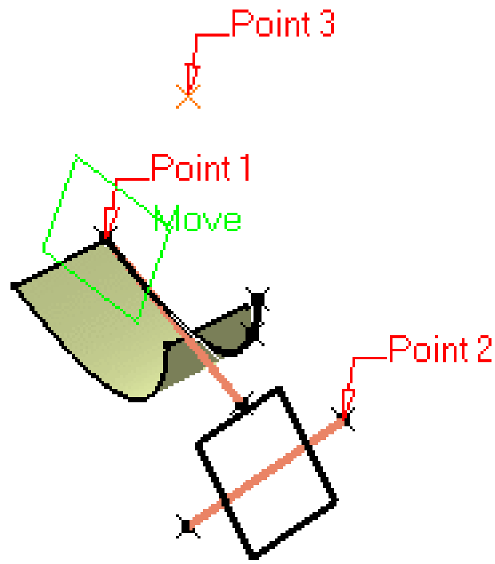
This plane type enables to edit the plane's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

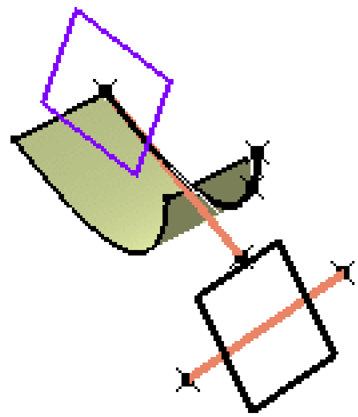
Through three points



- Select three points.

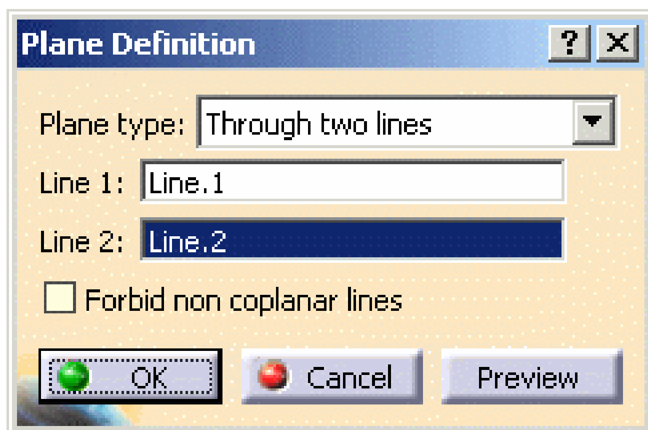
The plane passing through the three points is displayed. You can move it simply by dragging it to the desired location.



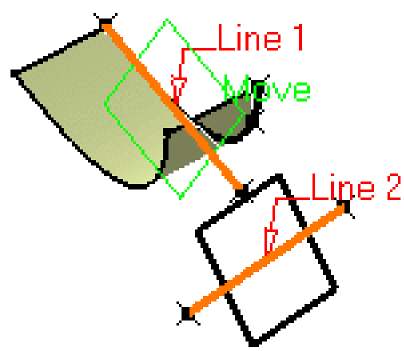


Through two lines

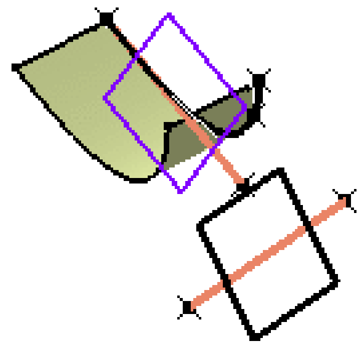
- Select two lines.

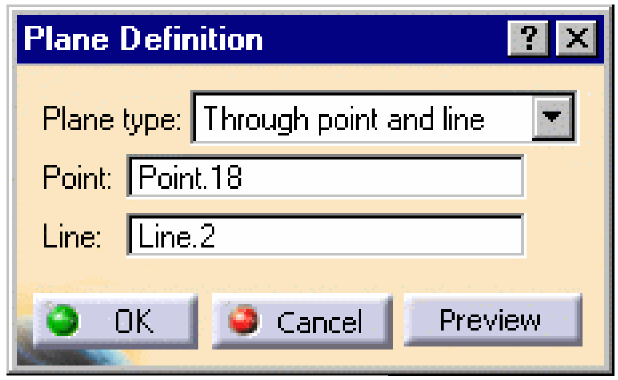


The plane passing through the two line directions is displayed.
When these two lines are not coplanar, the vector of the second line is moved to the first line location to define the plane's second direction.



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

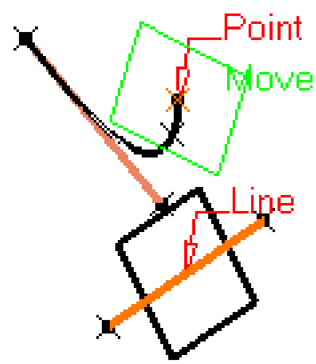


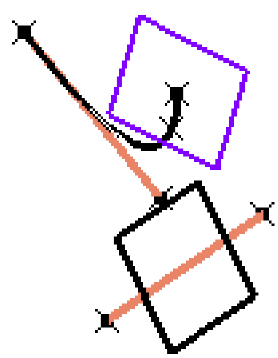


Through point and line

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

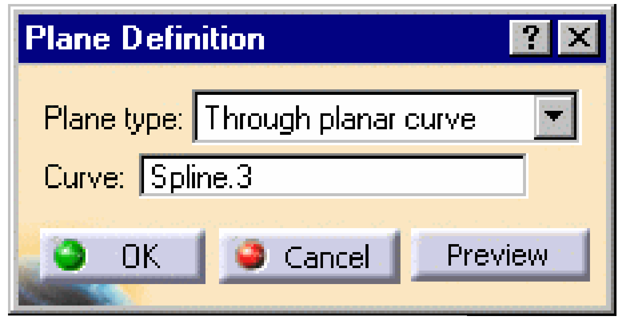
The plane passing through the point and the line is displayed.

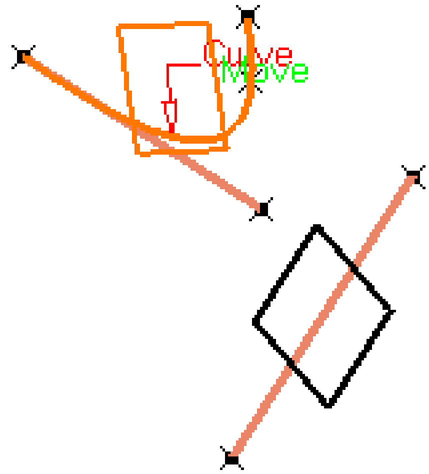




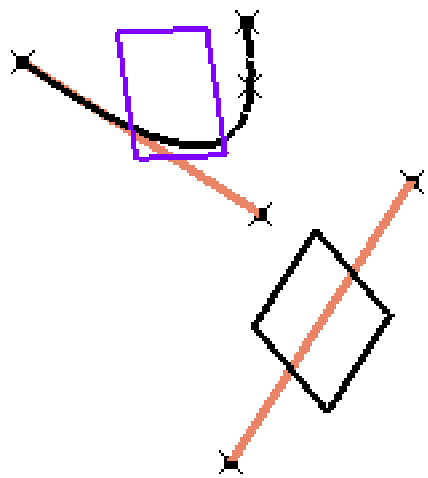
Through planar curve

- Select a planar **Curve**.



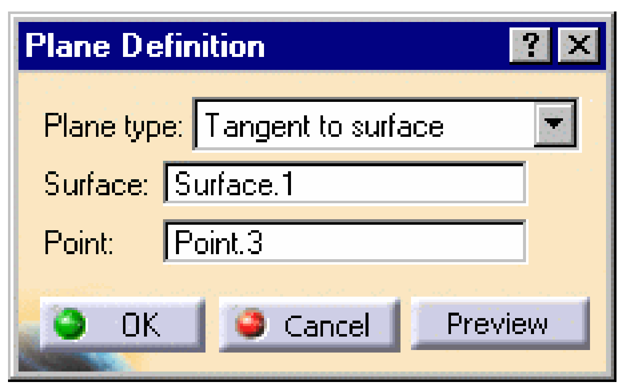


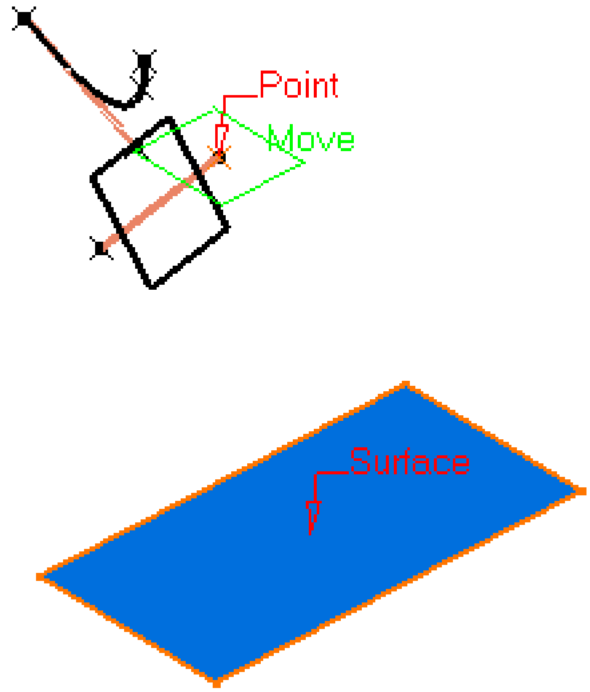
The plane containing the curve is displayed.



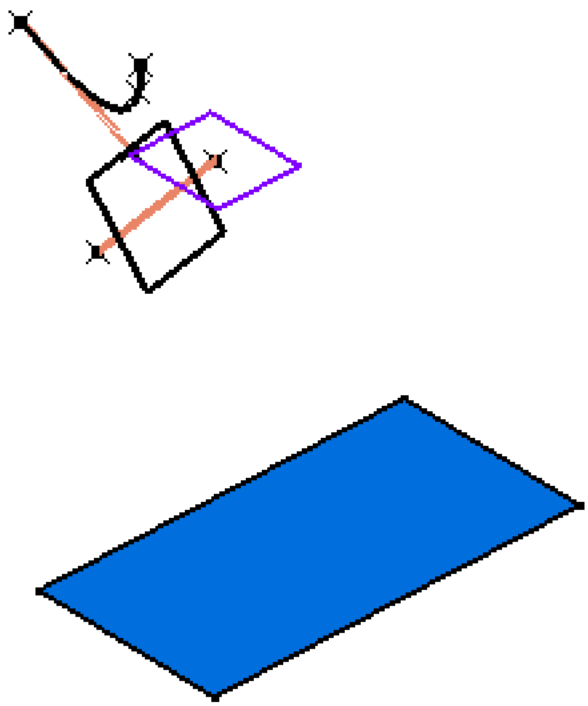
Tangent to surface

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



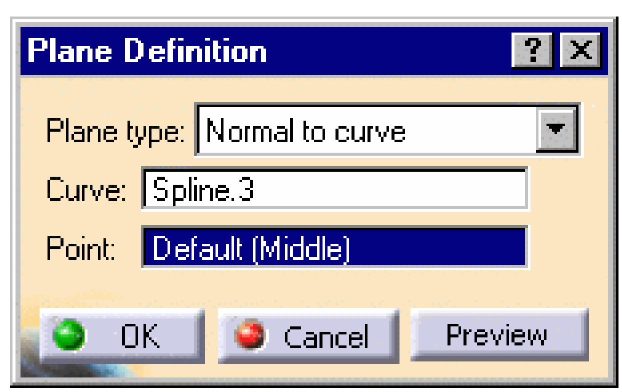


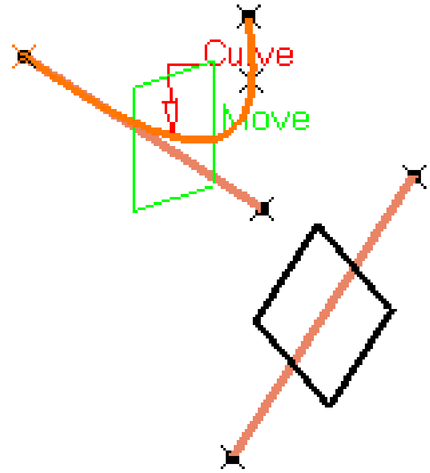
A plane is displayed tangent to the surface at the specified point.



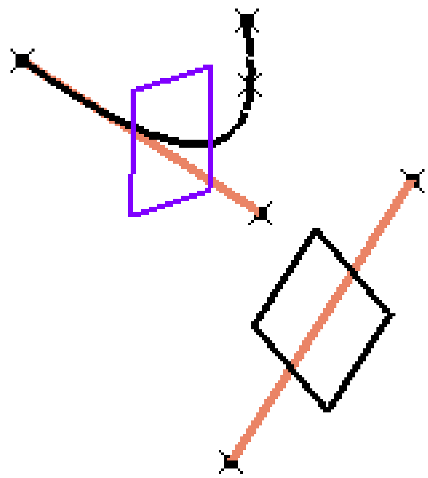
Normal to curve

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



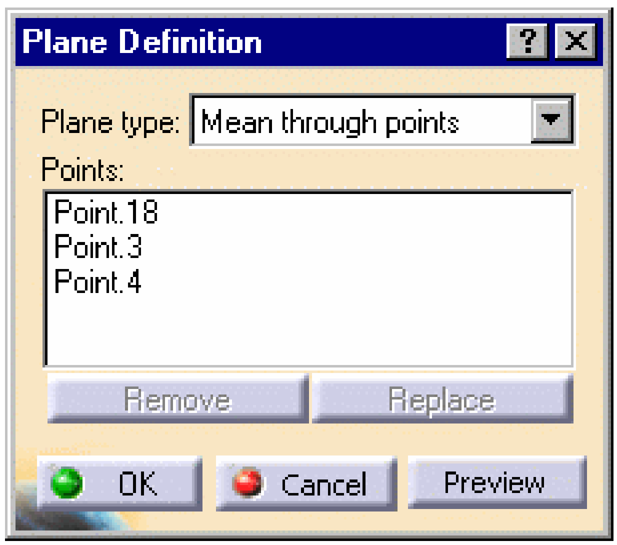


A plane is displayed normal to the curve at the specified point.



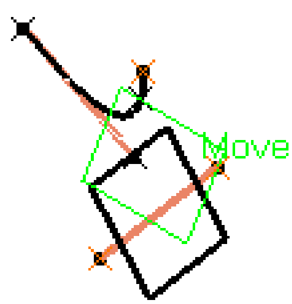
Mean through points

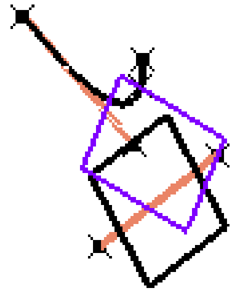
- Select three or more points to display the mean plane through these points.



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

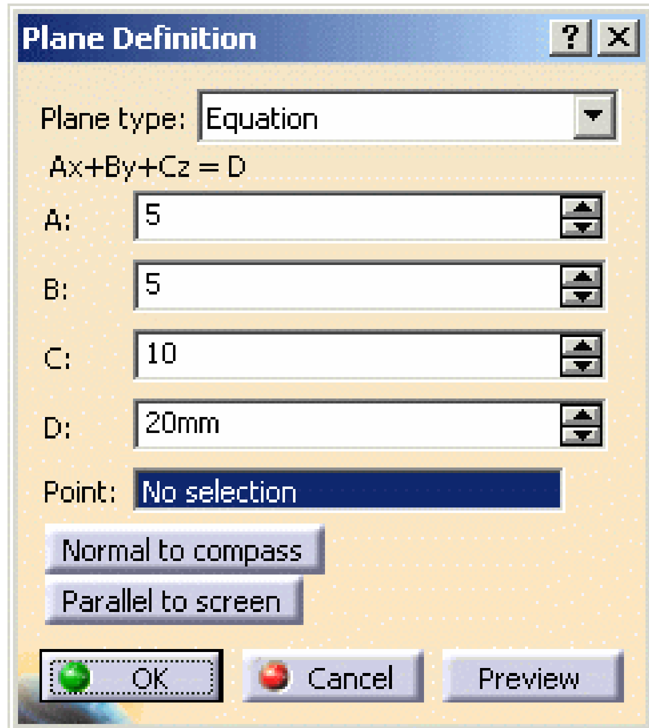
- **Remove** the selected point
- **Replace** the selected point by another point.



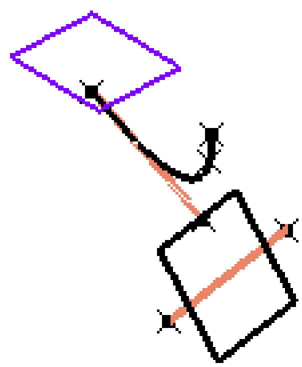


Equation

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

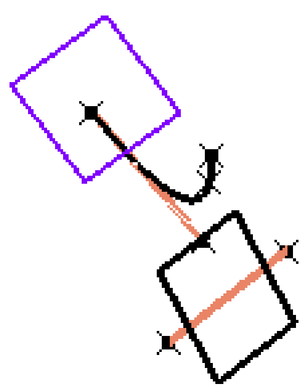


Select a point to position the plane through this point, you are able to modify **A**, **B**, and **C** components, the **D** component becomes grayed.



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.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a plane in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

Creating Planes Between Other Planes

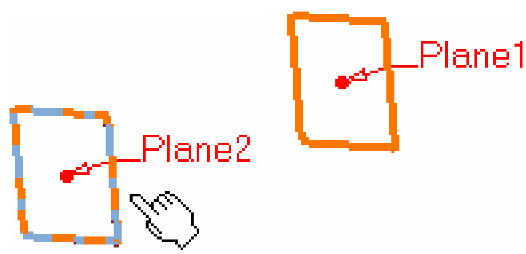
This task shows how to create any number of planes between two existing planes, in only one operation:

Open the Planes1.CATPart document.

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

The Planes Between dialog box appears.

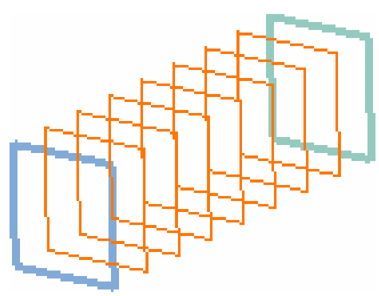
2. Select the two planes between which the new planes must be created.



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.



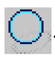
Check the **Create in a new geometrical set** button to create a new Geometrical Set containing only the repeated planes.

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
- center and axis
- bitangent and radius
- bitangent and point
- tritangent
- center and tangent

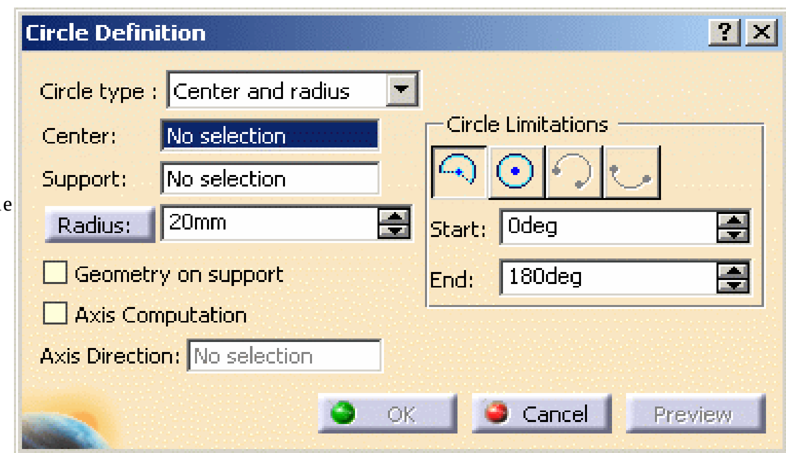
Open the Circles1.CATPart document.
Please note that you need to put the desired geometrical set in show to be able to perform the corresponding scenario.

1. Click the **Circle** icon .

The Circle Definition dialog box appears.

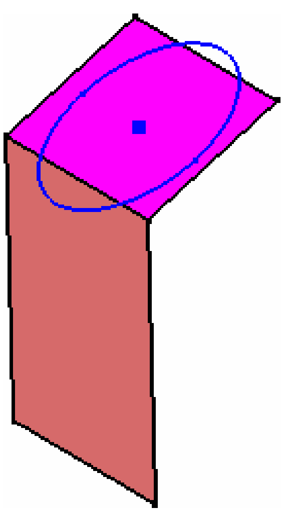
2. Use the drop-down list to choose the desired circle type.

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.

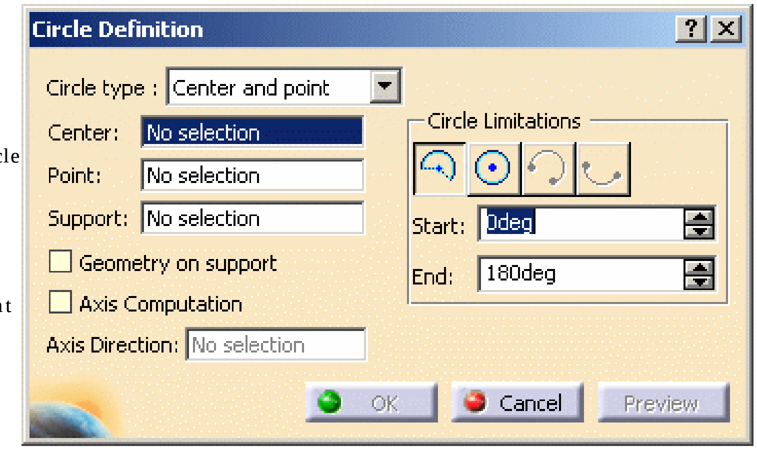


If a support surface is selected, the circle lies on the plane tangent to the surface at the selected point.

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

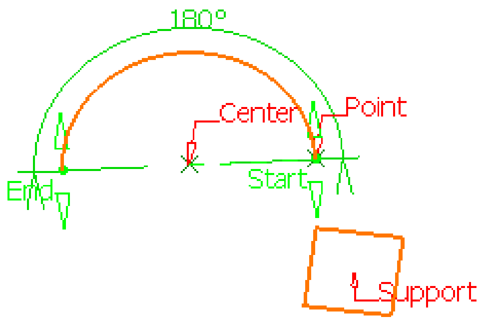
Center and point

- 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.



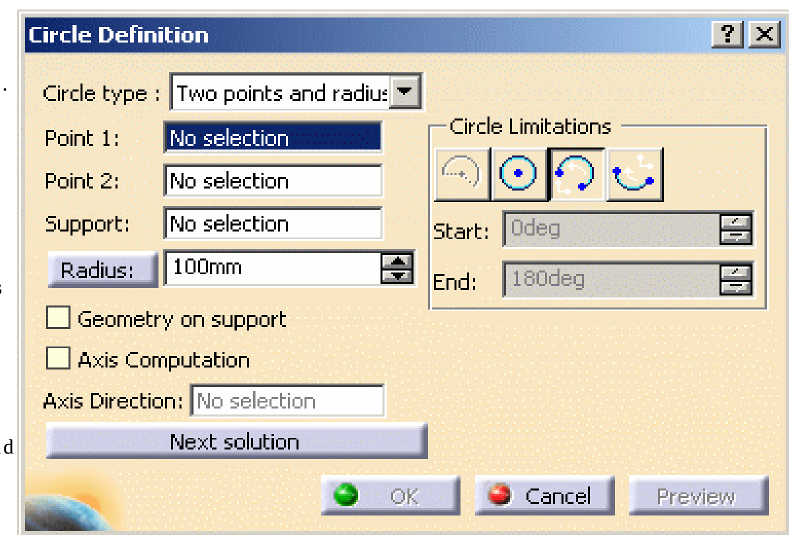
The circle, which center is the first selected point and passing through the second point or the projection of this second point on the plane tangent to the surface at the first point, is previewed.

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

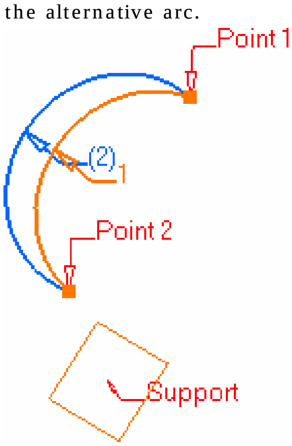
- Select two points on a surface or in the same plane.
- Select the **Support** plane or surface.
- Enter a **Radius** value.



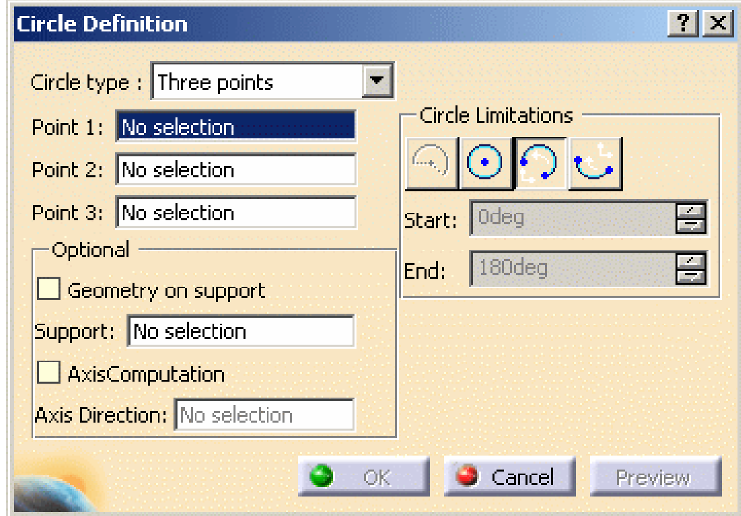
The circle, passing through the first selected point and the second point or the projection of this second point on the plane tangent to the surface at the first point, is previewed.

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.

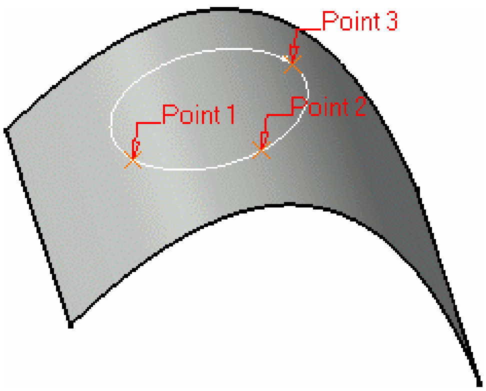


Three points

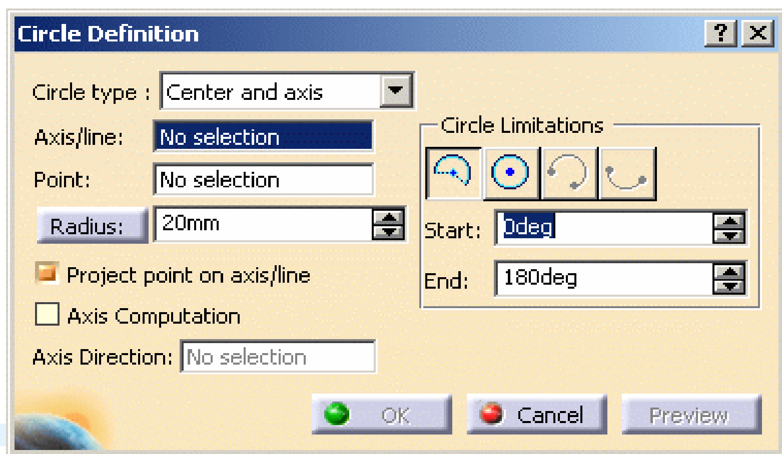


- Select three points where the circle is to be created.

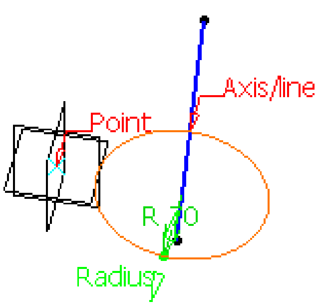
Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the trimmed or complementary arc using the two of the selected points as end points.



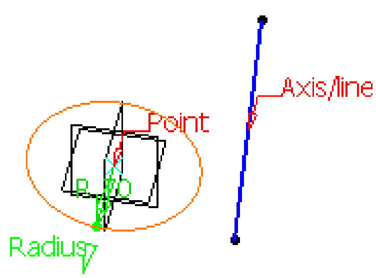
Center and axis



- Select the axis/line.
  It can be any linear curve.
- Select a point.
- Enter a **Radius** value.
- Set the **Project point on axis/line** option:
  - checked (with projection): the circle is centered on the reference point and projected onto the input axis/line and lies in the plane normal to the axis/line passing through the reference point. The line will be extended to get the projection if required.
  - unchecked (without projection): the circle is centered on the reference point and lies in the plane normal to the axis/line passing though the reference point.

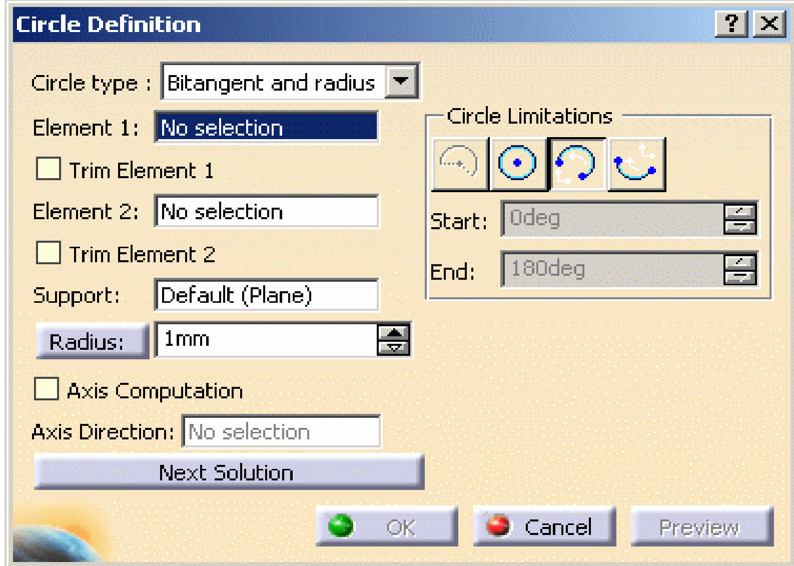


With projection



Without projection

Bi-tangent and radius



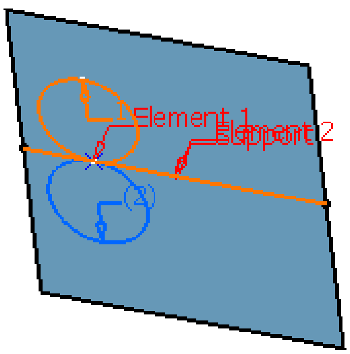
- Select two **Elements** (point or curve) to which the circle is to be tangent.
- Select a **Support** surface.

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.

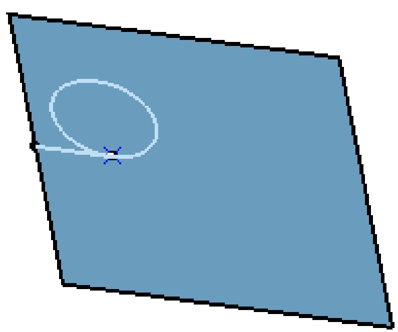
This automatic support definition saves you from performing useless selections.

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

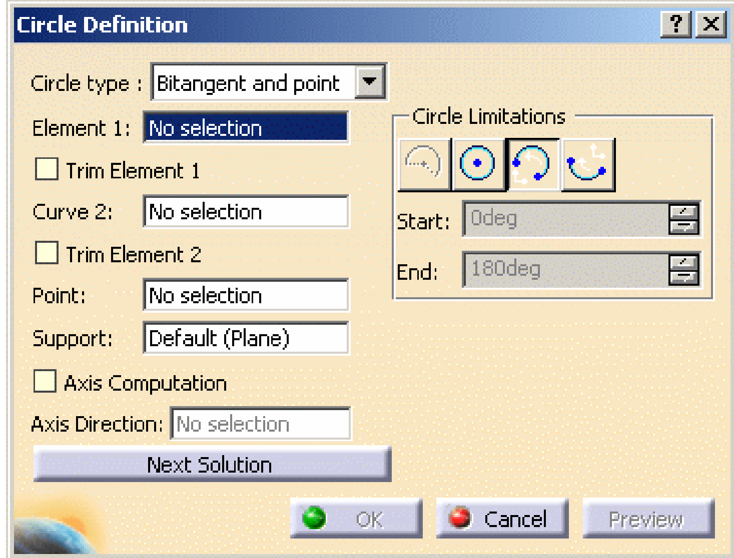
Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the trimmed or complementary arc using the two tangent points as end points.



You can select the **Trim Element 1** and **Trim Element 2** check boxes to trim the first element or the second element, or both elements.
Here is an example with Element 1 trimmed.



These options are only available with the Trimmed Circle limitation.



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.

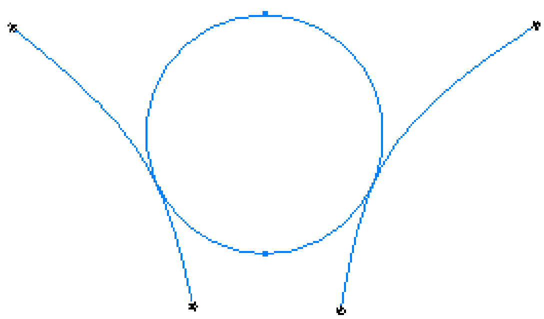
The point will be projected onto the curve.

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.

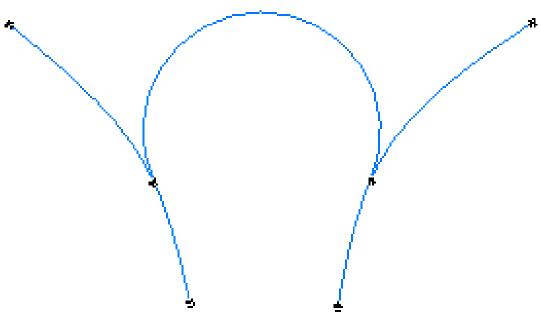
- Several solutions may be possible, so click in the region where you want the circle to be.

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

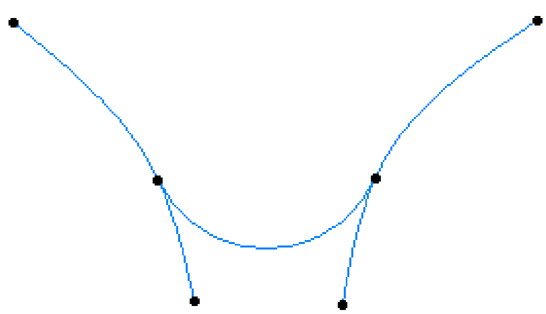


Complete circle

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

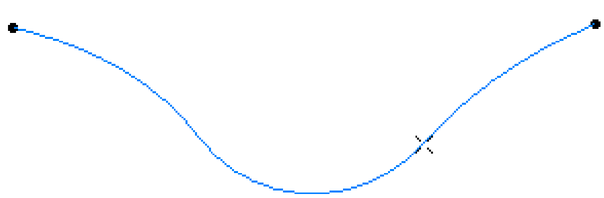


Trimmed circle



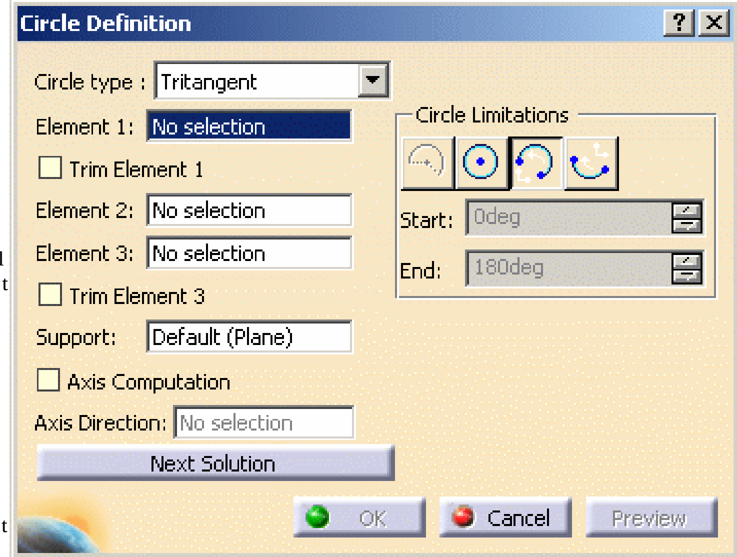
Complementary trimmed circle

You can select the **Trim Element 1** and **Trim Element 2** check boxes to trim the first element or the second element, or both elements.
Here is an example with both elements trimmed.



These options are only available with the Trimmed Circle limitation.

Tritangent



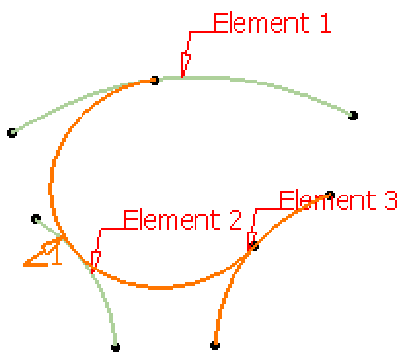
- Select three **Elements** to which the circle is to be tangent.
- Select a **Support** planar surface.

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.

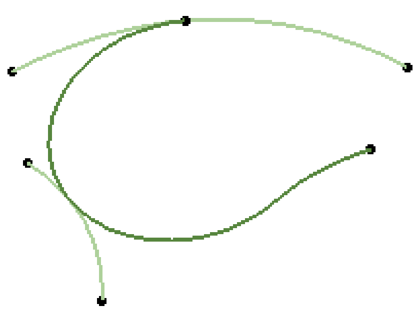
This automatic support definition saves you from performing useless selections.

- 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.



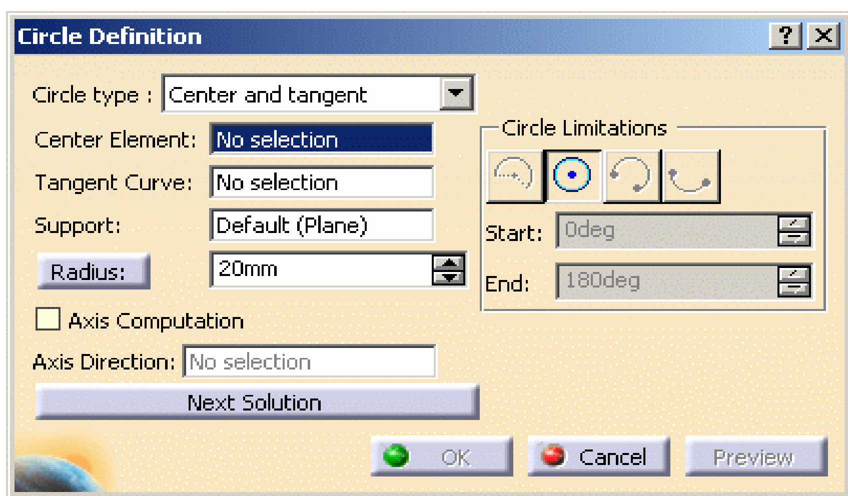
You can select the **Trim Element 1** and **Trim Element 3** check boxes to trim the first element or the third element, or both elements.
Here is an example with Element 3 trimmed.



These options are only available with the Trimmed Circle limitation.

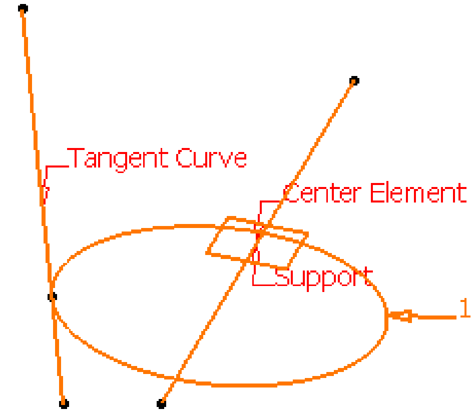
Center and tangent

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



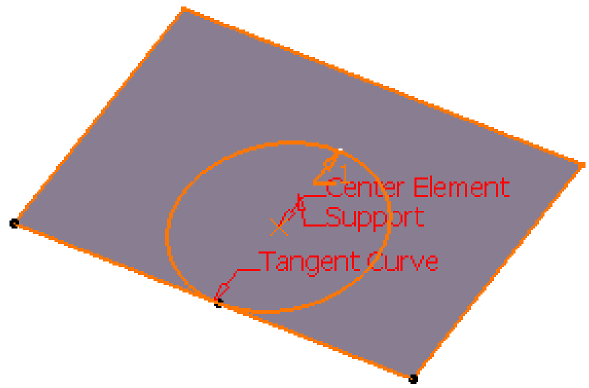
1. Center curve and radius

- 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.

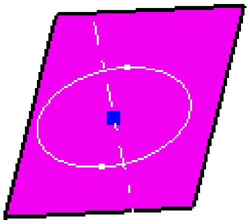
- You can click the **Diameter** button to switch to a Diameter value. Conversely, click the **Radius** button to switch back to the Radius value.
  This option is available with the **Center and radius**, **Two point and radius**, **Bi-tangent and radius**, **Center and tangent**, and **Center and axis** circle types.

Note that the value does not change when switching from **Radius** to **Diameter** and vice-versa.

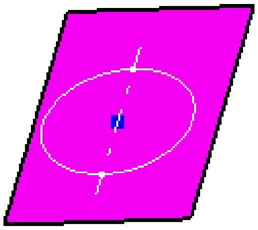
- You can select the **Axis computation** check box to automatically create axes while creating or modifying a circle. Once the option is checked, the Axis direction field is enabled.
  - If you do not select a direction, an axis normal to the circle will be created.
  - If you select a direction, two more axes features will be created: an axis aligned with the reference direction and an axis normal to the reference direction.

In the specification tree, the axes are aggregated under the Circle feature. You can edit their directions but cannot modify them.

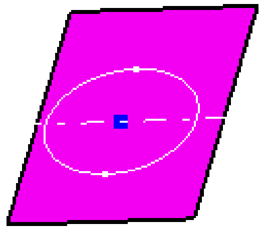
If the datum mode is active, the axes are not aggregated under the Circle features, but one ore three datum lines are created.



Axis normal to the circle



Axis aligned with the reference direction (yz plane)



Axis normal to the reference direction (yz plane)

If you select the **Geometry on Support** option and the selected support is not planar, then the Axis Computation is not possible.

- 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.

  This option is available with the **Center and radius**, **Center and point**, **Two point and radius**, and **Three points** circle types.

- 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.

A circle may have several points as center if the selected element is made of various circle arcs with different centers.

- Parameters can be edited in the 3D geometry. For more information, refer to the Editing Parameters chapter.
- You can isolate a plane in order to cut the links it has with the geometry used to create it. To do so, use the **Isolate** contextual menu. For more information, refer to the Isolating Features chapter.

Creating Conic Curves

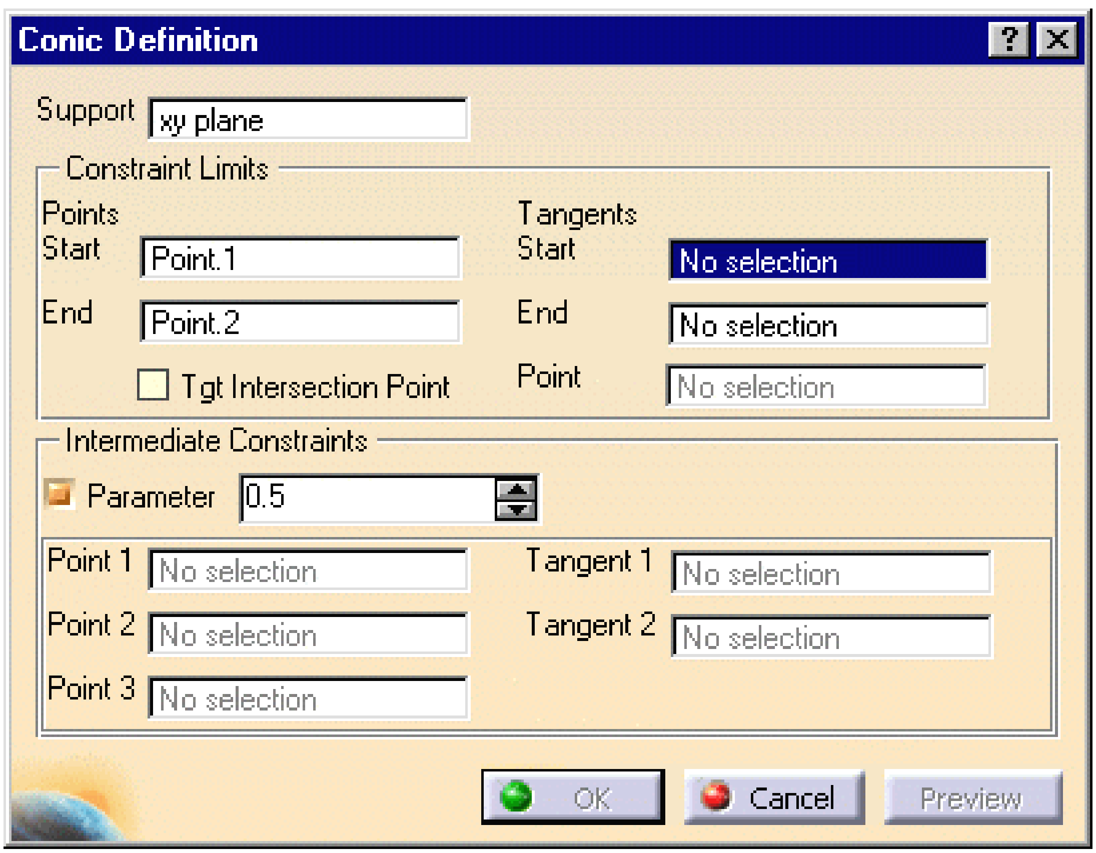
This task shows the various methods for creating conics, that is curves defined by five constraints: start and end points, passing points or tangents. The resulting curves are arcs of either parabolas, hyperbolas or ellipses.
The different elements necessary to define these curves are either:

- two points, start and end tangents, and a parameter
- two points, start and end tangents, and a passing point
- two points, a tangent intersection point, and a parameter
- two points, a tangent intersection point, and a passing point
- four points and a tangent
- five points.

Open the Conic1.CATPart document.

1. Click the Conic icon .

The **Conic Definition** dialog box opens.

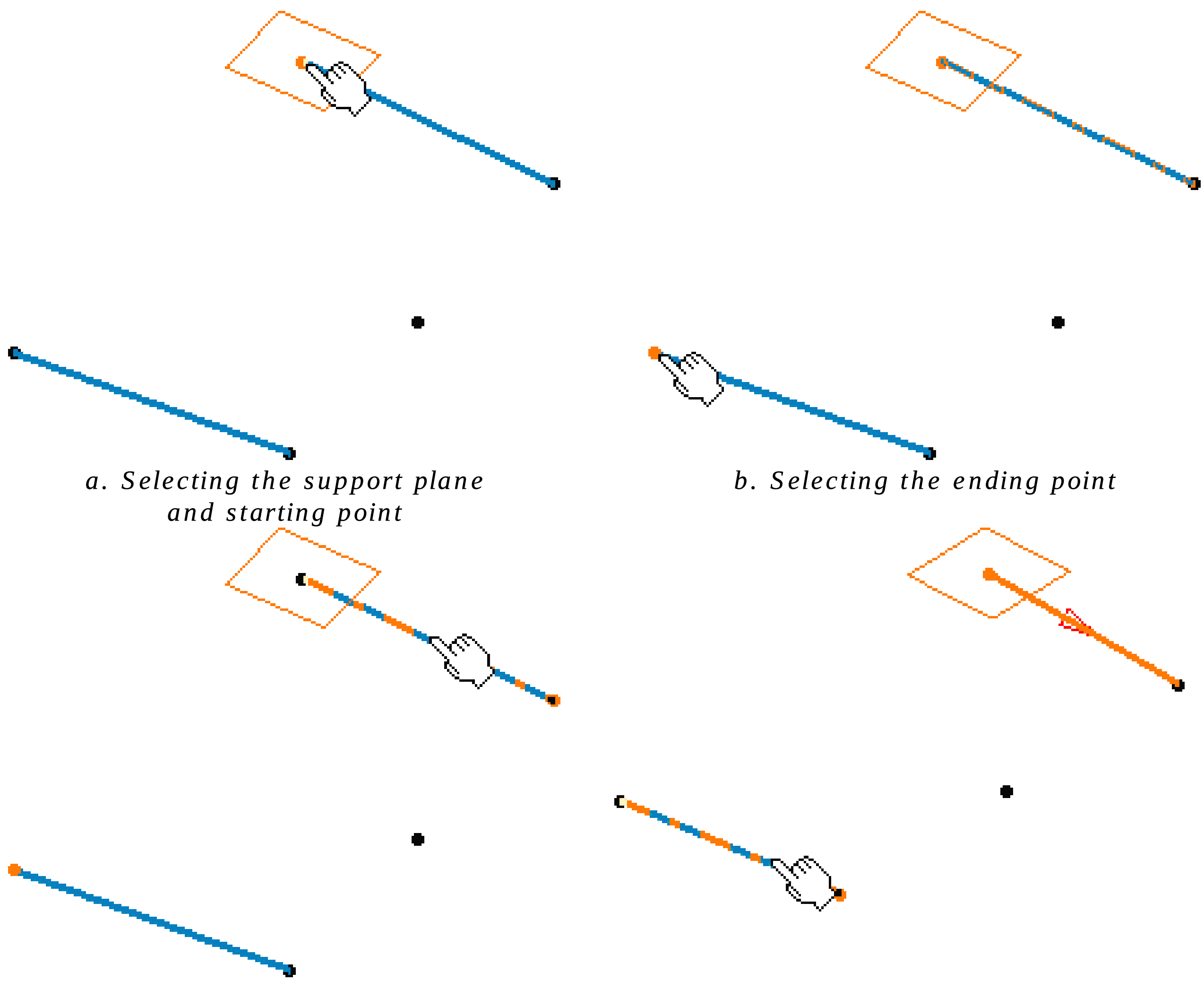


2. Fill in the conic curve parameters, depending on the type of curve to be created by selecting geometric elements (points, lines, etc.):

- **Support**: the plane on which the resulting curve will lie

Constraint Limits:

- **Start** and **End** points: the curve is defined from the starting point to the end point
- **Tangents Start** and **End**: if necessary, the tangent at the starting or end point defined by selecting a line
- **Tangent Intersection Point**: a point used to define directly both tangents from the start and end point. These tangents are on the virtual lines passing through the start (end) point and the selected point.

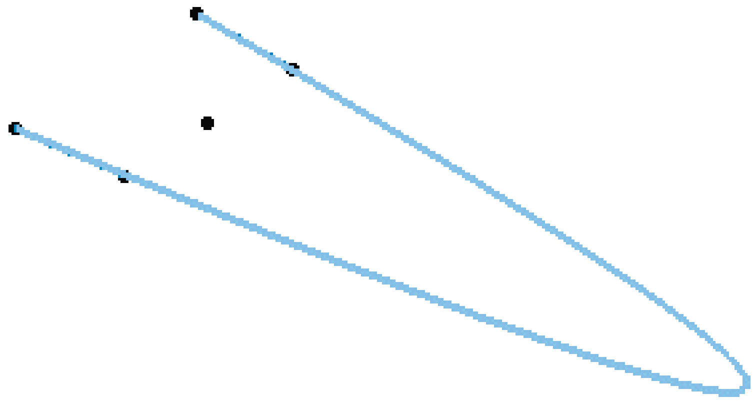


a. Selecting the support plane and starting point

b. Selecting the ending point

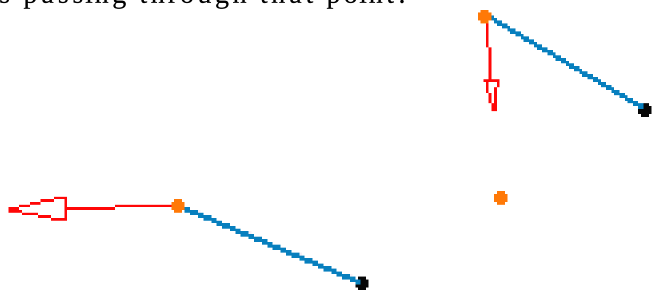
c. Selecting the tangent at the starting point

d. Selecting the tangent at the ending point

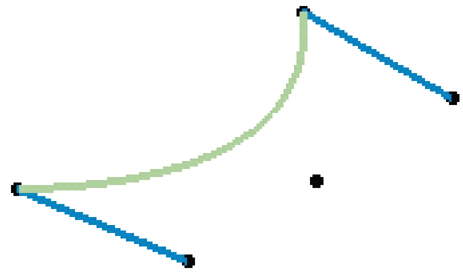


Resulting conic curve

If you check the **Tgt Intersection Point** option, and select a point, the tangents are created as passing through that point:



Using a tangent intersection point

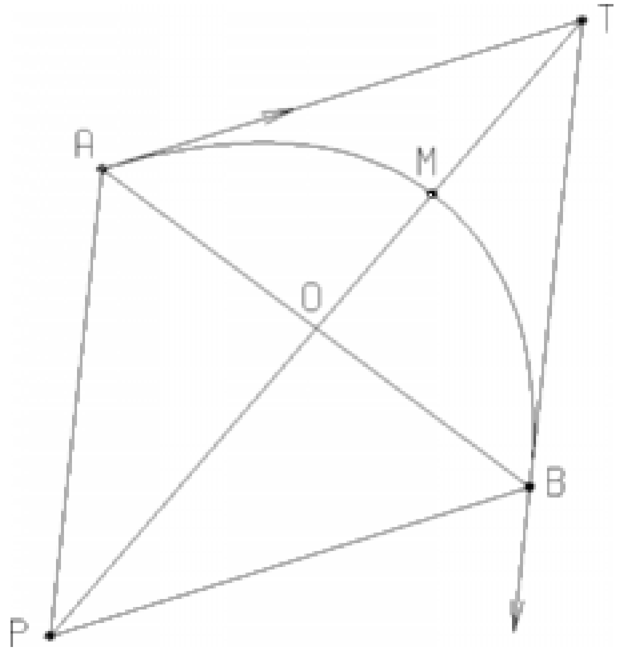


Resulting conic curve

Intermediate Constraints

- **Point 1, 2, 3**: possible passing points for the curve. These points have to be selected in logical order, that is the curve will pass through the start point, then through **Point 1**, **Point 2**, **Point 3** and the end point.
  Depending on the type of curve, not all three points have to be selected.
  You can define tangents on **Point 1** and **Point 2** (**Tangent 1** or **2**).

- **Parameter**: ratio ranging from 0 to 1 (excluded), this value is used to define a passing point (M in the figure below) and corresponds to the OM distance/OT distance.
  If parameter = 0.5, the resulting curve is a parabola
  If 0 < parameter < 0.5, the resulting curve is an arc of ellipse,
  If 1 > parameter > 0.5, the resulting curve is a hyperbola.



3. Click OK to create the conic curve.

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

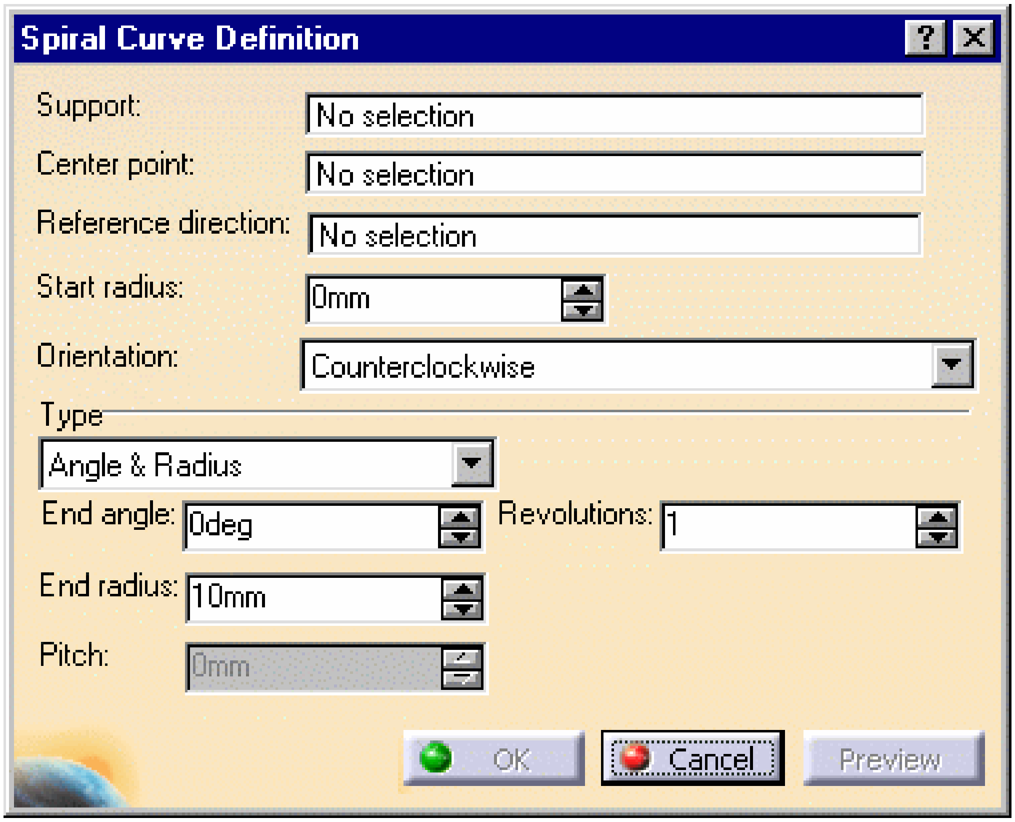
Creating Spirals

This task shows how to create curves in the shape of spirals, that is a in 2D plane, as opposed to the helical curves.

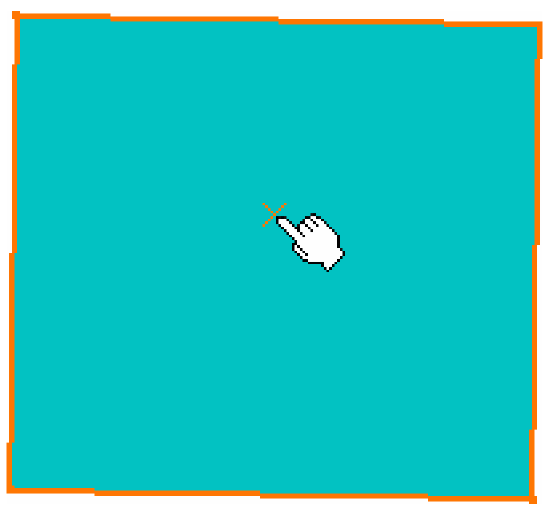
Open the Spiral1.CATPart document.

1. Click the **Spiral** icon.

   The Spiral Curve Definition dialog box appears.

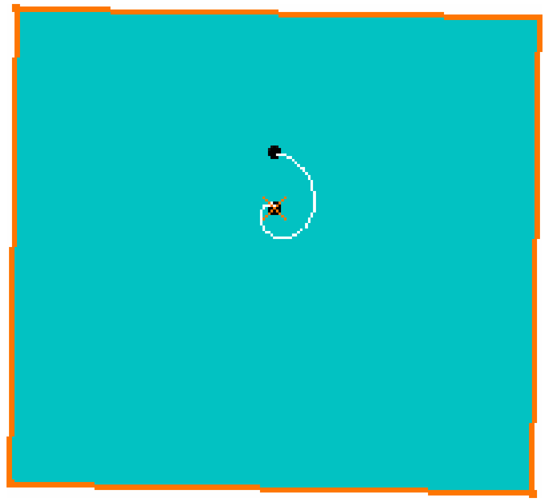


2. Select a supporting plane and the **Center point** for the spiral.



3. Specify a **Reference direction** along which the **Start radius** value is measured and from which the angle is computed, when the spiral is defined by an angle.

The spiral is previewed with the current options:

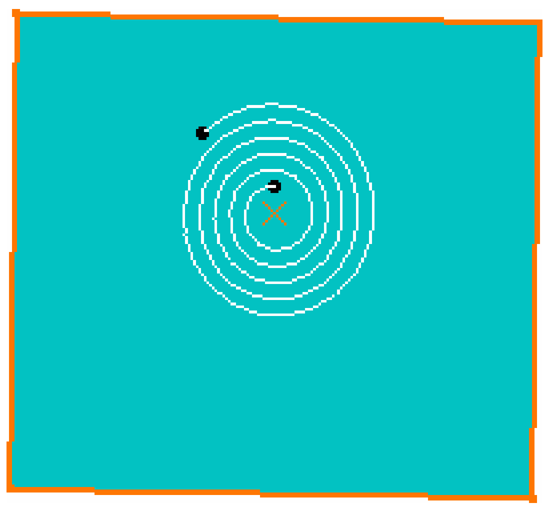


4. Specify the **Start radius** value, that is the distance from the **Center point**, along the **Reference direction**, at which the spiral's first revolution starts.

5. Define the spiral's **Orientation**, that is the rotation direction: clockwise or counter clockwise

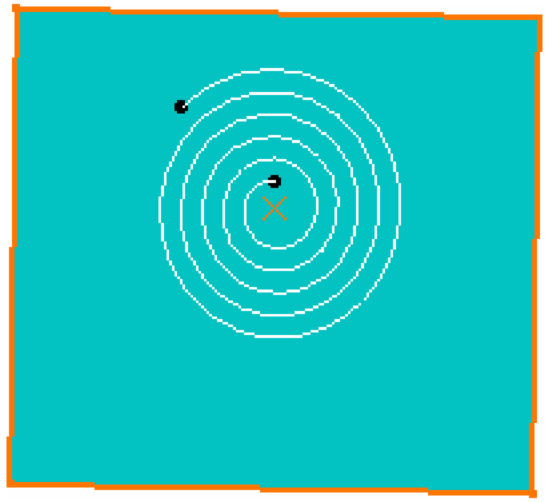
6. Specify the spiral creation mode, and fill in the corresponding values:

- **Angle & Radius**: the spiral is defined by a given **End angle** from the **Reference direction** and the radius value, the radius being comprised between the **Start** and **End radius**, on the first and last revolutions respectively (i.e. the last revolution ends on a point which distance from the center point is the **End radius** value).



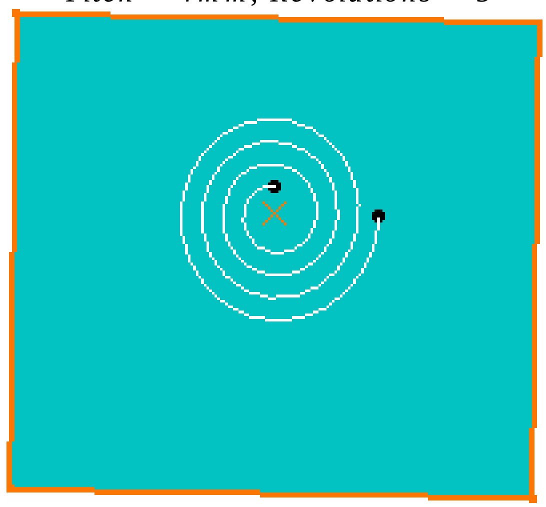
Ref. direction = Z, Start radius = 5mm, Angle = 45°, End radius = 20mm, Revolutions = 5

- **Angle & Pitch**: the spiral is defined by a given **End angle** from the **Reference direction** and the pitch, that is the distance between two revolutions of the spiral.



Ref. direction = Z, Start radius = 5mm, Angle = 45°, Pitch = 4mm, Revolutions = 5

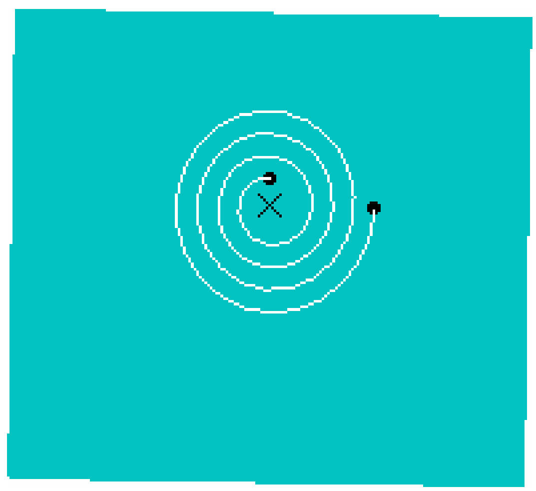
- **Radius & Pitch**: the spiral is defined by the **End radius** value and the pitch. The spiral ends when the distance from the center point to the spiral's last point equals the **End radius** value.



Ref. direction = Z, Start radius = 5mm, End radius = 20mm, Pitch = 4mm

Depending on the selected creation mode, the **End angle**, **End radius**, **Pitch**, and **Revolutions** fields are available or not.

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



The curve (identified as Spiral.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.

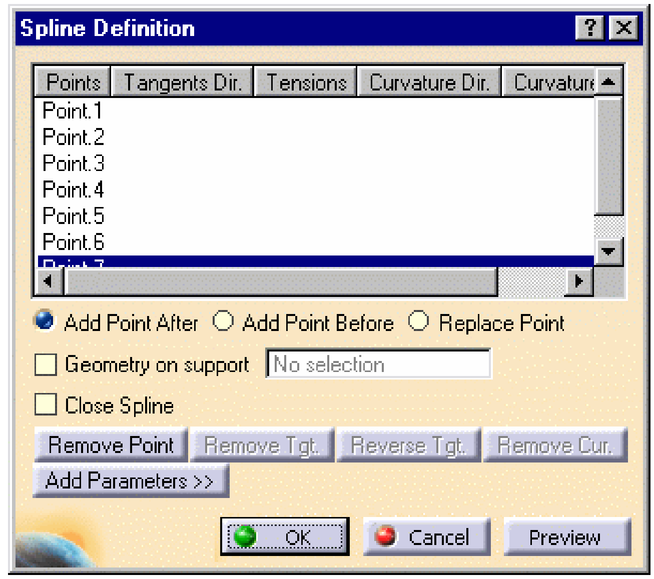
Creating Splines

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 be created.

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

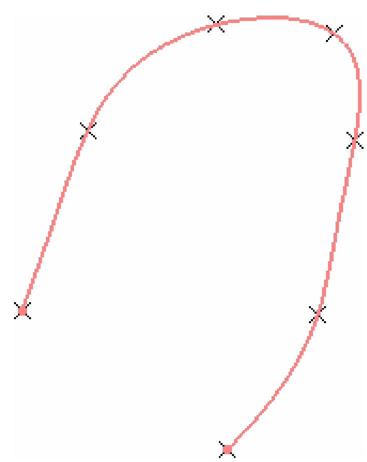
3. It is possible to edit the spline by first selecting a point in the dialog box list then choosing a button to either:

- 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

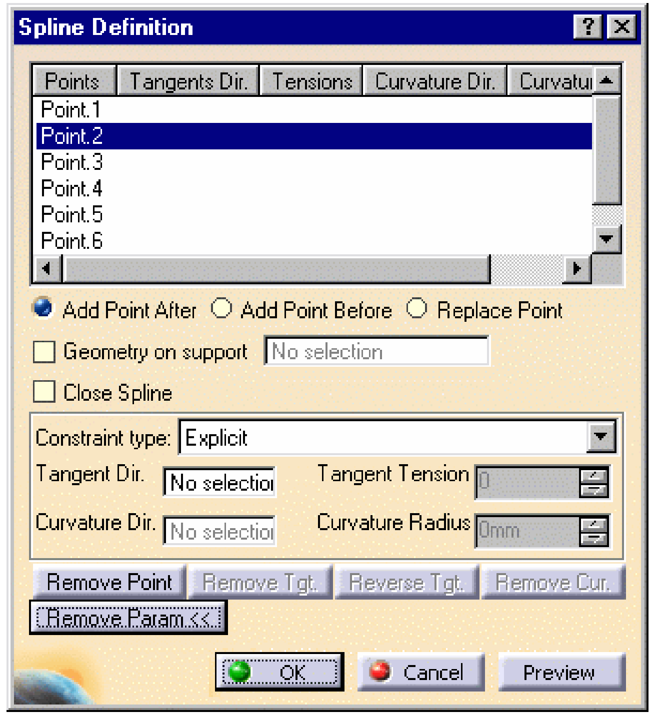
4. You can select the **Geometry on support** check box, and select a support (plane, surface), if you want the spline to be projected onto a support surface.

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.



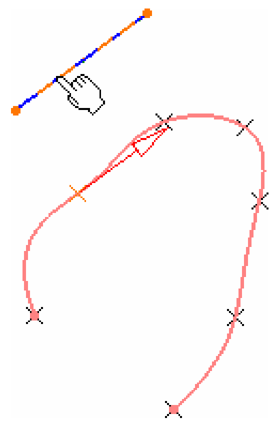
In the figure above, the spline was created on a planar support grid.



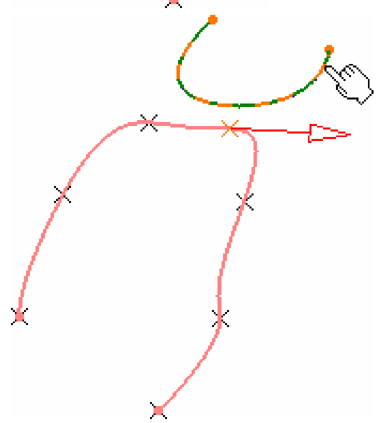
5. Click on the Add Parameter button to display further options.

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:

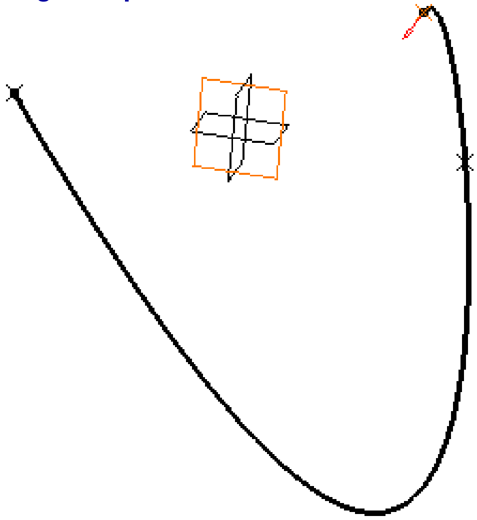


1. **Explicit**: select a line or plane to which the tangent on the spline is parallel at the selected point

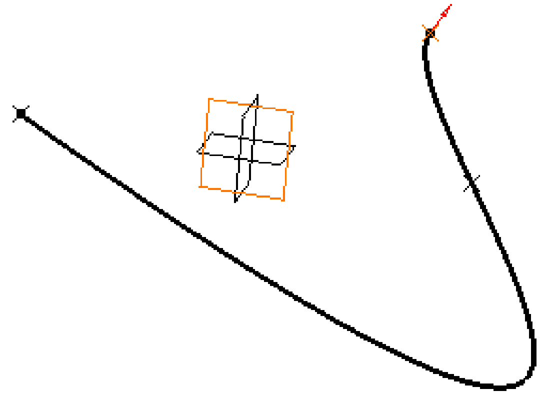


2. **From curve**: select a curve to which the spline is tangent at the selected point.

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



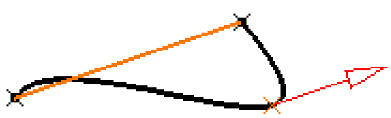
Spline with a tangency constraint on endpoint (tension = 2)



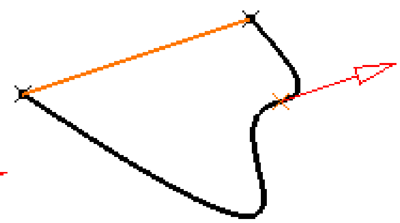
Spline with reversed tangent

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:

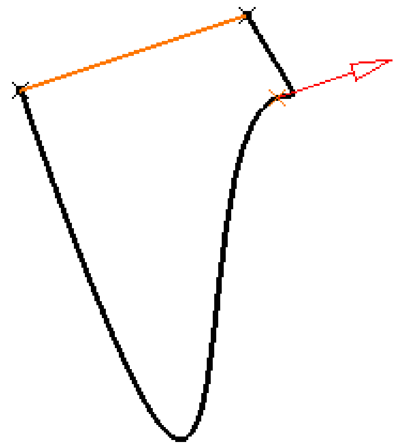
The curvature direction is projected onto a plane normal to the tangent direction.
If you use the **Create line** contextual menu, and want to select the same point as a point already used to define the tangent direction, you may have to select it from the specification tree, or use the pre-selection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

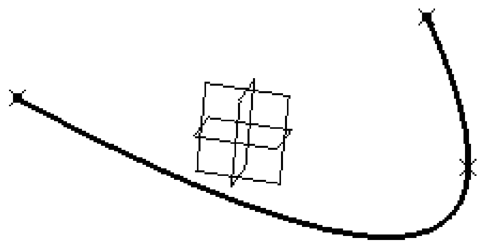
8. Click **OK** to create the spline.

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

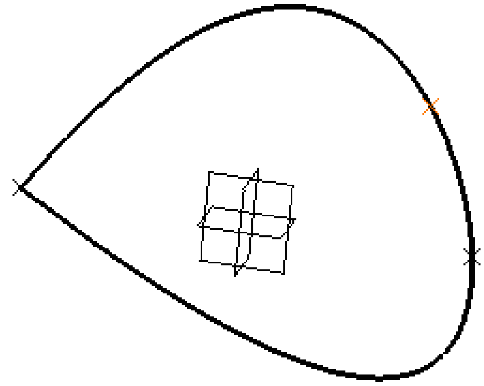
To add a parameter to a point, select a line in the Points list. This list is highlighted.
You have two possibilities:

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

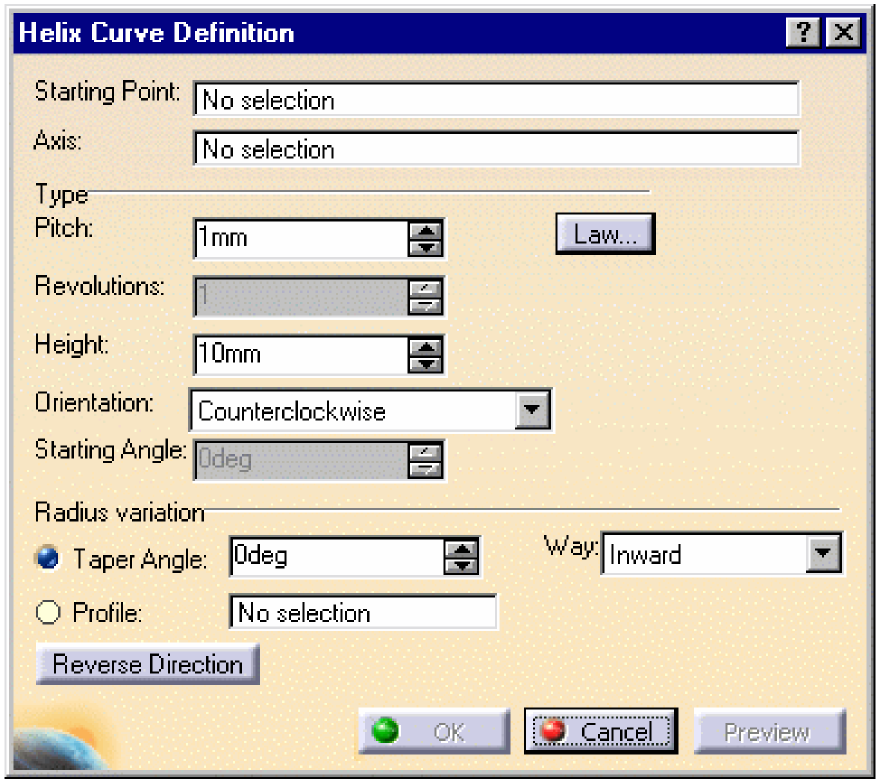
Creating a Helix

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

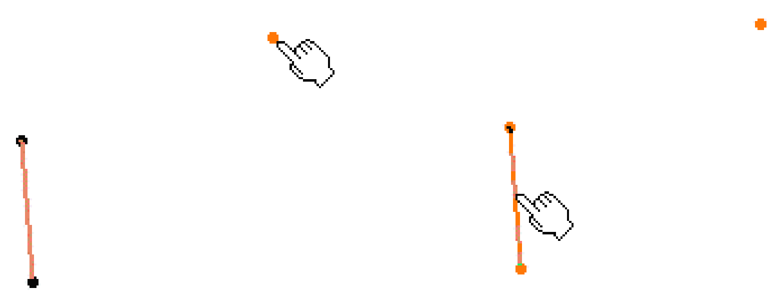
Open the Helix1.CATPart document.

1. Click the Helix icon .

The Helix Curve Definition dialog box appears.



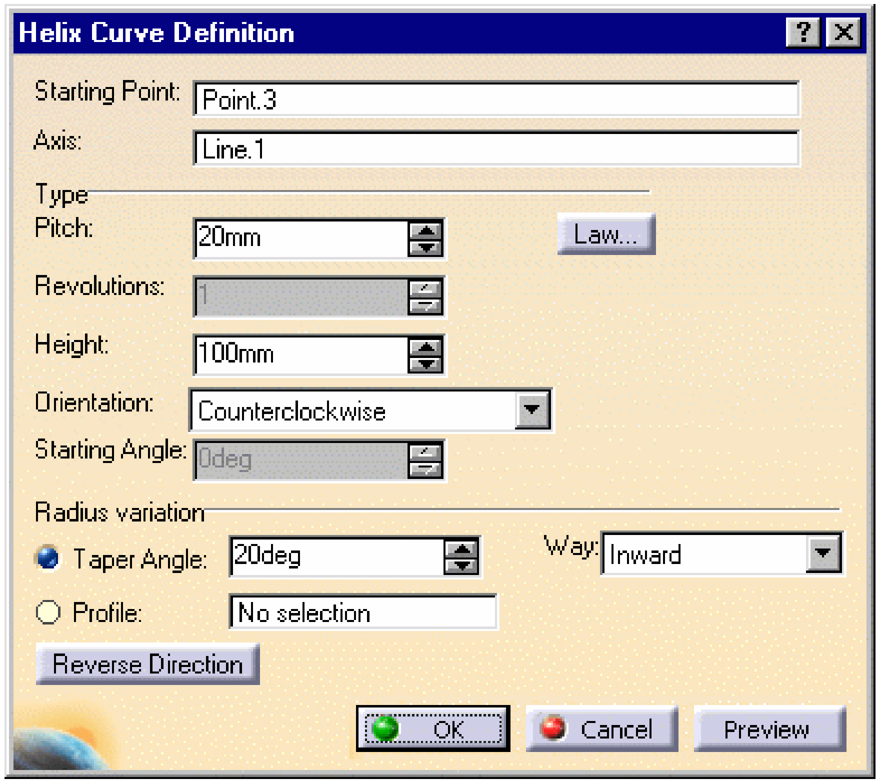
2. Select a starting point and an axis.



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.

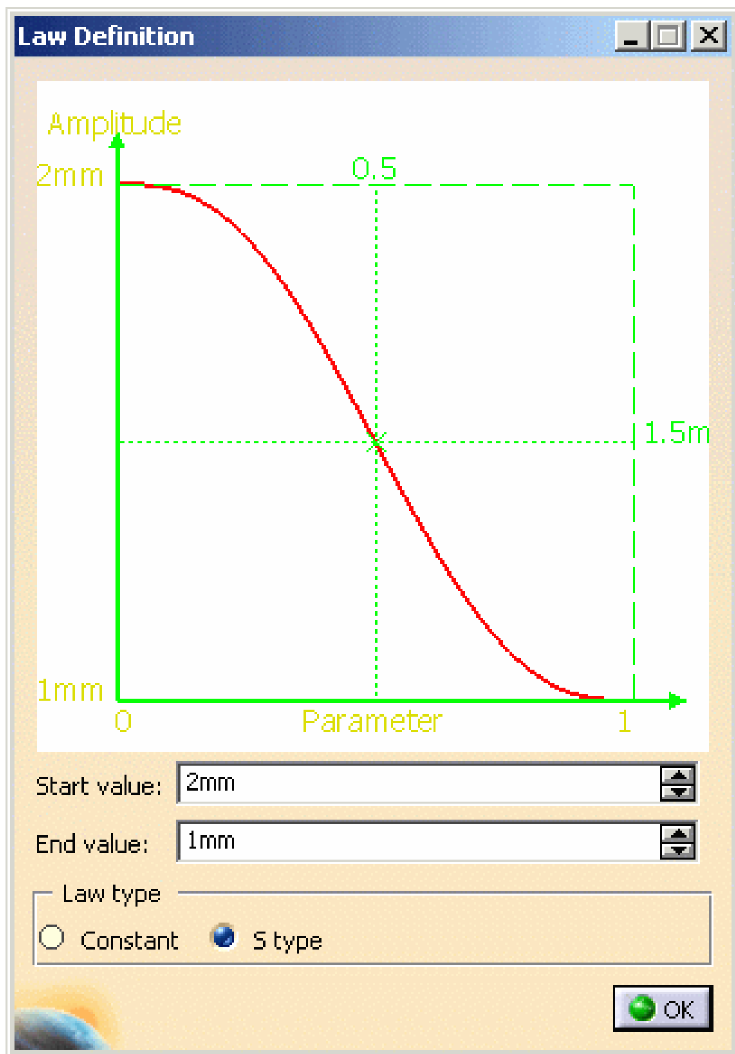


Defining Laws

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.

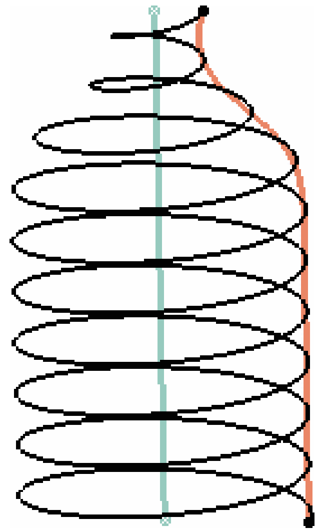
For the S type pitch, you need to define a second pitch value. The pitch distance will vary between these two pitch values, over the specified number of revolutions.



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)).

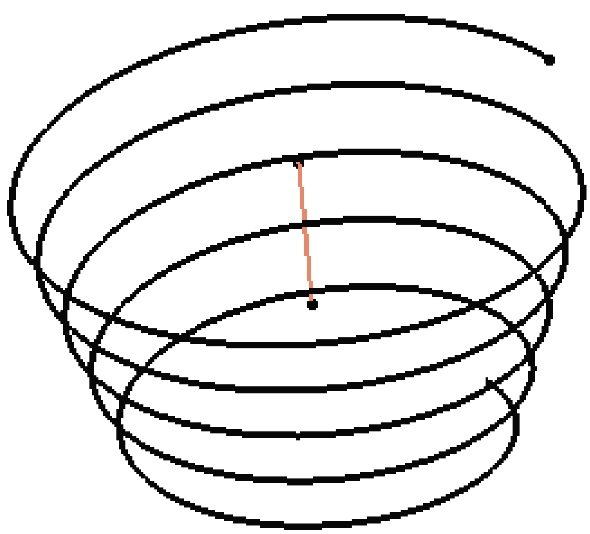
4. Click OK to return to the Helix Curve Definition dialog box.

- **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.

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.

Creating a Spine

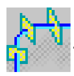
This command is only available with the Generative Shape Design 2 product.

This task shows how to create a spine, that is a curve normal to a list of ordered planes or planar curves. These spines are useful when creating complex surfaces such as swept, lofted, or filleted surfaces.

- Creating a Spine Based on Planes
- Creating a Spine Based on Guiding Curves
- Reversing the Spine's Starting Direction

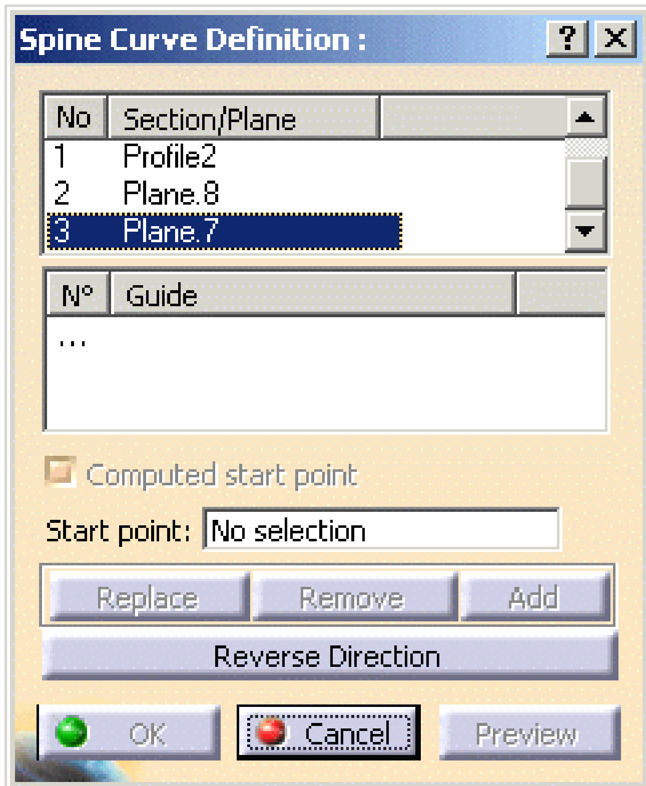
Creating a Spine Based on Planes

Open the Spine1.CATPart document.
Display the Curves toolbar by clicking and holding the arrow from the **Spine** icon.

1. Click the **Spine** icon .

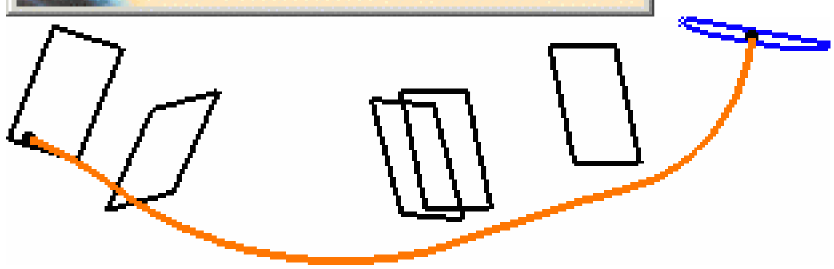
   The Spine Curve Definition dialog box is displayed.

2. Successively select planes or planar profiles.



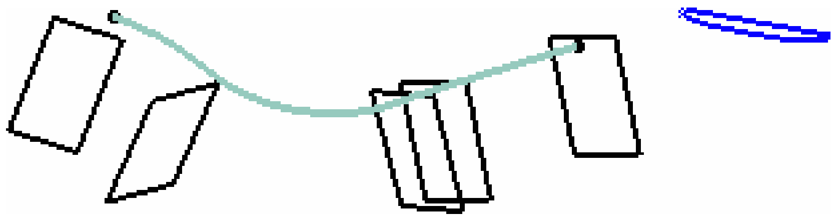
3. Click Preview.

   The spine is displayed.



4. You can also select a start point.

   The point is projected onto the first plane as the spine starting point, as illustrated here (point.3 is selected) except if it is already lying onto this first plane.

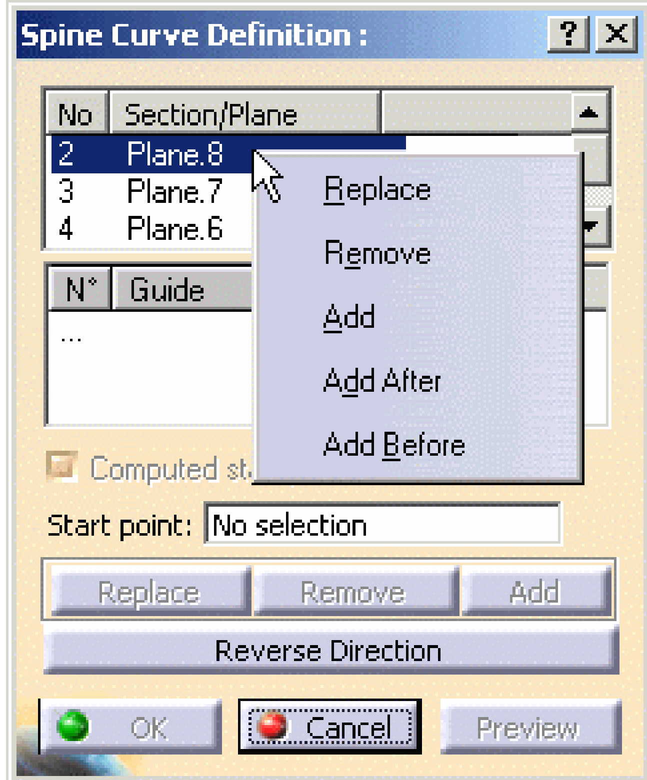


- Use the contextual menu on the Start point field to create a point. (See Stacking Commands).
- If you do not select a start point (default mode) one is computed automatically.
- To remove a selected point, check the **Computed start point** button.

5. Select one of the elements in the dialog box, then click:

- **Replace**, then select the replacing element in the geometry or the specification tree
- **Remove** to delete it from the spine definition
- **Add** then select a new element to be added after the last one.

Using the contextual menu, you can choose to **Add After** or **Add Before** the selected element.



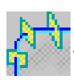
6. Click OK.

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

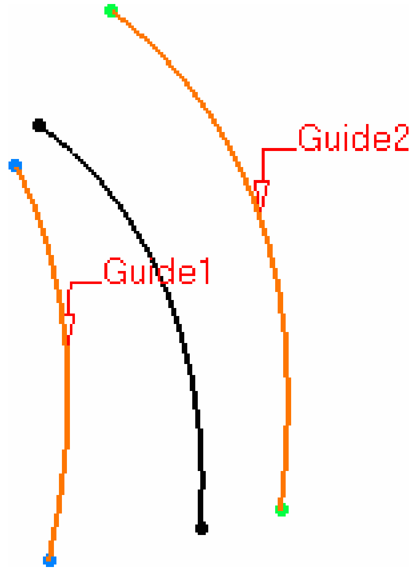
When non planar curves are selected, their mean planes are used to compute the spine.

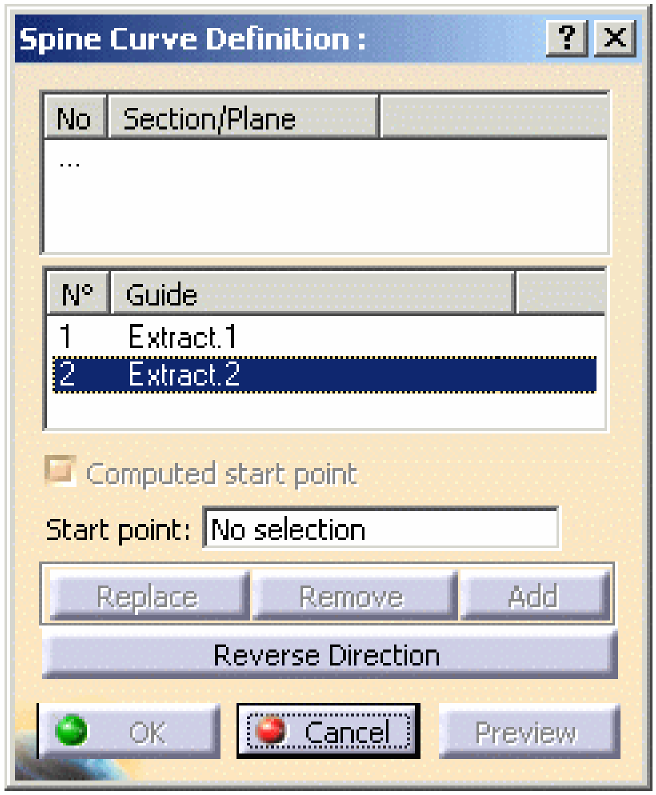
Creating a Spine Based on Guiding Curves

Open the Spine2.CATPart document.

1. Click the **Spine** icon .
   The Spine Curve Definition dialog box is displayed.
2. Click within the Guide list and successively select two guiding curves.

The spine is immediately previewed.

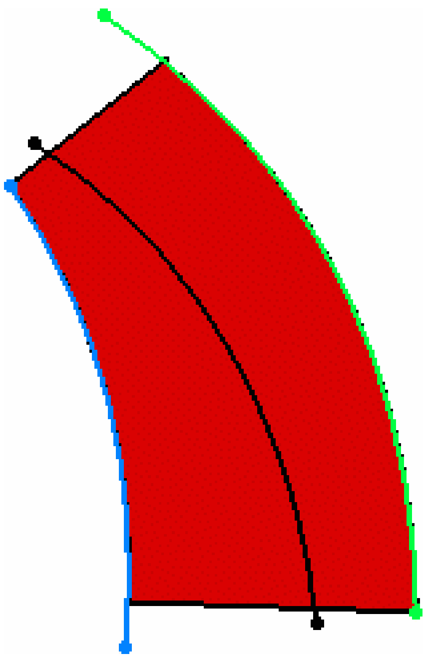




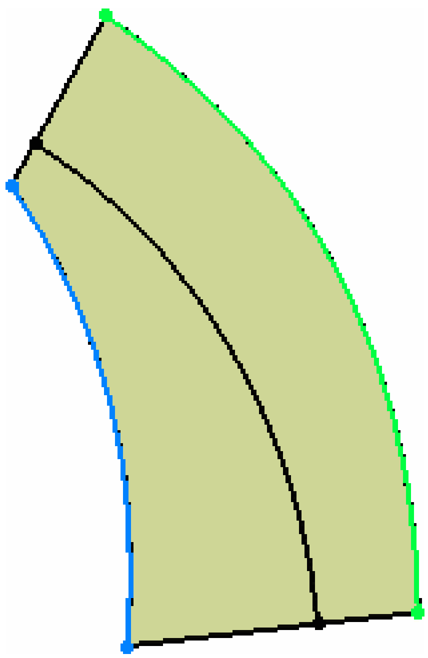
3. Click OK to create the spine.

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

This type of spine is very useful when creating a swept surface, as illustrated below:



Swept surface without any spine

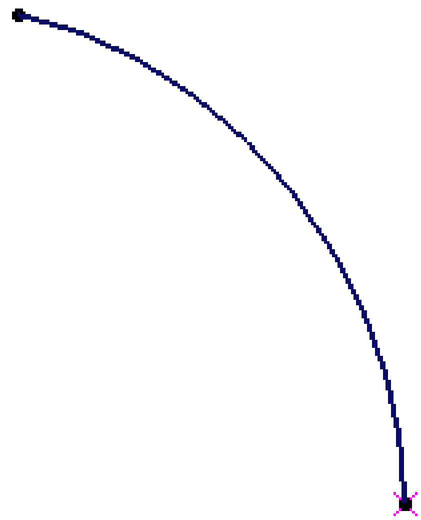


Swept surface with specified spine

Reversing the Spine's Starting Direction

This option can only be used if the first profile is a plane.

Open the Spine3.CATPart document.

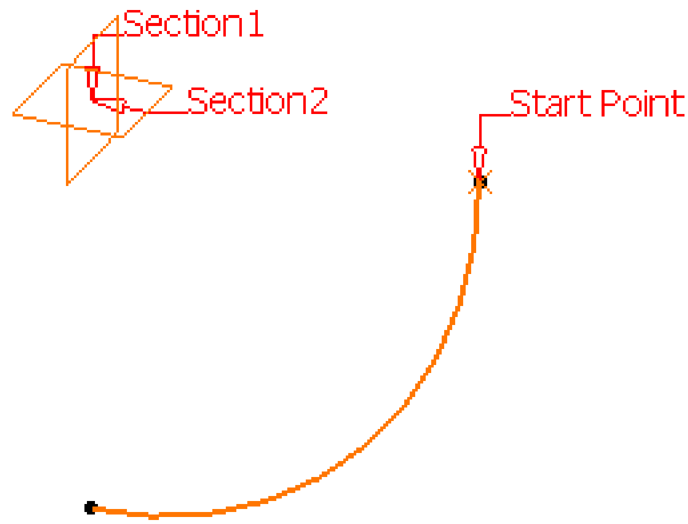


1. Double-click the spine.

   The Spine Curve Definition dialog box is displayed.

2. Click the **Reverse Direction** button.

3. Click OK to create the reverse spine.



Creating Corners

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

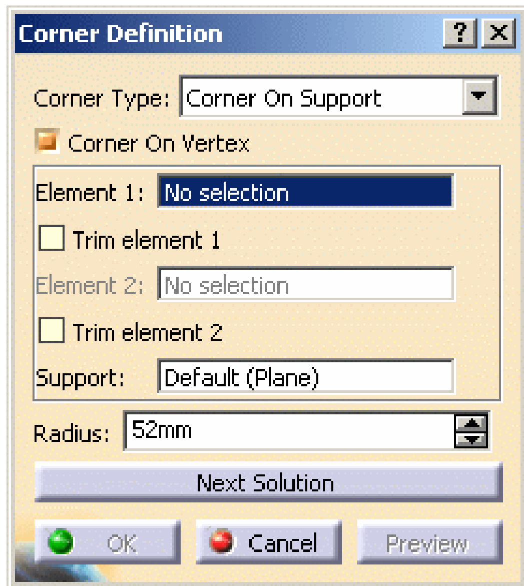
Open the Corner1.CATPart document.

1. Click the **Corner** icon.

The Corner Definition dialog box appears.

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).
It is checked by default.

2. Choose the **Corner Type**:

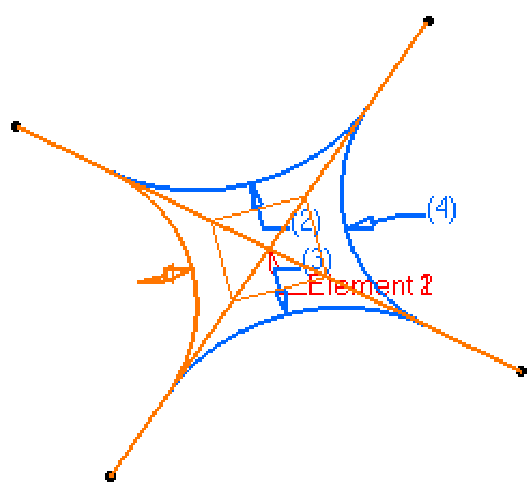


Corner on Support:

the support can be a surface or a plane

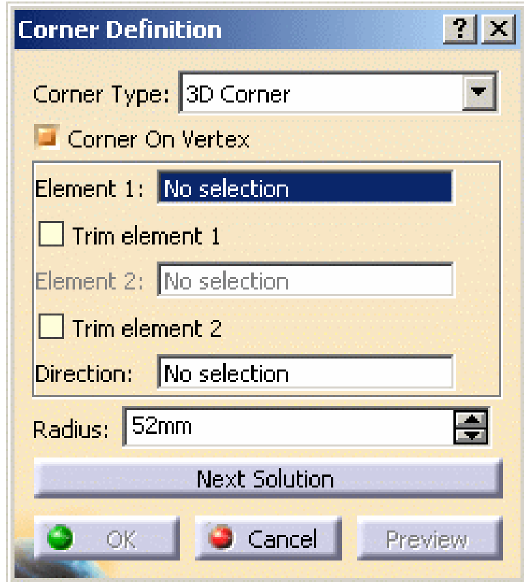
1. Select a curve or a point as first reference element.
2. Select a curve as second reference element.
   The corner will be created between these two references.
3. Select the **Support** surface.
   Here we selected the zx plane.

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.

3D Corner:



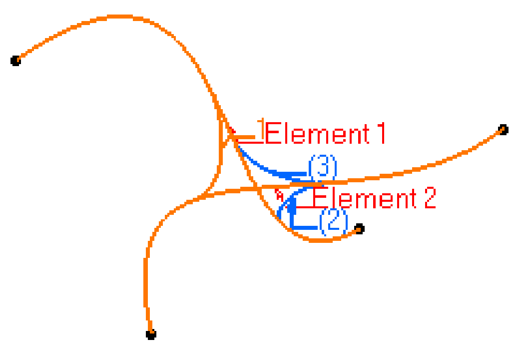
corner between two 3D curves.

1. Select a 3D curve or a point as first reference element.
2. Select a 3D curve as second reference element.

   The corner will be created between these two references.
3. Select a **Direction**.

   Here we selected Line.2.

The resulting corner is a 3D curve seen as an arc of circle along the user input Direction.



The input elements must not be collinear to the 3D Corner direction. Moreover, if the plane projection of an input element along the user input direction is singular or is self intersected, some corner solution might not be computed.

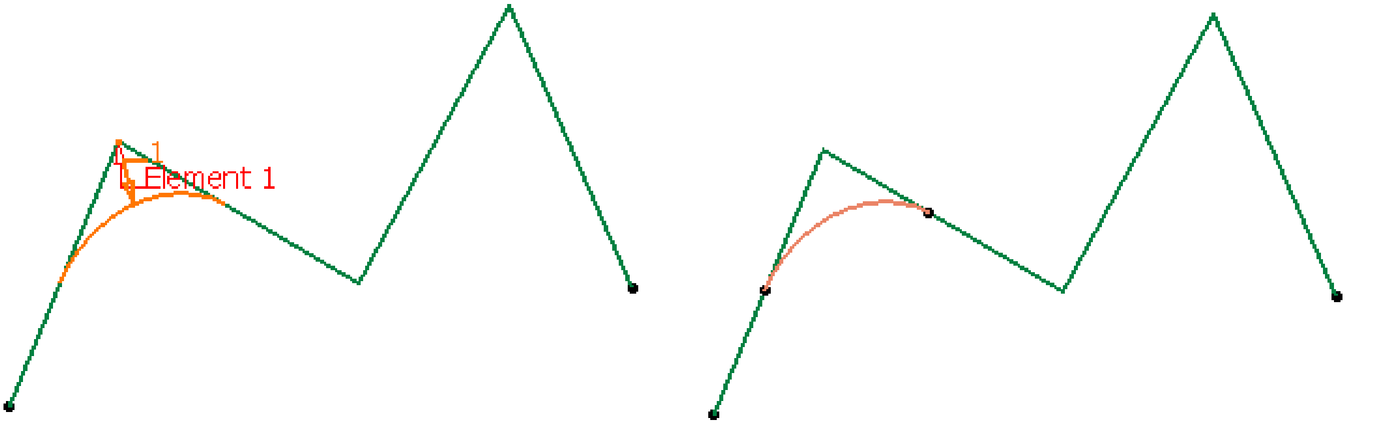
3. Enter a **Radius** value.

In the case of a curve as Element 1, note that:

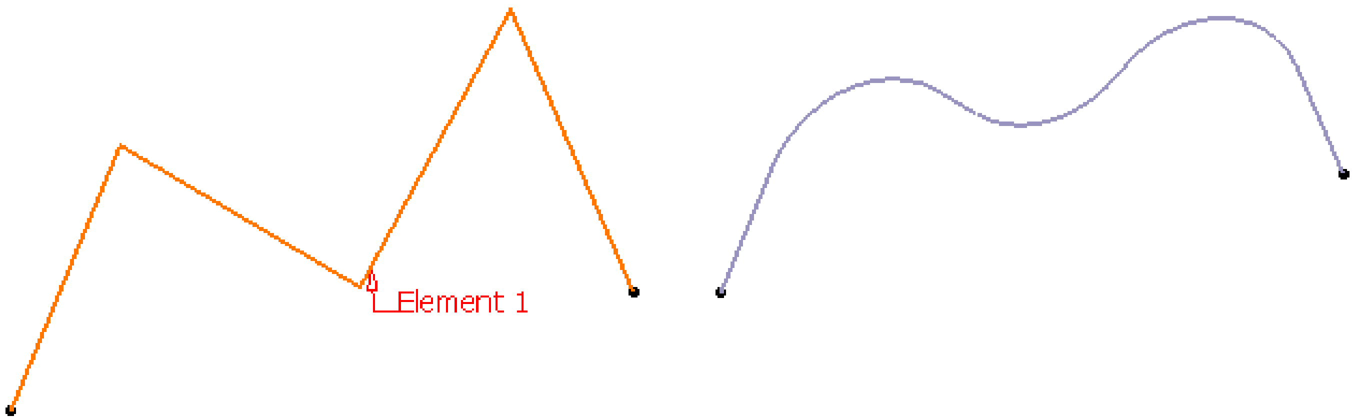
- all corners have the same radius
- closed wires can be selected

4. Select a direction or a support depending on the corner type you chose.

In case there are several solutions, the corner closest to the selected point will be created.

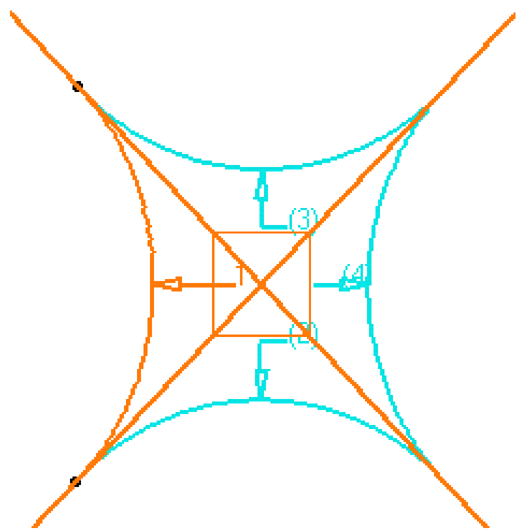


The example above shows a corner defined by a point as Element 1

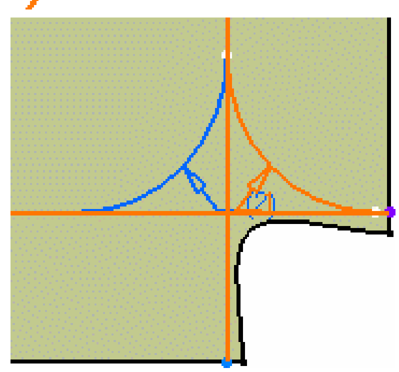


The example above shows a corner defined by a curve as Element 1

5. 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).

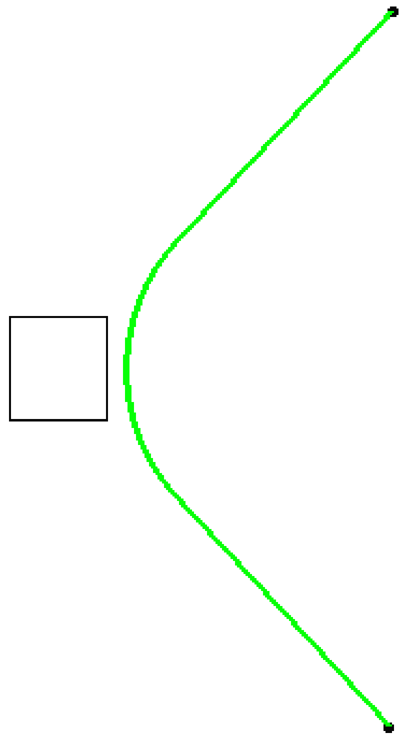


6. You can select the **Trim element** check buttons if you want to trim and assemble the two reference elements to the corner.

The elements can be trimmed and assembled individually.

7. Click OK to create the corner.

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

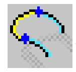


- When the selected curves are coplanar, the default support is the background plane. However, you can explicitly select any support.
- When the selected curves are not coplanar, an implicit plane can be created, provided the curves intersect and are locally coplanar at this intersection. However, you can explicitly select any support.
- You can edit the corner's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

Creating Connect Curves

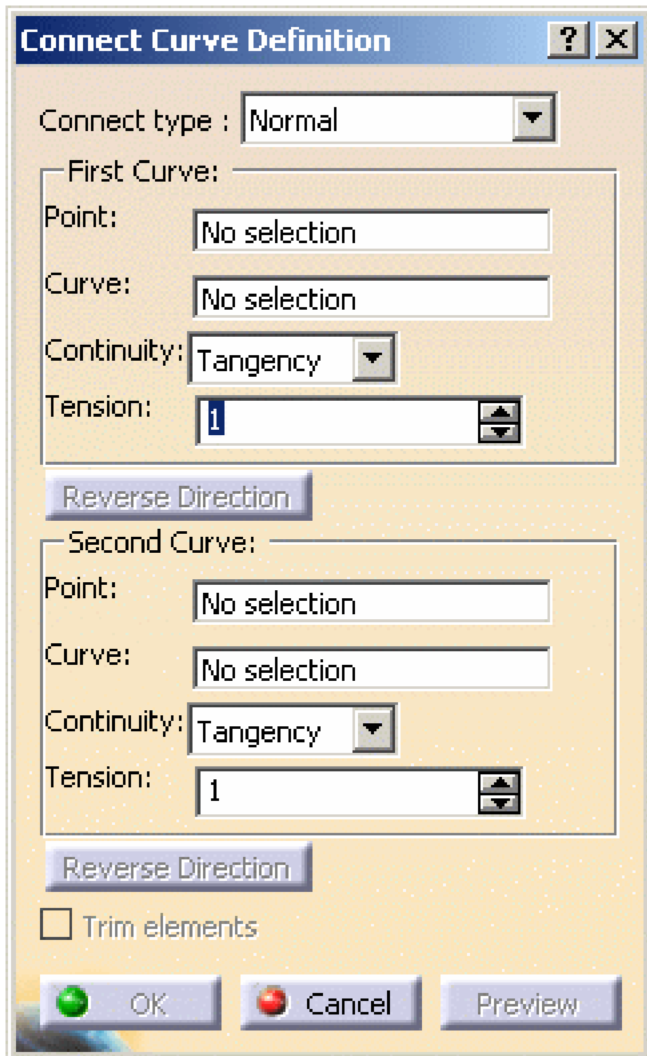
This task shows how to create a connecting curve between two curves.

Open the Connect1.CATPart document.

1. Click the **Connect Curve** icon .

   The Connect Curve Definition dialog box appears.

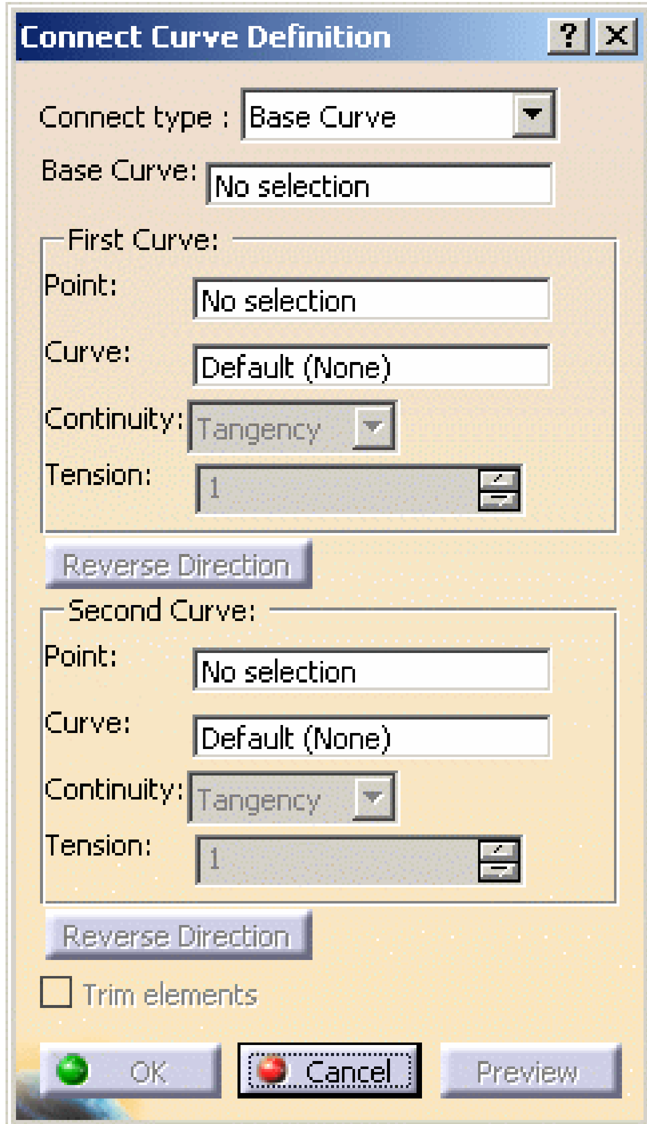
2. Select the **Connect type**.



Normal

1. Select a first **Point** on a curve then a second **Point** on a second curve.
   The **Curve** fields are automatically filled.

Base Curve



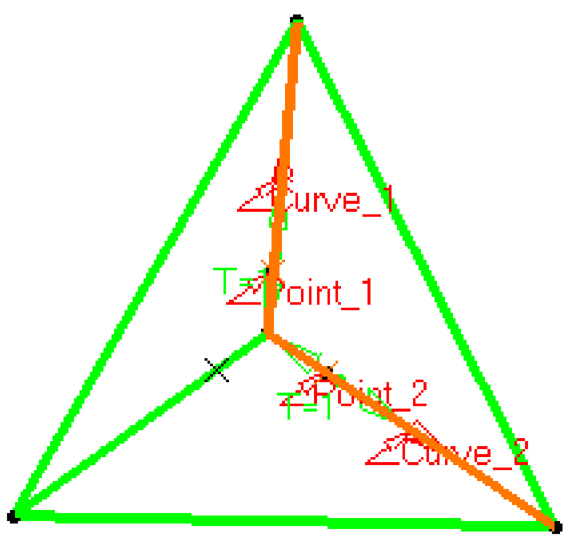
1. Select a base curve as the curve reference.
   The orientation of the connect curve will be the orientation of the base curve.
2. Select a first **Point** on a curve then a second **Point** on a second curve.

The support **Curve** is optional (it is set as **Default**)

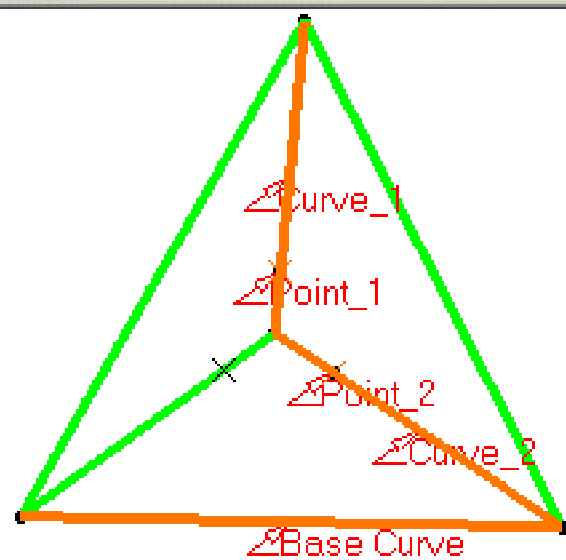
The first point can be either on the base curve or on the support curve.

The Base Curve option is useful when creating several profiles or guides that have the same shape.

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



Normal curve



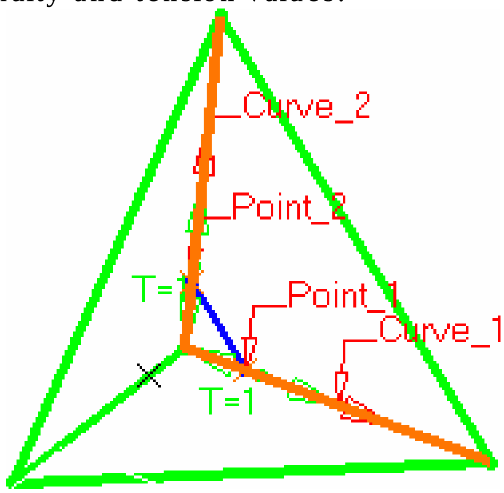
Base Curve

3. Use the combos to specify the desired **Continuity** type: **Point**, **Tangency** or **Curvature**.

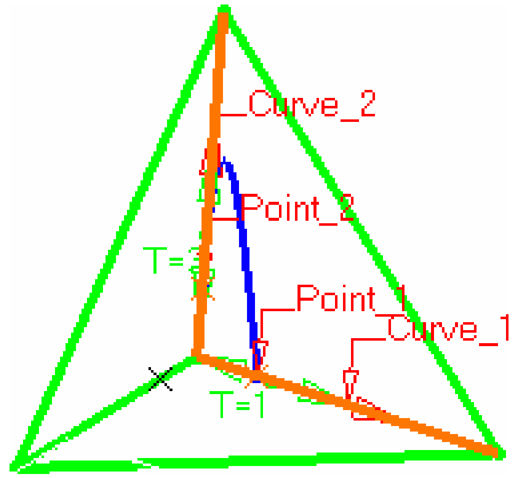
4. If needed, enter tension values.

If the **Base Curve** type is selected, the **Continuity** and **Tension** options for the first and the second curve are grayed out and set to Tangency and 1 respectively.

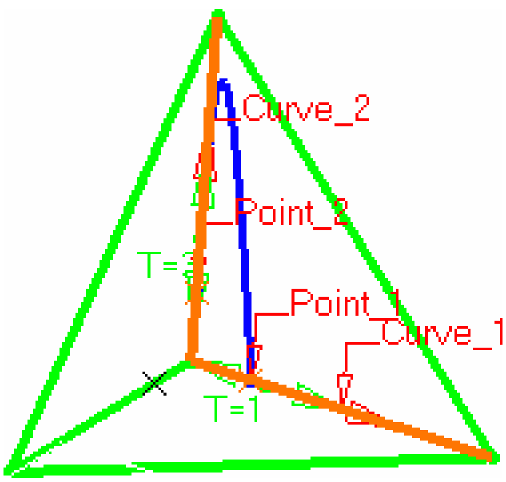
The connect curve is displayed between the two selected points according to the specified continuity and tension values.



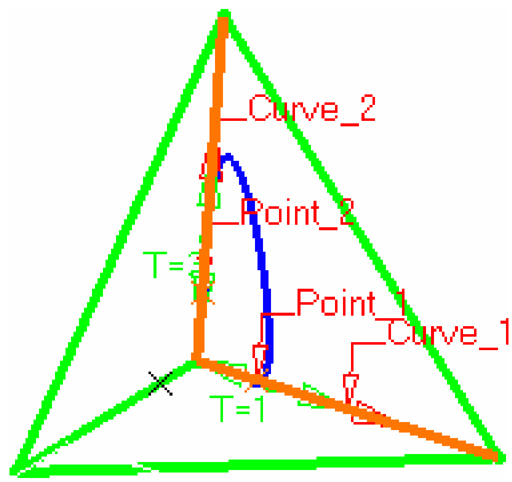
Normal curve with point continuity at both points



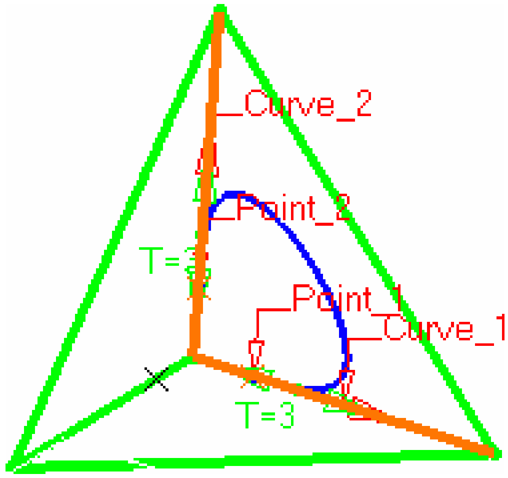
Normal curve with point continuity at one point and tangent continuity at the other



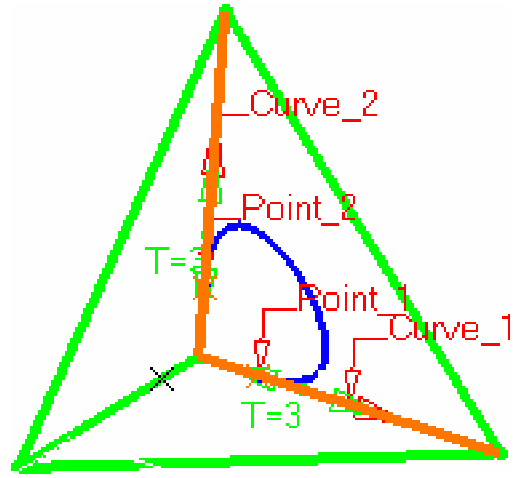
Normal curve with point continuity at one point and curvature continuity at the other



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



Normal curve with curvature continuity at both points



Normal curve with tangent continuity at both points

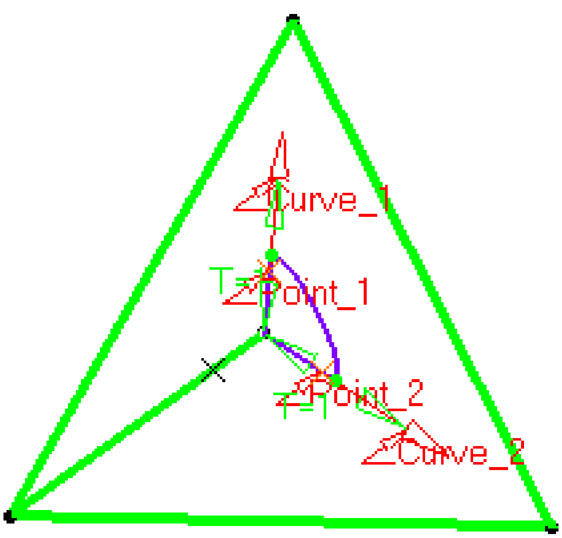
5. An arrow is displayed at each extremity of the curve. You can click the arrow to reverse the orientation of the curve at that extremity or click the **Reverse** button..

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

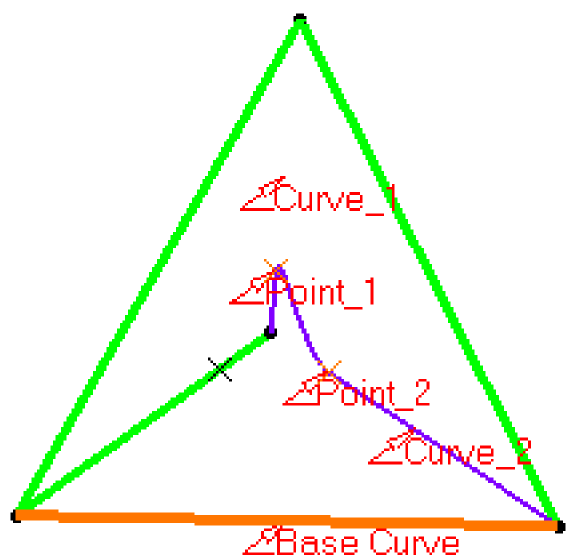
If the **Base Curve** type is selected, the **Reverse Direction** options are grayed out.

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

If no **Base Curve** type is selected, the **Trim** option is grayed out.



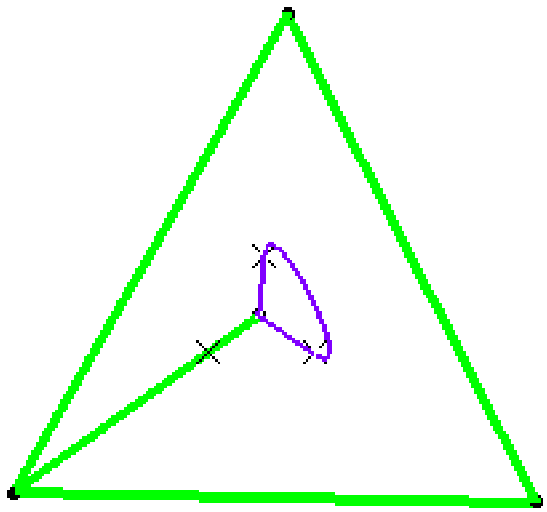
Normal curve



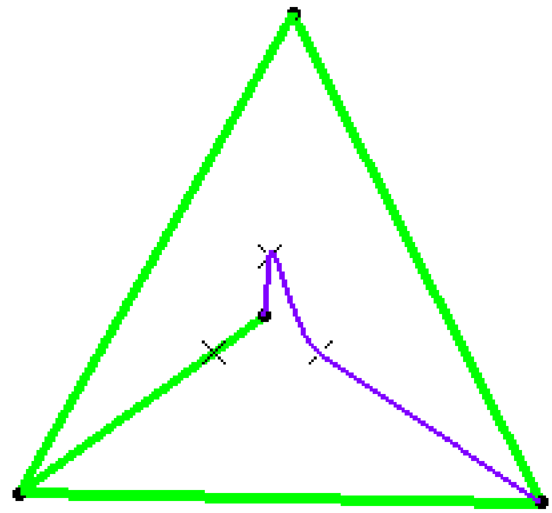
Base Curve

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

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



Normal curve



Base Curve

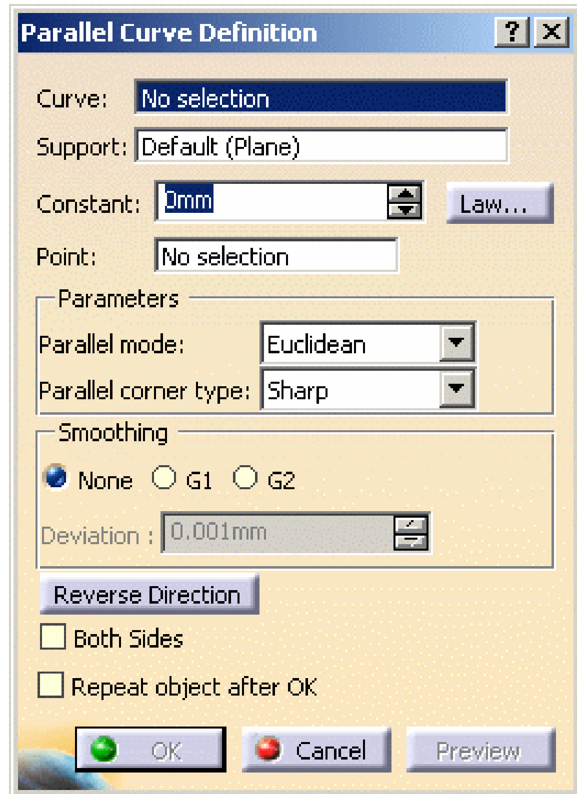
Creating Parallel Curves

This task shows you how to create a curve that is parallel to a reference curve.

Open the ParallelCurves1.CATPart document.

1. Click the **Parallel Curve** icon .

The Parallel Curve Definition dialog box appears.

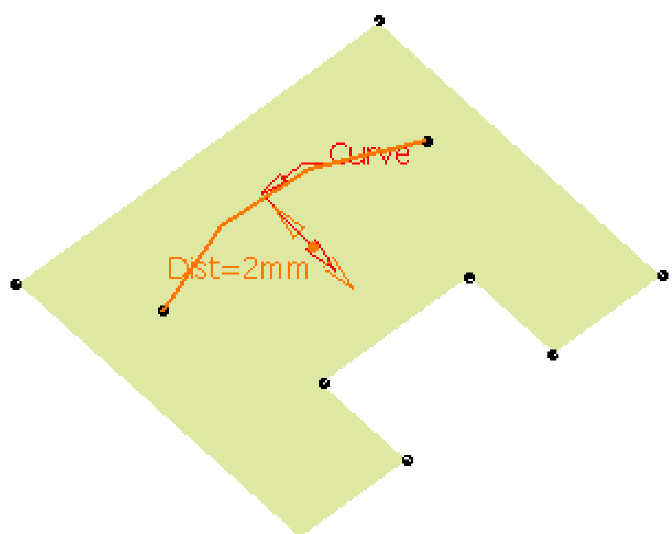


2. Select the reference **Curve** to be offset.

3. Select the **Support** plane or surface on which the reference curve lies.

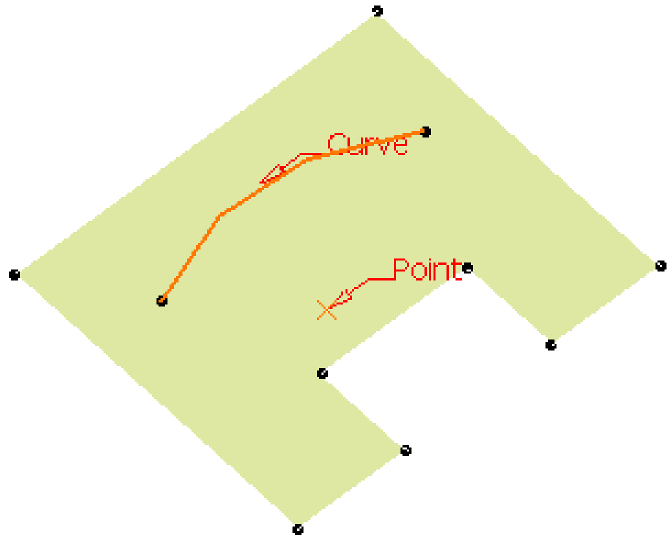
4. Specify the offset of the parallel curve either by:

   a. entering a value or using the graphic manipulator in the **Constant** field.



b. selecting a point in the **Point** field (in both Geodesic and Euclidean mode)

In that case, the **Constant** field is grayed.



5. Choose the parallelism mode to create the parallel curve:

- **Euclidean**: the distance between both curves will be the shortest possible one, regardless of the support.

If you select this mode, you can choose to offset the curve at a constant distance from the initial element, or according to a law. In this case, you need to select a law as defined in Creating Laws.

Defining Laws

The law can be negative, providing the curves are curvature continuous.
Please note that:

- it is advised to use curvature continuous laws,
- it is advised not to use laws with a vertical tangency,
- it is possible to create a parallel curve with a law that reverses (which means becoming either positive or negative) **only** on a curve that is tangency continuous.

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

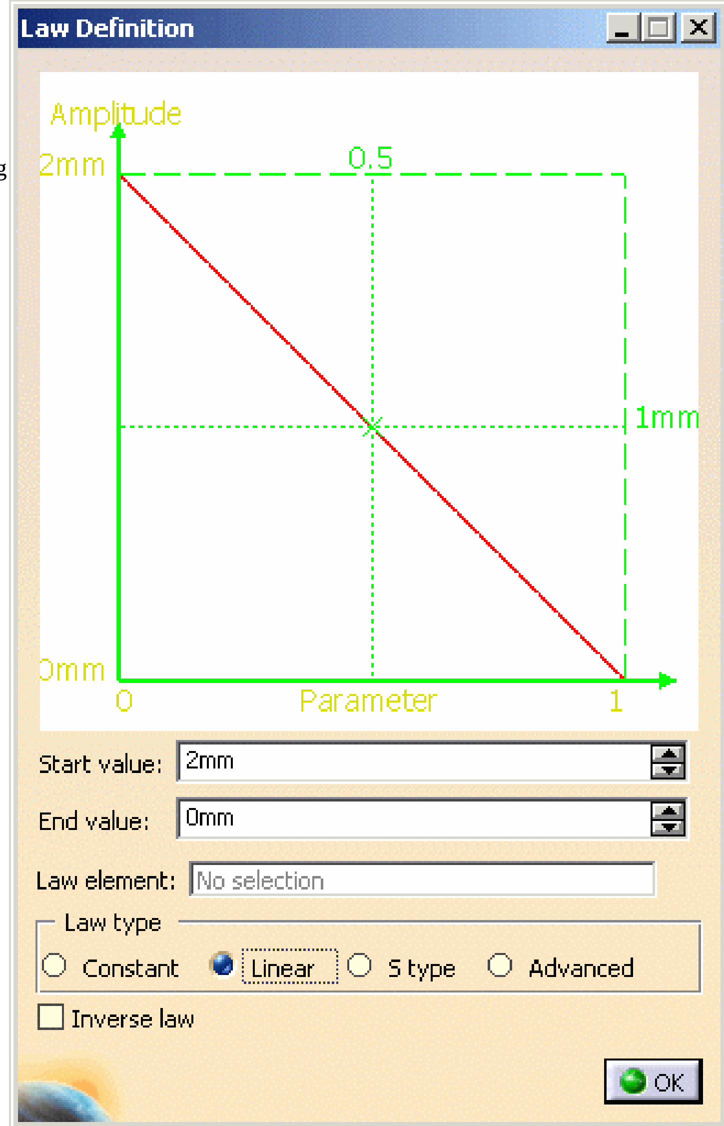
1. Click the **Law...** button to display the Law Definition dialog box. The 2D viewer enables you to previsualize the law evolution before applying it.
2. Enter **Start** and **End values**.
3. Choose the law type to be applied to the pitch.
   Four law types are available:

   a. **Constant**: a regular law, only one value is needed.
   b. **Linear**: a linear progression law between the **Start** and **End** indicated values
   c. **S type**: an S-shaped law between the two indicated values
   d. **Advanced**: allowing to select a Law element as defined in Creating Laws.

   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.

4. 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)).

5. Check the **Inverse law** button to reverse the law as defined using the above options.
6. Click **OK** to return to the Parallel Curve Definition dialog box.

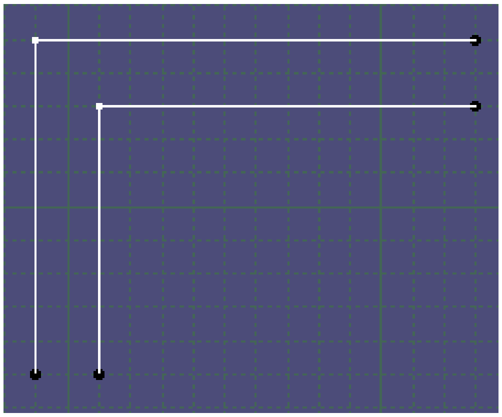
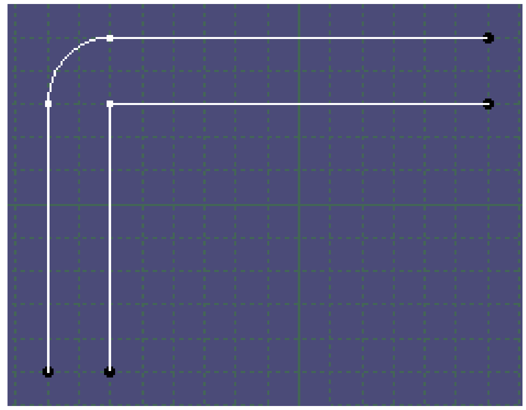


- **Geodesic**: the distance between both curves will be the shortest possible one, taking the support curvature into account.
  In this case, the offset always is constant in every points of the curves and you do not need to select a corner type.

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

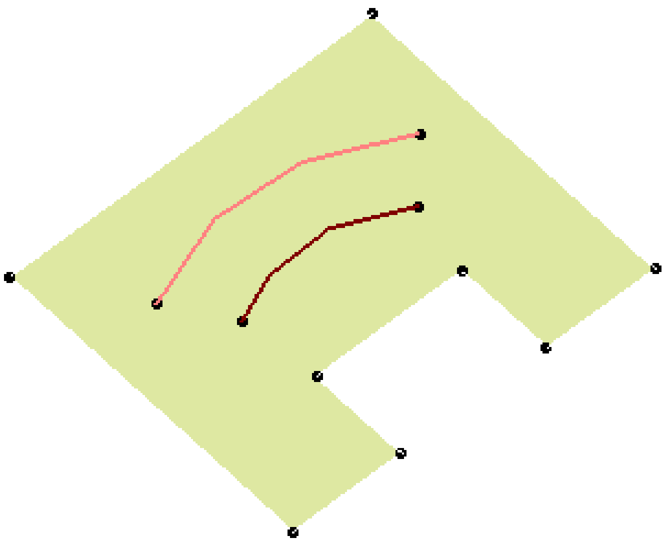
6. Select **corner type** (useful for curves presenting sharp angles):

- **Sharp**: the parallel curve takes into account the angle in the initial curve
- **Round**: the parallel curve is rounded off as in a corner
  In this case, the offset always is constant in every points of the curves and you do not need to select a corner type.

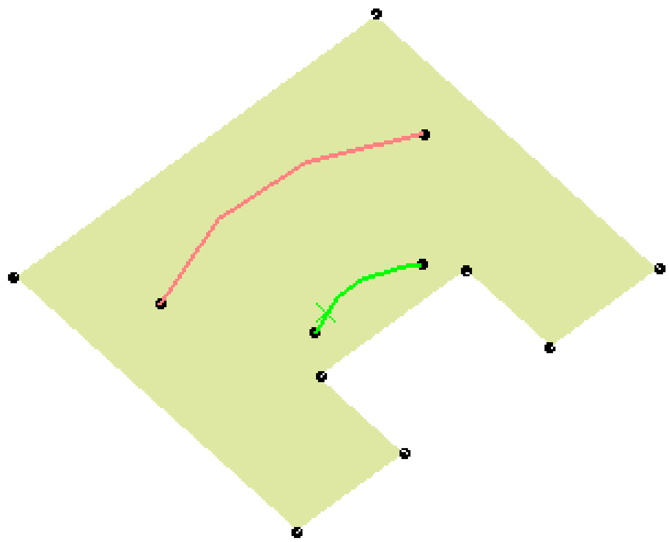



The parallel curve is displayed on the support surface and normal to the reference curve.

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



Parallel curve defined by an constant offset value

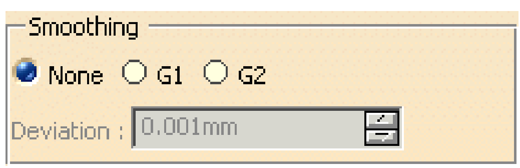


Parallel curve defined by a passing point

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

- You can smooth the curve 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.



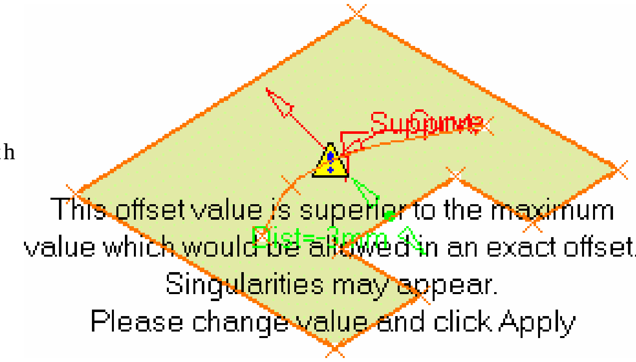
If the curve cannot be smoothed correctly, a warning message is issued.

Moreover, a topology simplification is automatically performed for G2 vertices: cells with a curvature continuity are merged.

Please refer to the Customizing General Settings chapter.

The Smoothing option is only available with the Generative Shape Design 2 product.

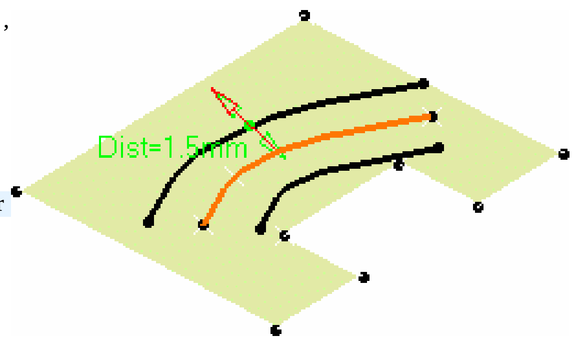
- You can use the **Reverse Direction** button to display the parallel curve on the other side of the reference curve or click the arrow directly on the geometry.
- When the selected curve is a planar curve, its plane is selected by default. However, you can explicitly select any support.
- when you modify an input value through the dialog box, such as the offset value or the direction, the result is computed only when you click on the Preview or OK buttons.
- 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.



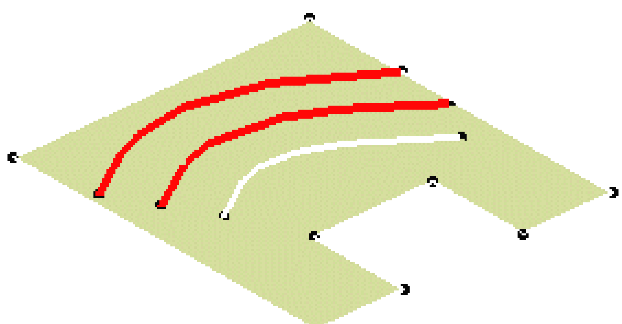
- Check the **Both Sides** button to create two parallel curves, symmetrically in relation to the selected curve, and provided it is compatible with the initial curve's curvature radius.

The second parallel curve has the same offset value as the first parallel curve. In that case it appears as aggregated under the first element.
Therefore both parallel curves can only be edited together and the aggregated element alone cannot be deleted.

If you use the Datum mode, the second parallel is not aggregated under the first one, but two datum elements are created.



- Use the **Repeat object after OK** checkbox to create several parallel curves, each separated from the initial curve 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.



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

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

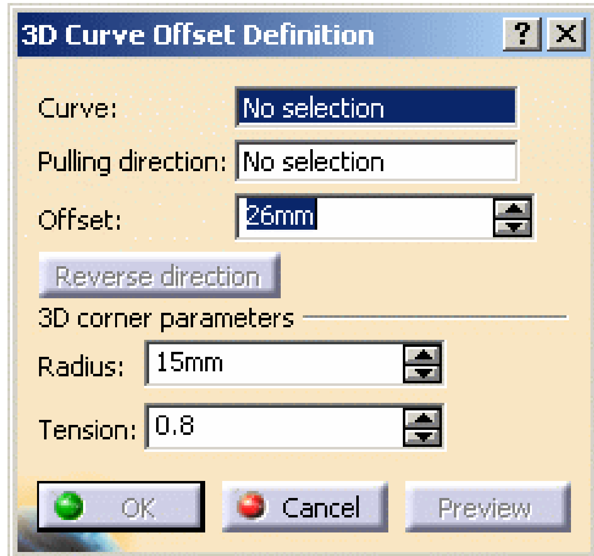
Creating a 3D Curve Offset

This task shows you how to create a 3D curve that is offset from a reference curve.

Open the 3DCurveOffset1.CATPart document.

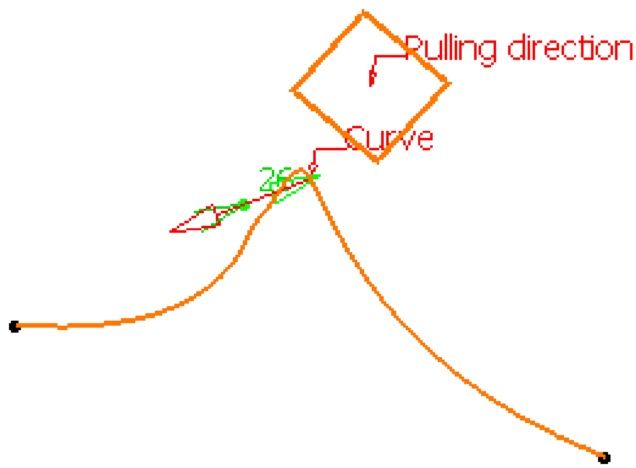
1. Click the **3D Curve Offset** icon .

The 3D Curve Offset Definition dialog box appears.



2. Select the reference **Curve** to be offset.
3. Select the offset **Pulling Direction**.

   A direction arrow appears in the 3D geometry and lets you change the direction.

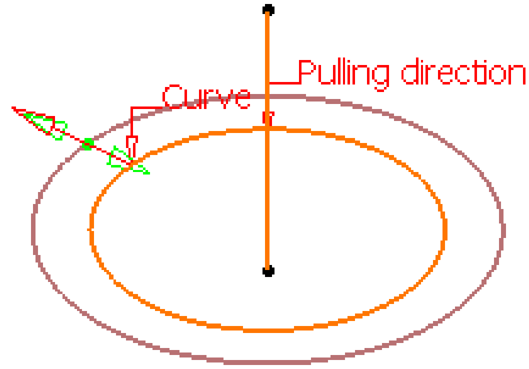


The pulling direction corresponds to a draft direction. It does not correspond to the direction of the 3D curve.
If you select a plane as the pulling direction, the latter is normal to the plane.

4. Specify the **Offset** value by entering a value or using the spinners.
5. Click **Preview** to see the offset curve.
6. Click **OK**.

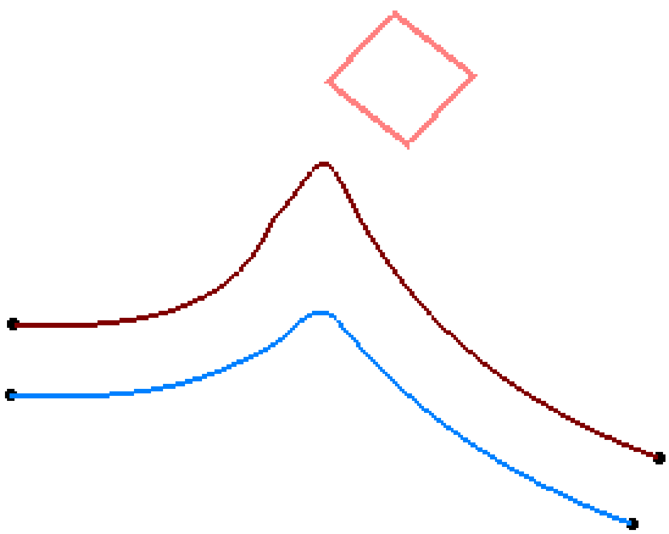
The curve (identified as 3D curve offset.xxx) is added to the specification tree.

Here is the case of a closed curve:

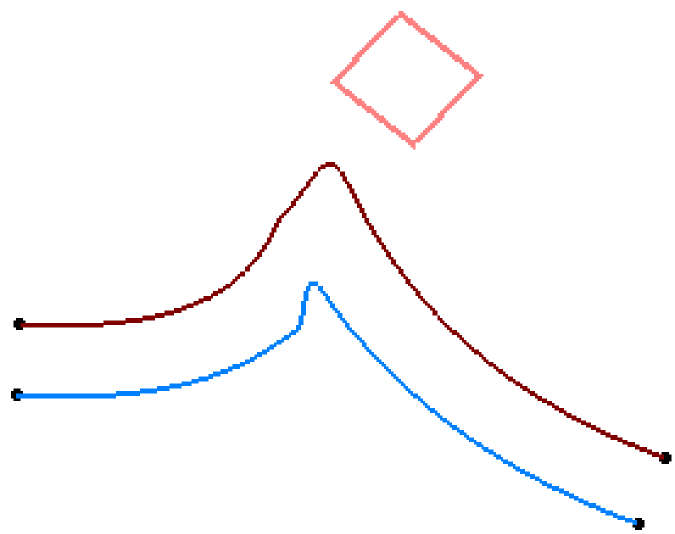


3D corner parameters enable to manage singularities:

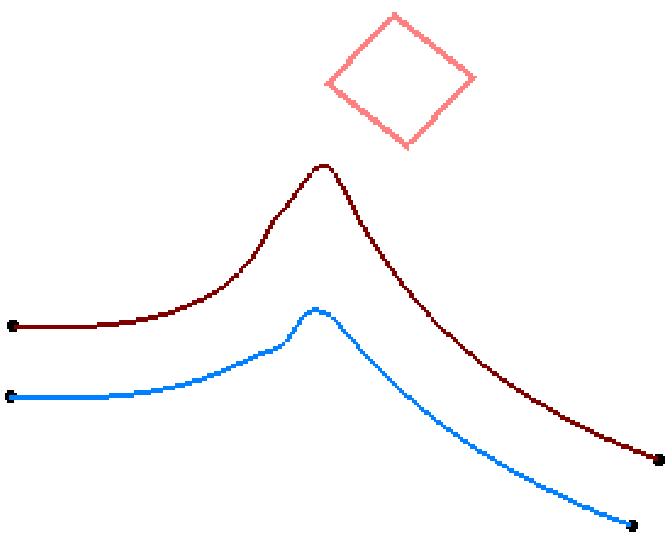
- **Radius**: if the curvature radius of the input curve is smaller than the offset value, 3D corner curves are created to fill the holes
- **Tension**: for the 3D corner curves, if needed



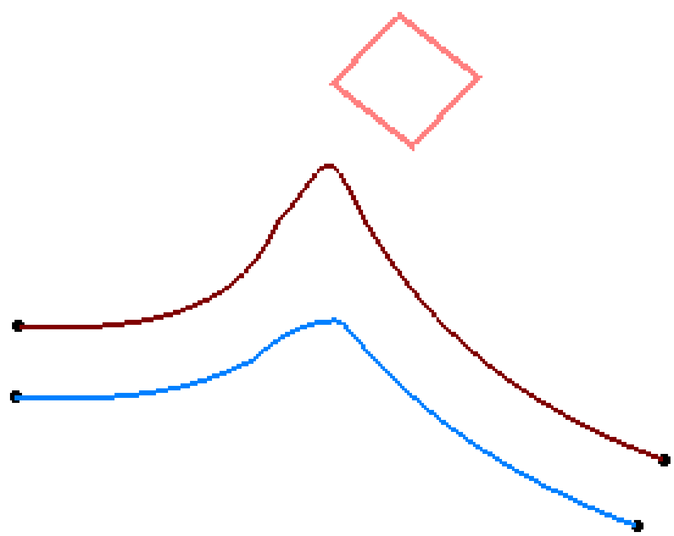
3D Curve Offset with a radius of 20 mm and a tension of 1



3D Curve Offset with a radius of 5 mm and a tension of 1



3D Curve Offset with a tension of 2 and a radius of 20 mm



3D Curve Offset with a tension of 0.3 and a radius of 20 mm

The input curve must be tangency continuous and must not be collinear to the offset direction.

Creating Projections

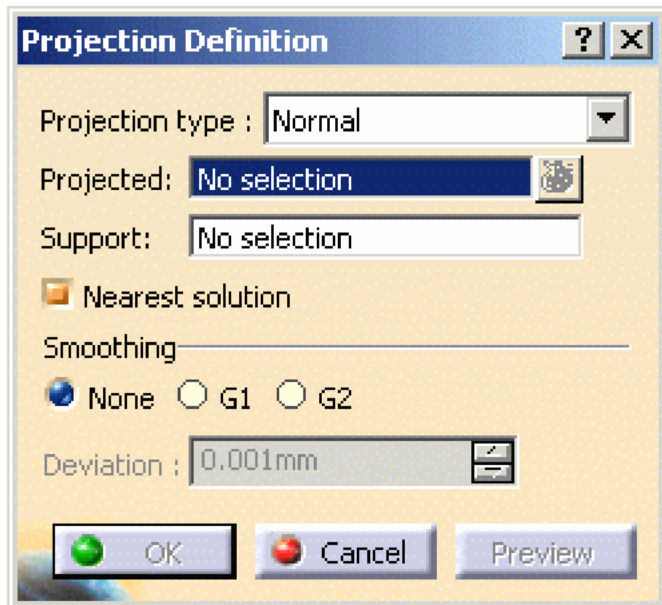
This task shows you how to create geometry by projecting one or more elements onto a support. The projection may be normal or along a direction.
You can project:

- 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.

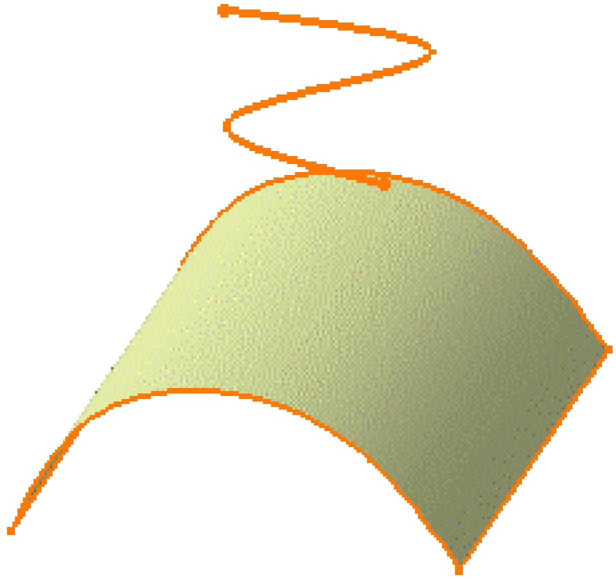
1. Click the **Projection** icon.



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

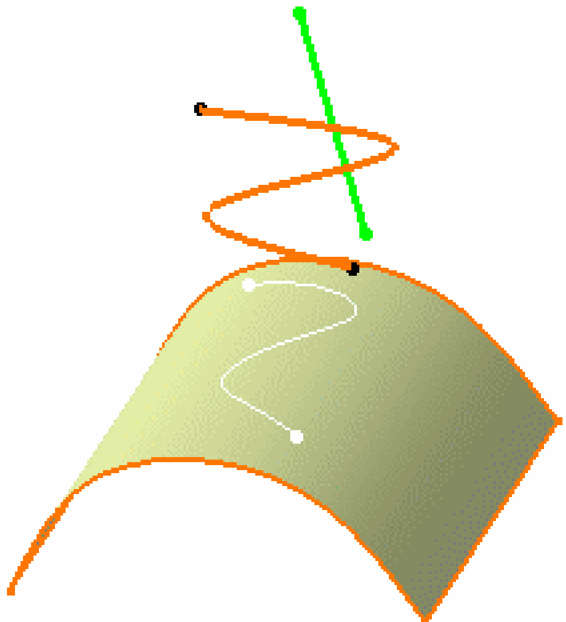
2. Select the element to be **Projected**.

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



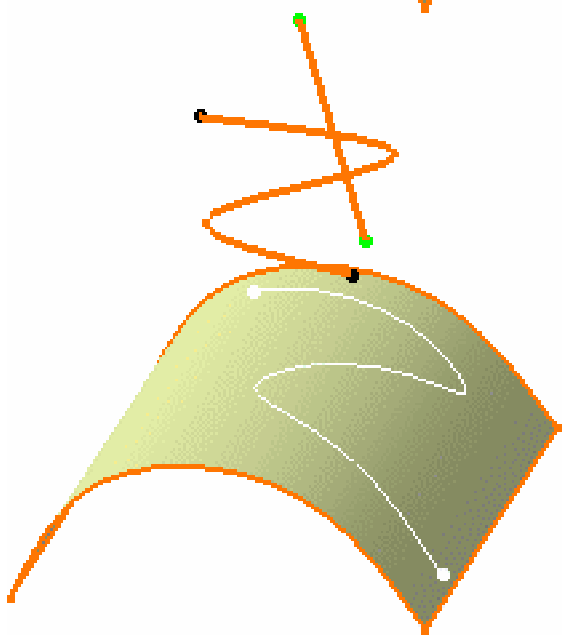
3. Select the **Support** element.

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.

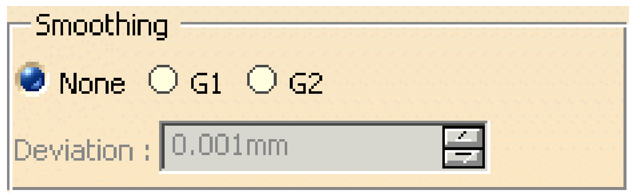


You can also specify the direction by means of X, Y, Z vector components by using the contextual menu on the **Direction** field.

- 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.

5. Click OK to create the projection element.

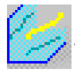
The projection (identified as Project.xxx) is added to the specification tree.

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

Creating Combined Curves

This task shows you how to create combined curves, that is a curve resulting from the intersection of the extrusion of two curves.

Open the Combine1.CATPart document.
Display the Project-Combine toolbar by clicking and holding the arrow from the **Projection** icon.

1. Click the **Combine** icon .

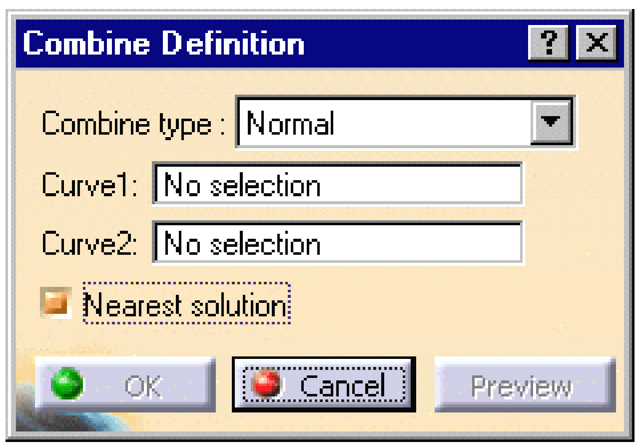
The Combine Definition dialog box appears.

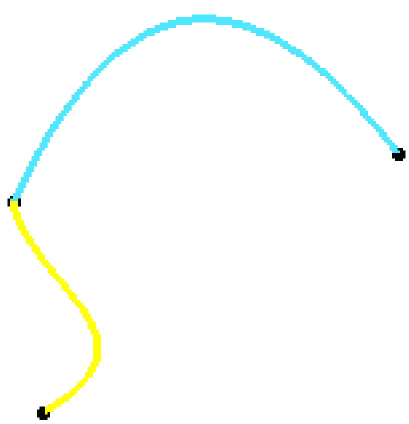
2. Choose the combine type: normal or along directions.

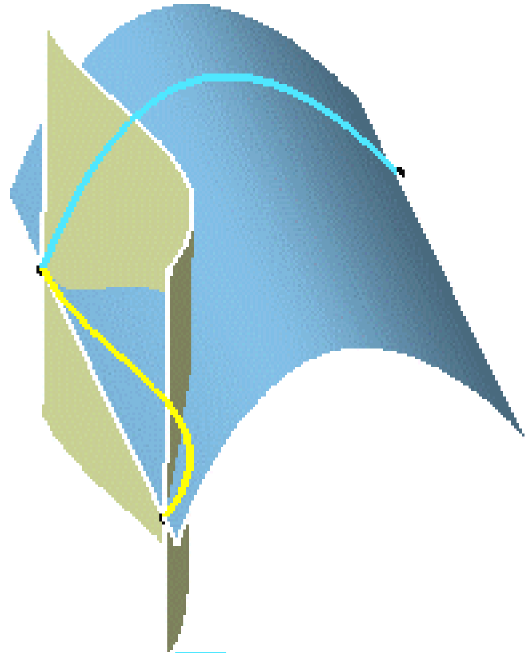
- **Normal**: the virtual extrusion are computed as normal to the curve planes
- **Along directions**: specify the extrusion direction for each curve (Direction1 and Direction2 respectively).

Normal Type

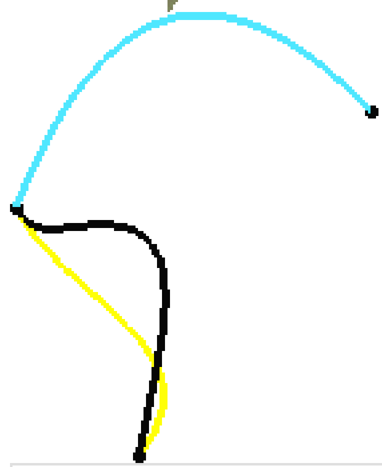
3. Successively select the two curves to be combined.







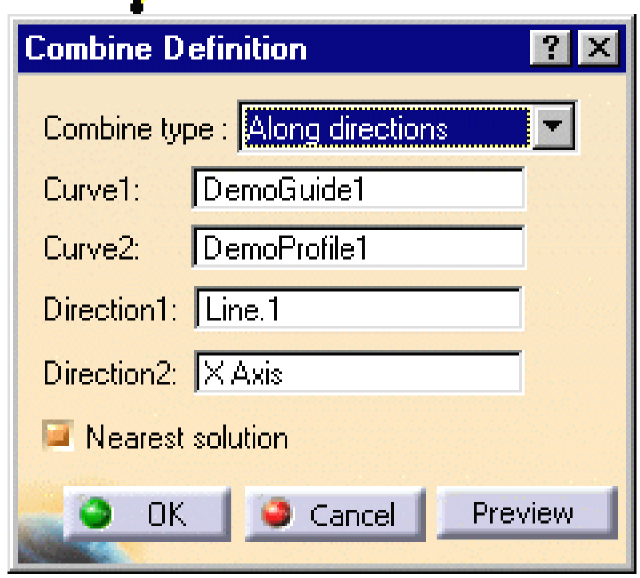
Using the **Normal** type, the combine curve is the intersection curve between the extrusion of the selected curves in virtual perpendicular planes.
This illustration represent the virtual extrusions, allowing the creation of the intersection curve that results in the combine curve.



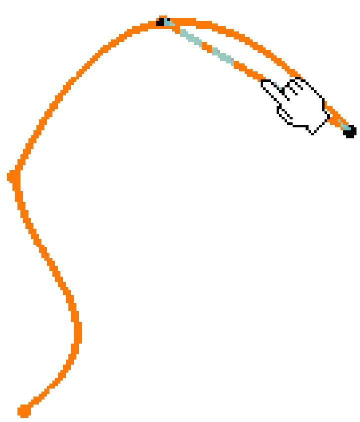
4. Click **OK** to create the element.

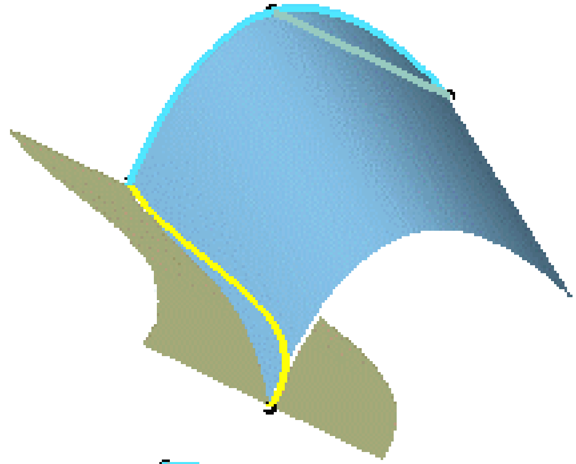
The curve (identified as Combine.xxx) is added to the specification tree.

Along Directions Type

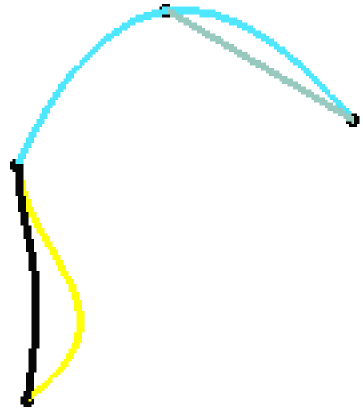


3. Successively select the two curves to be combined and a direction for each curve.





Using the **Along directions** type, the combine curve is the intersection curve between the extrusion of the selected curves along the selected directions, as illustrated here:



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

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

The **Nearest solution** option, allows to automatically create the curve closest to the first selected curve, in case there are several possible combined curves.

Creating Reflect Lines

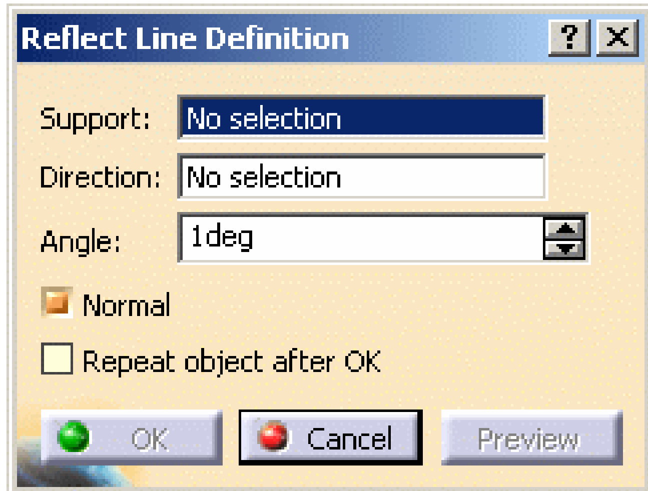
This task shows you how to create reflect lines, whether closed or open. Reflect lines are curves for which the normal to the surface in each point present the same angle with a specified direction.

Open the ReflectLine1.CATPart document.

Display the Project-Combine toolbar by clicking and holding the arrow from the **Projection** icon.

1. Click the **Reflect Lines** icon .

The **Reflect Line Definition** dialog box appears.



2. Successively select the support surface and a direction.

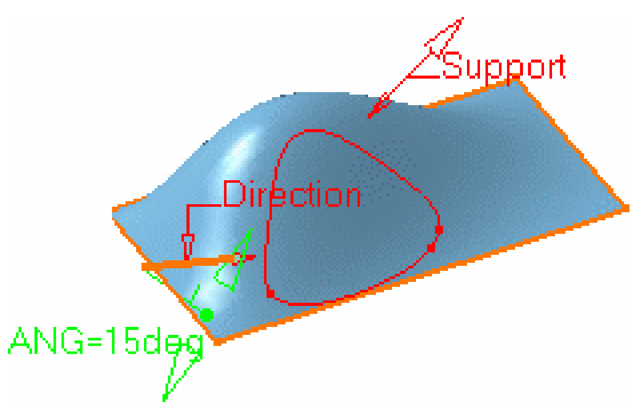
3. Key in an angle, representing the value between the selected direction and the normal to the surface.

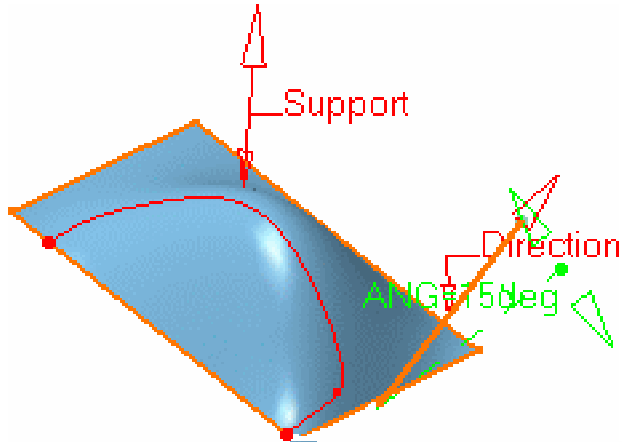
   Here we keyed in 15°.

You can also use the displayed manipulators to modify the angle value (ANG manipulator) or to reverse its direction (Support arrow).

The **Normal** option lets you choose whether the angle should be computed:

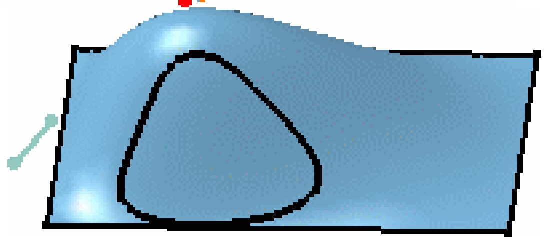
- between the normal to the support and the direction (option checked)



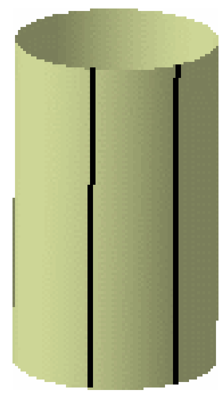


- between the plane tangent to the support and the direction (option unchecked)

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



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



- Use the **Repeat object after OK** checkbox to create several reflect lines, each separated from the initial line 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.

- When several reflect lines are created, as for example on a cylinder as illustrated here, you are prompted to choose to either keep both elements within the Reflect Line object, or to choose one as the reference, as described in Creating the Nearest Entity of a Multiple Element.

Do not use a null angle value on a closed surface issued from a circle for example.

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

Creating Intersections

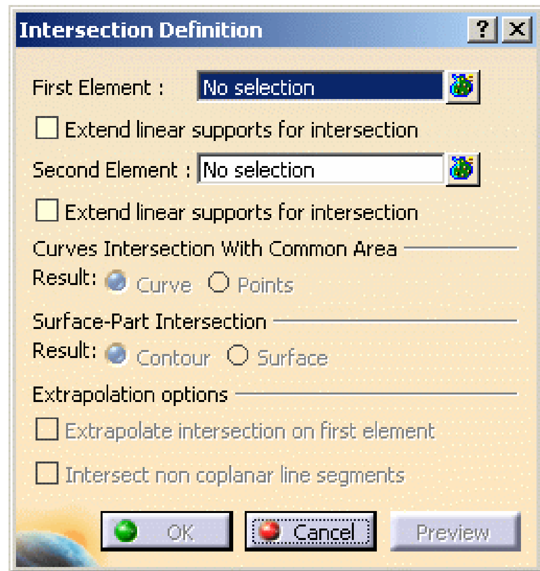
This task shows you how to create wireframe geometry by intersecting elements.

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.

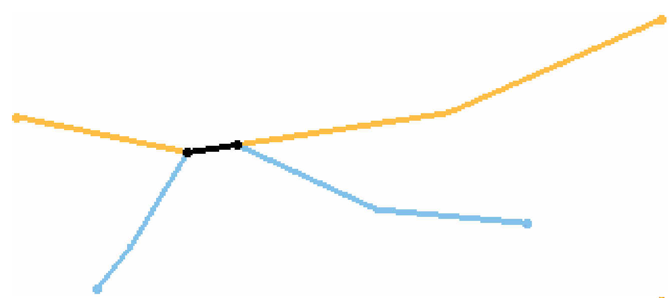
2. Select the two elements to be intersected.

   The intersection is displayed.

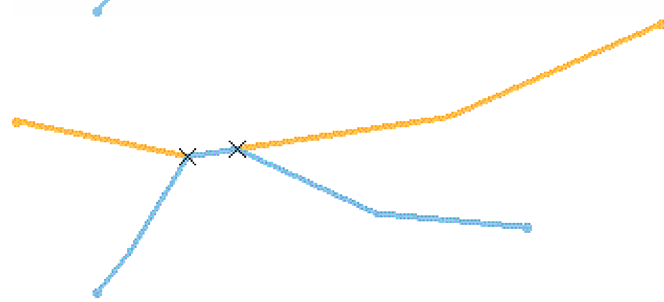
Multi-selection is available on the first selection, meaning you can select several elements to be intersected, but only one intersecting element.

3. Choose the type of intersection to be displayed:

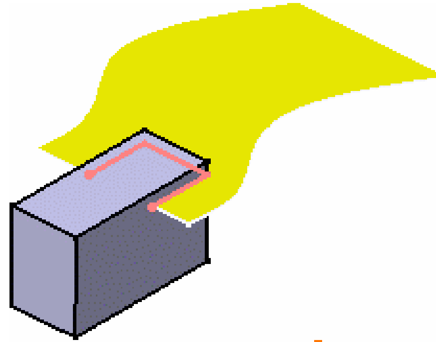
- A **Curve**: when intersecting a curve with another one



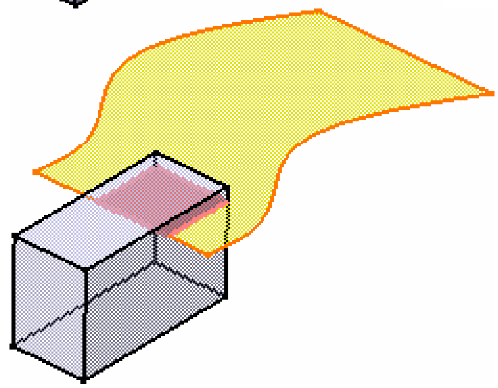
- **Points**: when intersecting a curve with another one



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



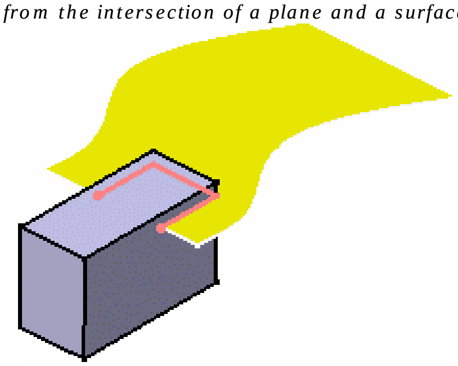
- A **Face**: when intersecting a solid element with a surface (we increased the transparency degree on the pad and surface)



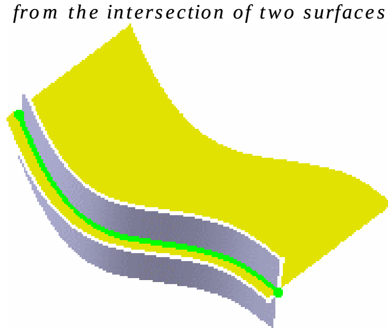
4. Click OK to create the intersection element.

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



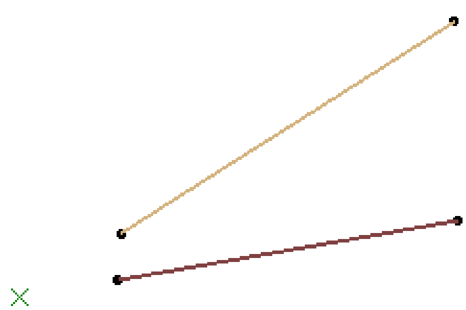
Several options can be defined to improve the preciseness of the intersection.

Open the Intersection2.CATPart document.

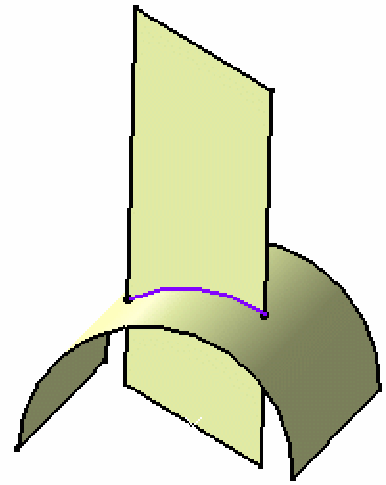
- The **Extend linear supports for intersection** option enables you to extend the first, second or both elements.

Both options are unchecked by default.

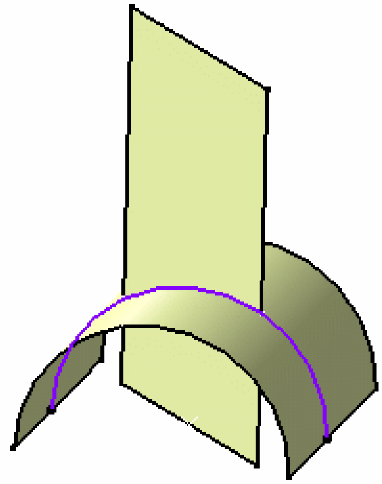
Here is an example with the option checked for both elements.



- 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.



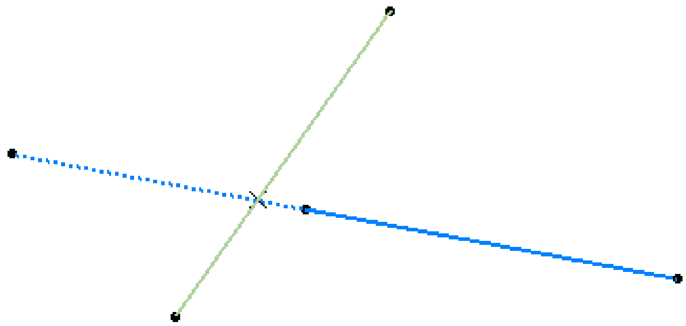
Intersection without the Extrapolation option checked



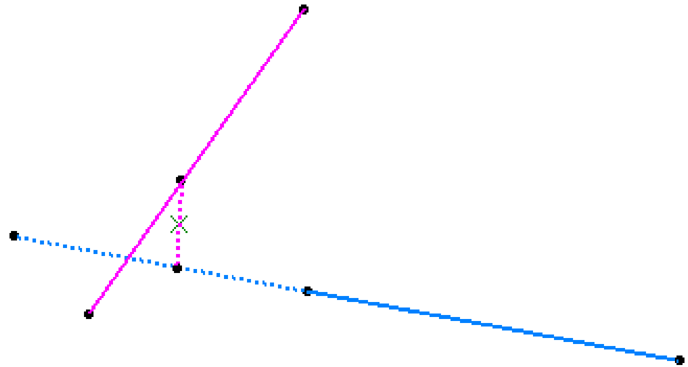
Intersection with the Extrapolation option checked

- The **Intersect non coplanar 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.

When checking this option, both **Extend linear supports for intersection** options are checked too.



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 geometrical sets.

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

Creating Surfaces

Generative Shape Design allows you to model both simple and complex surfaces using techniques such as lofting, sweeping and filling.

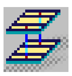

Create extruded surfaces: select a profile, specify the extrusion direction, and define the start and end limits of the extrusion


Create revolution surfaces: select a profile, a rotation axis, and define the angular limits of the revolution surface


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.

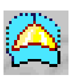

Create offset surfaces: select the surface to be offset, enter the offset value and specify the offset direction

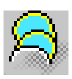

Create variable offset surfaces: select the surface to be offset, select the surfaces to remove and specify a constant or variable offset direction


Create rough offset surfaces: select the surface to be offset, enter the offset value and specify the deviation


Create swept surfaces: select one or more guiding curves, the profile to be swept, possibly a spine, reference surface, and start and end values


Create adaptive swept surfaces: select a guiding curve, a profile to be swept, points to define more sections if needed, set the constraints on each section, and choose a spine


Create fill surfaces: select curves, or surface edges, forming a closed boundary, and specify the continuity type

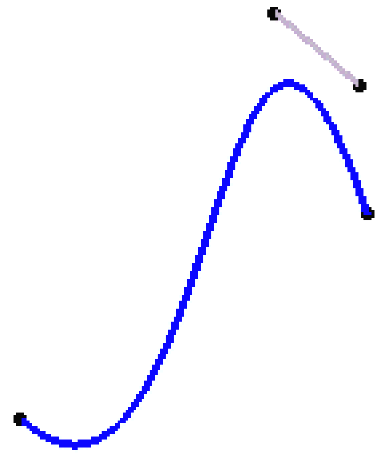

Create multi-sections surfaces: select two or more planar section curves, possibly guide curves and a spine, and specify tangency conditions

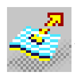

Create blend surfaces: select two curves, and possibly their support, specify the tension, continuity, closing point and coupling ratio, if needed

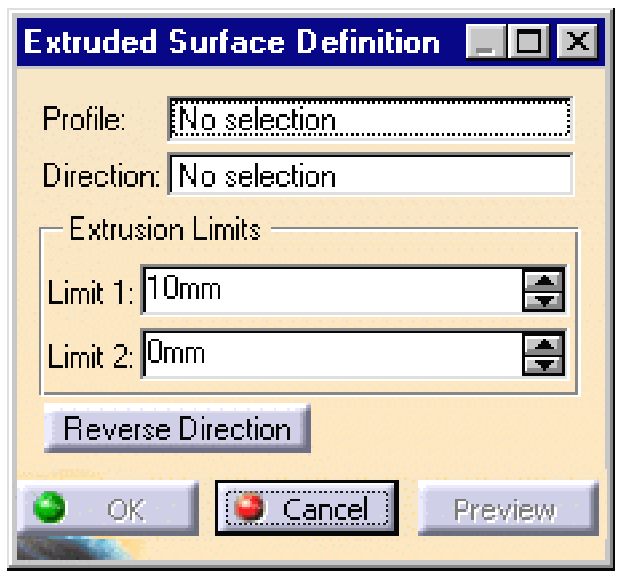
Creating Extruded Surfaces

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

Open the Extrude1.CATPart document.



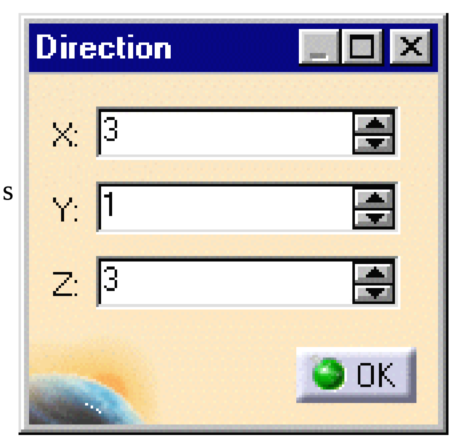
1. Click the **Extrude** icon .



The Extruded Surface Definition dialog box appears.

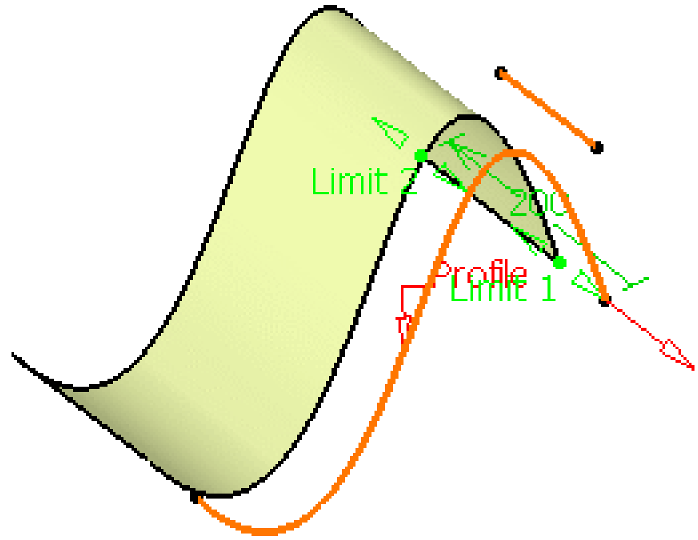
2. Select the **Profile** to be extruded and specify the desired extrusion **Direction**.

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



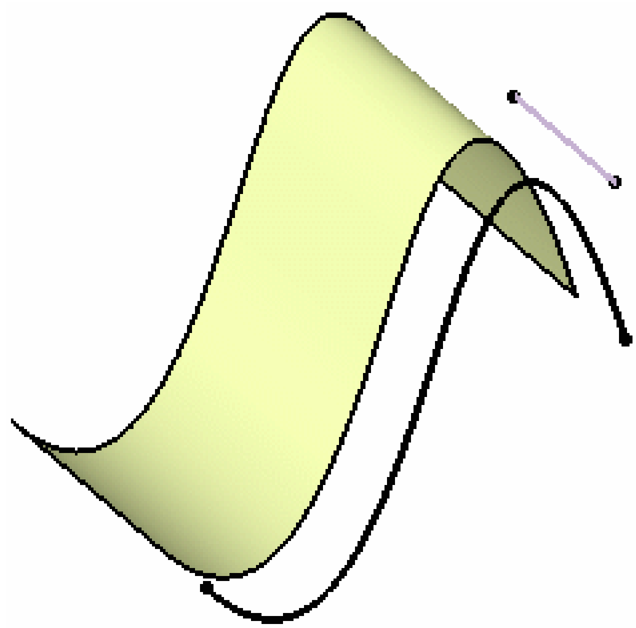
You can also specify the direction by means of X, Y, Z vector components by using the contextual menu on the Direction area.

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



4. Click OK to create the surface.

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



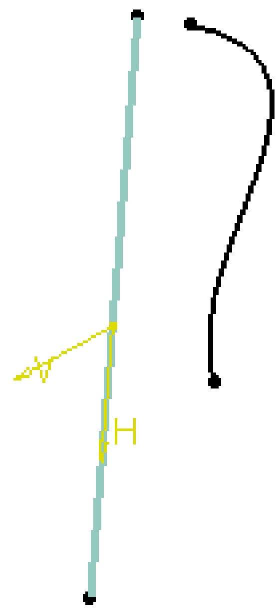
You can click the **Reverse Direction** button to display the extrusion on the other side of the selected profile or click the arrow in the 3D geometry.

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

Creating Revolution Surfaces

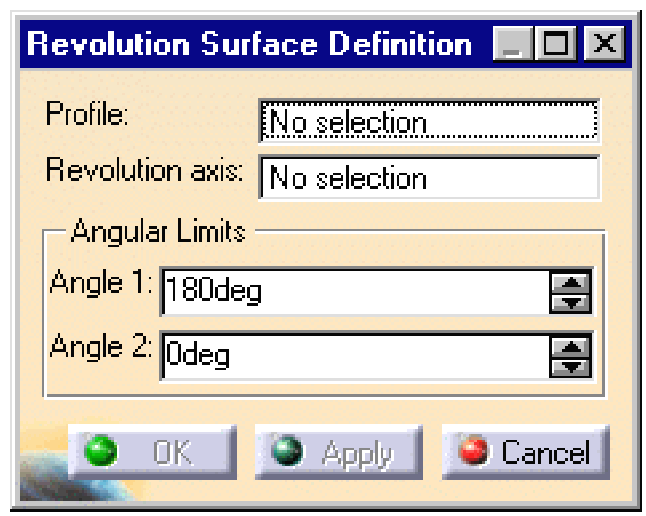
This task shows how to create a surface by revolving a planar profile about an axis.

Open the Revolution1.CATPart document.



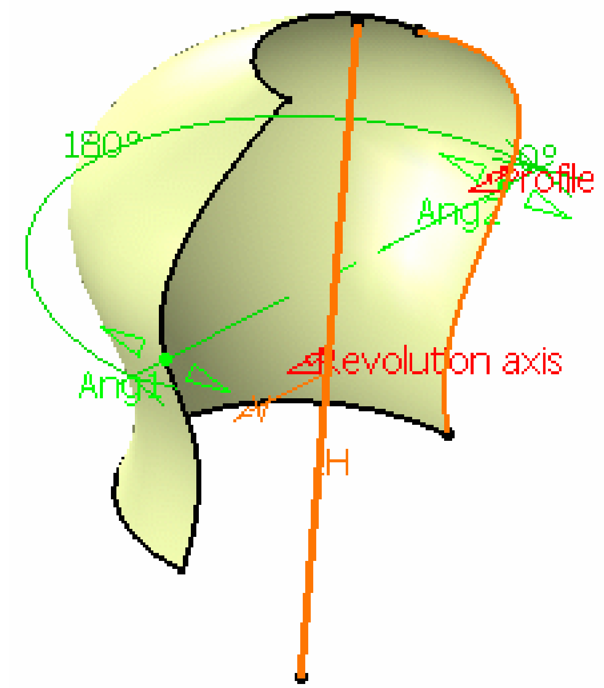
1. Click the **Revolve** icon .

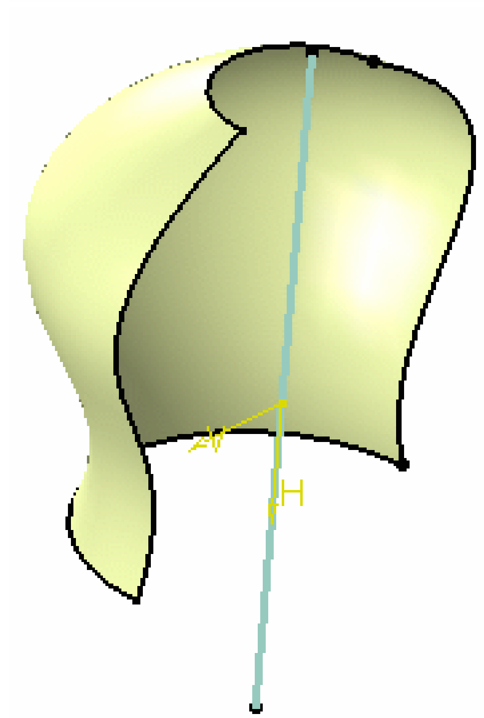
   The Revolution Surface Definition dialog box appears.



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.

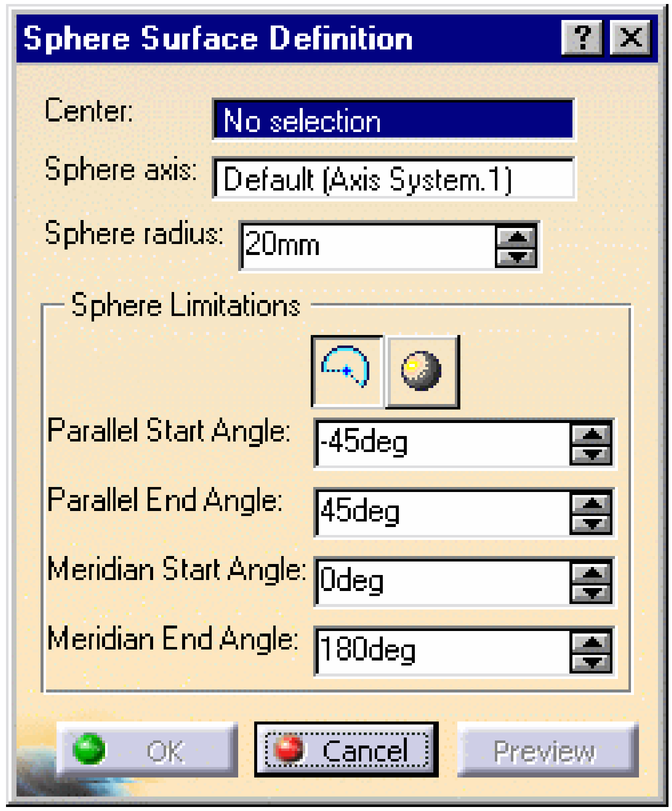
Creating Spherical Surfaces

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.

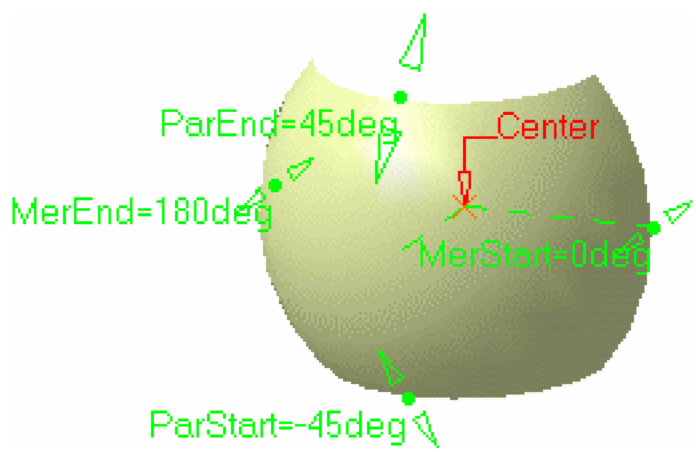
The Sphere Surface Definition dialog box is displayed.



2. Select the center point of the sphere.

3. Select an axis-system.

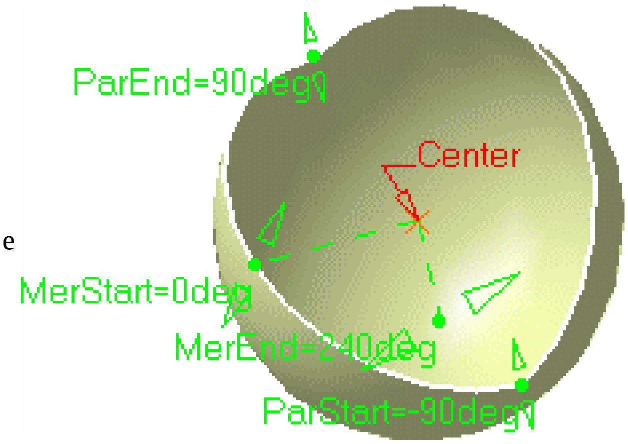
This axis-system determines the orientation of the meridian and parallel curves, and therefore of the sphere.



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.

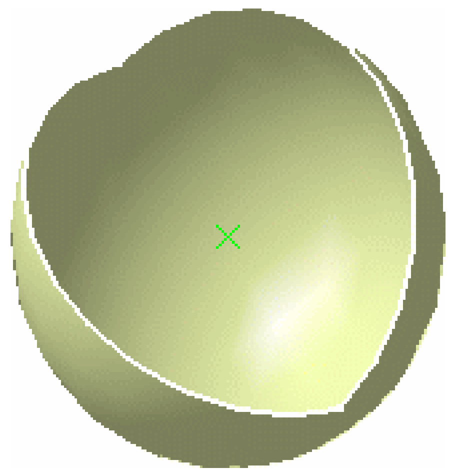
4. Click Preview to preview the surface.

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.



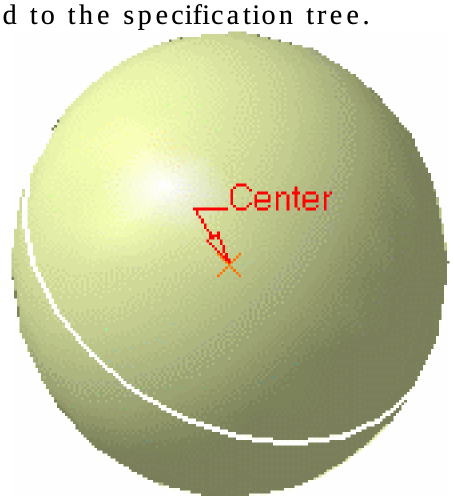
Parallel angular limits are comprised within the -90° and 90° range.
Meridian angular limits are comprised within the -360° and 360° range.

6. Click OK to create the spherical 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.

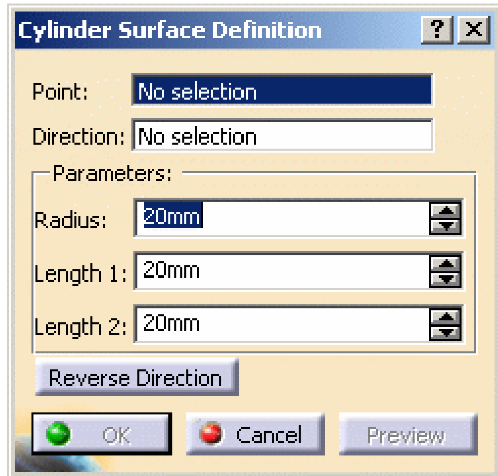
Creating Cylindrical Surfaces

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.

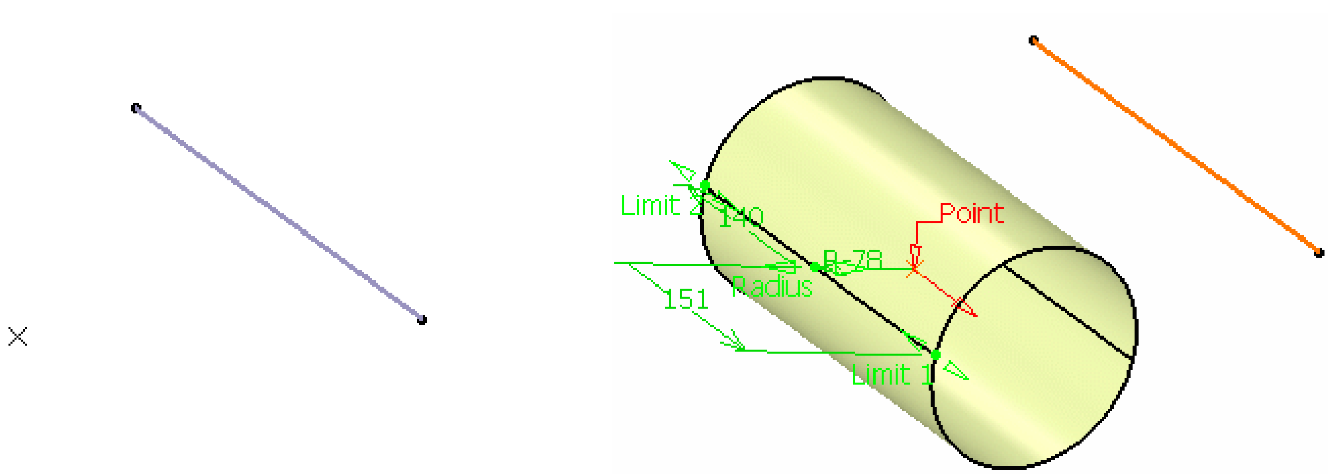


2. Select the **Point** that gives the center of the circle to be extruded and specify the desired **Direction** of the cylinder axis.

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.

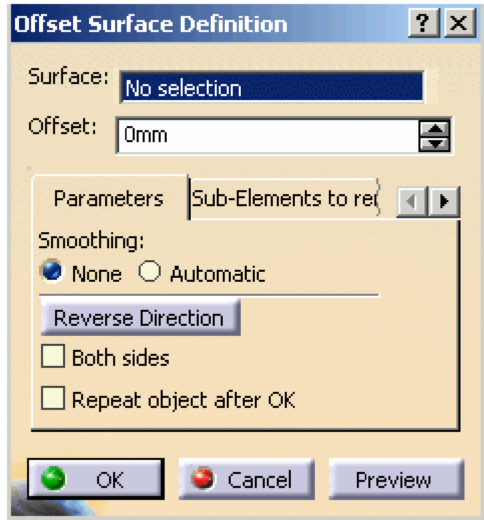
Creating Offset Surfaces

This task shows you how to create a surface, or a set of surfaces, by offsetting an existing surface, or a set of surfaces. It can be any type of surface, including multi-patch surfaces resulting from fill or any other operation.

Open the Offset1.CATPart document.

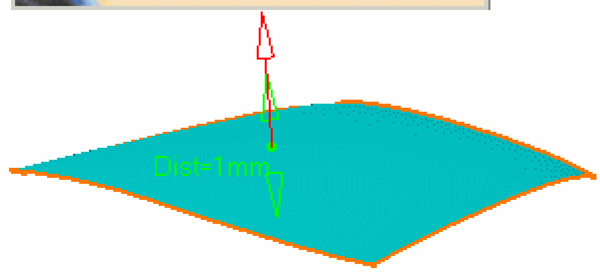
1. Click the **Offset** icon .

The Offset Surface Definition dialog box appears.



2. Select the **Surface** to be offset.

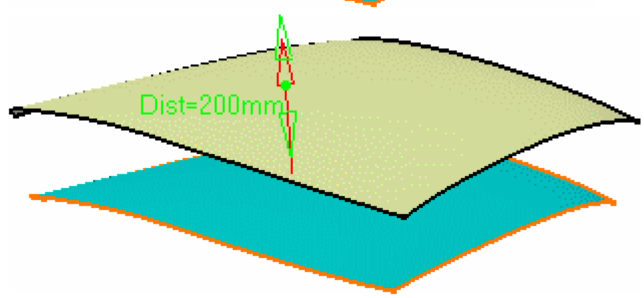
   An arrow indicates the proposed direction for the offset.



3. Specify 200mm as the **Offset** value.

4. Click **Preview** to preview the offset surface.

The offset surface is displayed normal to the reference surface.



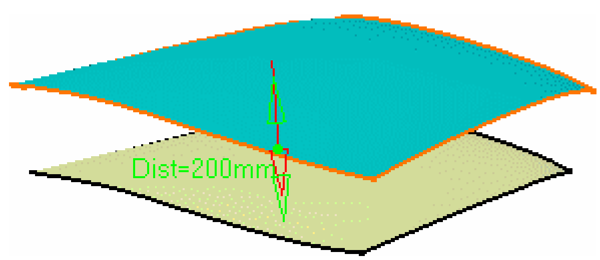
5. Click OK to create the surface.

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

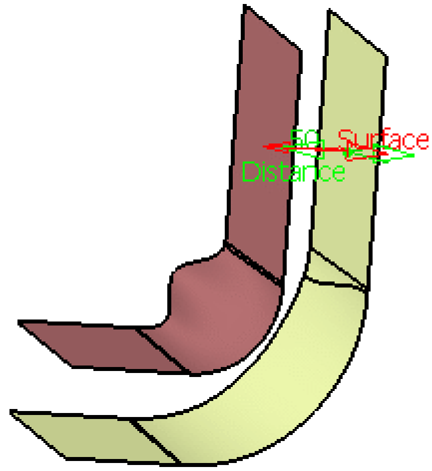
Depending on the geometry configuration and the offset value, an offset may not be allowed as it would result in a debased geometry. In this case, you need to decrease the offset value or modify the initial geometry.

The Parameters tab allows you to:

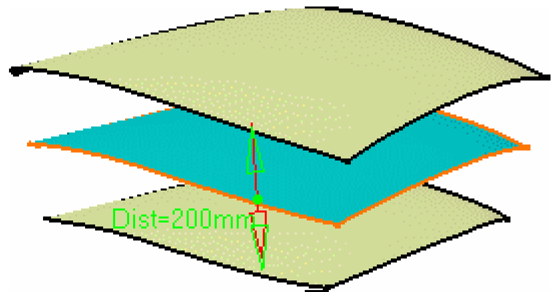
- display the offset surface on the other side of the reference surface by clicking either the arrow or the **Reverse Direction** button.



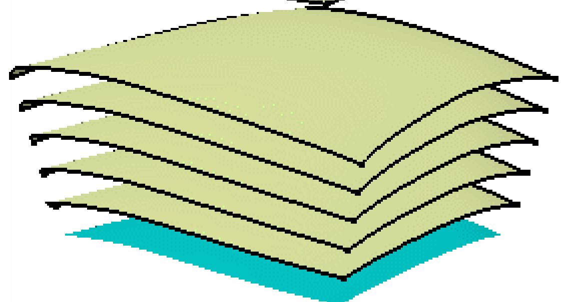
- define the smoothing:
  - **None:** the smoothing is constant
  - **Automatic** (you can use the Offset3.CATPart document): a local smoothing is applied only if the constant offset cannot be performed. It cleans the geometry of the surface and enables the offset.
    A warning panel is launched and the modified surface is shown in the 3D geometry.
    If a surface still cannot be offset, no smoothing is performed and a warning message is issued (as in the constant offset mode).



- generate two offset surfaces, one on each side of the reference surface, by checking the **Both sides** option.

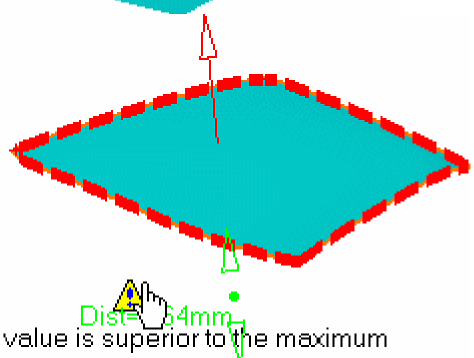


- create several offset surfaces, each separated from the initial surface by a multiple of the **offset** value, by checking the **Repeat object after OK** option.
  Simply indicate in the Object Repetition dialog box the number of instances that should be created and click OK.
  Remember however, that 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 Preview.



- The options set in the dialog box are retained when exiting then returning to the Offset function.
- When you modify an input value through the dialog box, such as the offset value or the direction, the result is computed only when you click on the Preview or OK buttons.

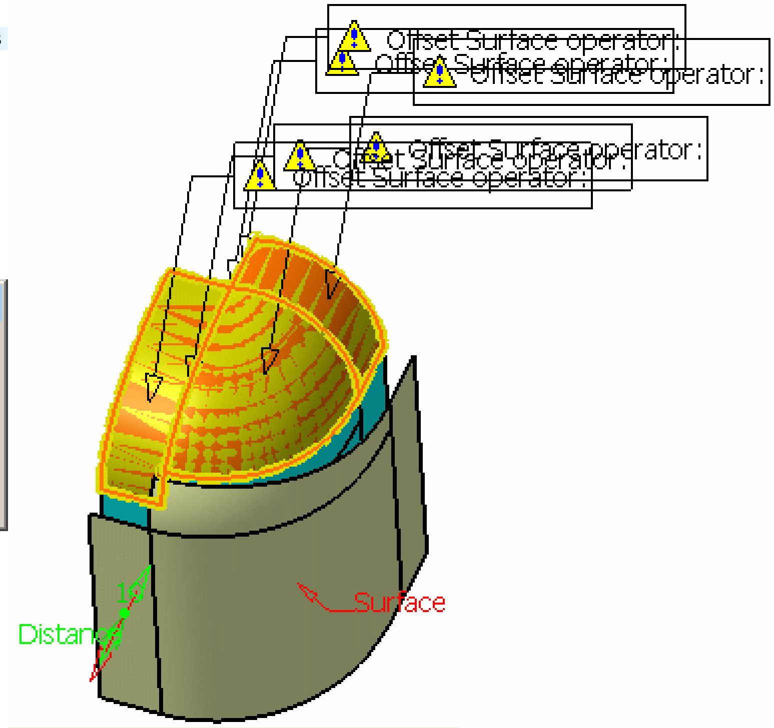
Removing Sub-Elements

The Sub-Elements to remove tab helps you for the analysis in case the offset encounters a problem.

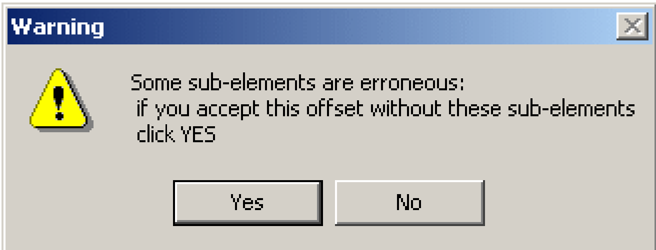
Open the Offset2.CATPart document.

1. Perform steps 1 to 4.
2. Click **Preview**.

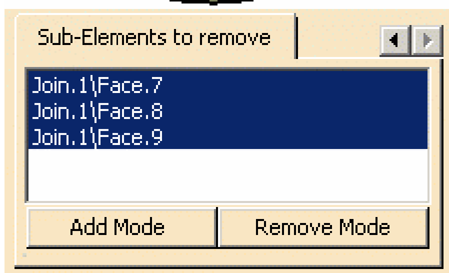
The Warning dialog box appears, the geometry shows the erroneous sub-elements, and flag notes display sub-elements to remove.



3. Click **Yes** to accept the offset.



In the Offset Surface Definition dialog box, the Sub-Elements to remove tab lists the erroneous sub-elements and a preview of the offset is displayed.



- If you move the mouse over a flag note, a longer message giving an accurate diagnosis is displayed.
- You can remove a sub-element by right-clicking it and choosing **Clear Selection** from the contextual menu.

The following modes are optional, you may use them if you need to add or remove a sub-element to create the offset.

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

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.

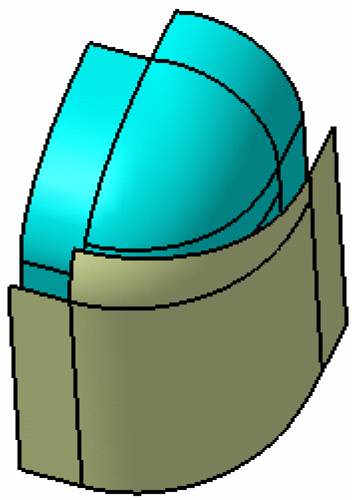
The list of sub-elements to remove is updated each time an element is added.
Note that if you modify an input in the Offset dialog box, the list is re-initialized.

4. Click Preview.

   The offset surface is displayed normal to the reference surface.

5. Click OK to create the surfaces.

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



Performing a Temporary Analysis

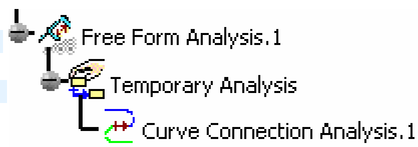
While in the Offset command, you can perform a temporary analysis in order to check the connections between surfaces or curves.

Open the Offset4.CATPart document.

1. Perform steps 1 to 4 (set 20mm as the Offset value).
2. Click **Preview**.

   The Temporary Analysis icon is available from the Tools toolbar.
3. Click the **Temporary Analysis mode** icon .
4. Select the analysis to be performed in the Analysis toolbar by clicking either the **Connect checker** icon or the **Curve connect checker** icon .
5. Click OK in the Connect checker or Curve connect checker dialog box.

You must activate the temporary analysis mode before running any analysis. Otherwise, a persistent analysis will be performed.
The Temporary Analysis node is displayed in the specification tree and the associated analysis (here Curve Connection Analysis.1) appears below.



The analysis is not persistent. Thus when you click OK in the Offset Definition dialog box to create the curve, the Temporary Analysis node disappears from the specification tree.

An option is available from **Tools -> Options** to let you automatically set the analysis as temporary. Refer to the Customizing section.

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

Creating Swept Surfaces

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 by default)
- along one or two guide curves while respecting a specified 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.

You can sweep an **implicit linear profile** along a spine. This profile is defined by:

- two guide curves and two length values for extrapolating the profile
- a guide curve and a middle curve
- a guide curve, a reference curve, an angle and two length values for extrapolating the profile
- a guide curve, a reference surface, an angle and two length values for extrapolating the profile
- a guide curve, and a reference surface to which the sweep is to be tangent
- a guide curve and a draft direction

You can sweep an **implicit circular profile** along a spine. This profile is defined by:

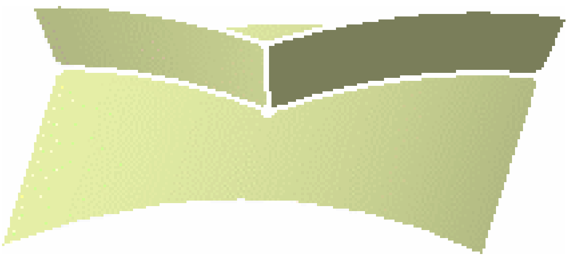
- three guide curves
- two guide curves and a radius value
- a center curve and two angle values defined from a reference curve (that also defines the radius)
- a center curve and a radius.

You can sweep an **implicit conical profile** along a spine. This profile is defined by:

- three guide curves
- two guide curves and a radius value
- a center curve and two angle values defined from a reference curve (that also defines the radius)
- a center curve and a radius.

- 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.

- Generally speaking, the spine must present a tangency continuity.
  However, in a few cases, even though the spine is not tangent continuous, the swept surface is computed:
  - when the spine is by default the guide curve and is planar, as the swept surface is extrapolated then trimmed to connect each of its segments. Note that if a spine is added by the user, the extrapolation and trim operations are not performed.
  - when consecutive segments of the resulting swept surface do not present any gap.



Tangency discontinuous spine with connex swept segments (the sweep is created)

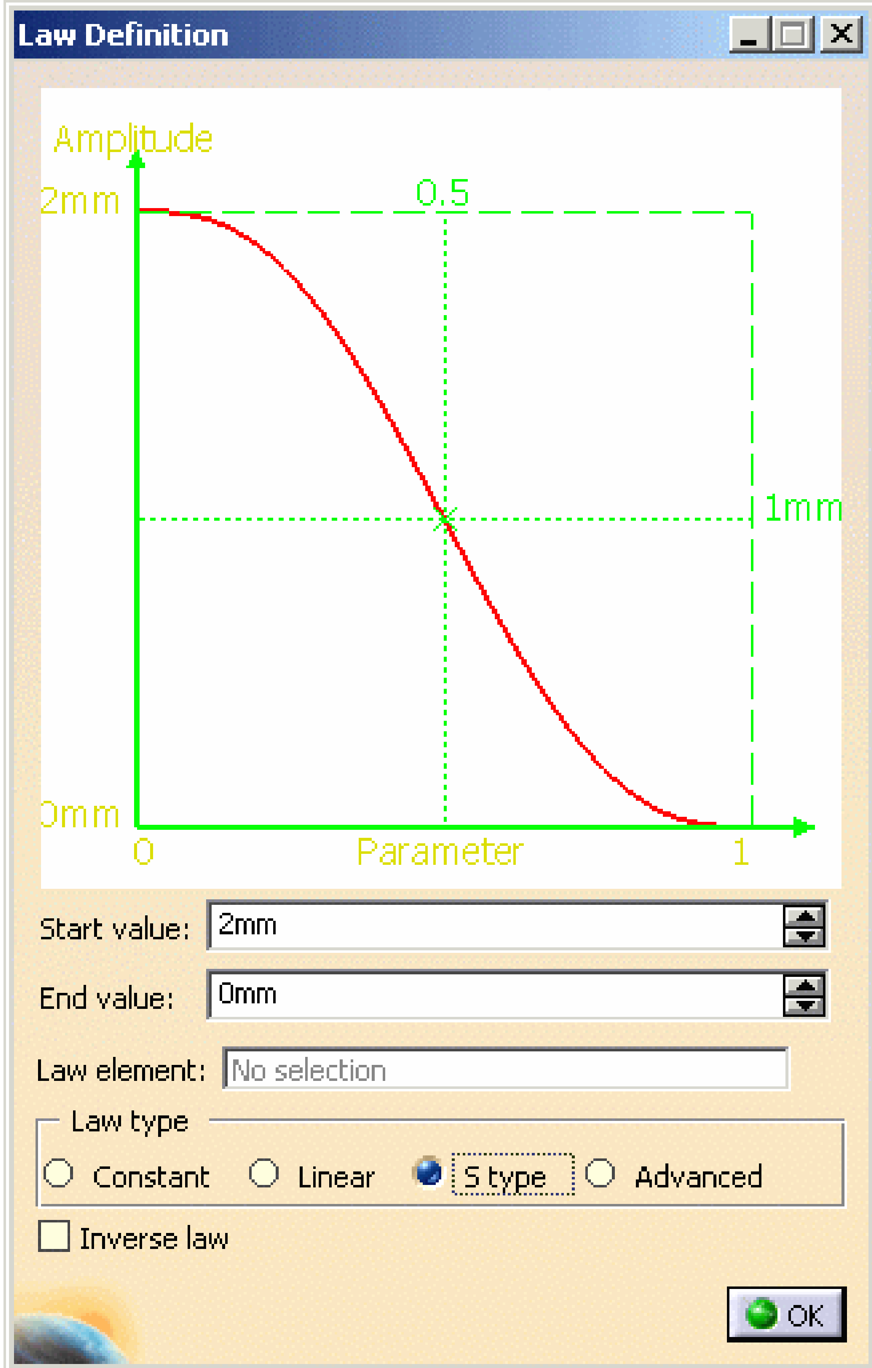
Tangency discontinuous spine with non connex swept segments (the sweep is not created)

Defining Laws for Swept Surfaces

Whatever the type of sweep, whenever a value is requested (angle or length) you can click the **Law** button to display the Law Definition dialog box. It allows you to define your own law to be applied rather than the absolute value.

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)).



Four law types are available:

a. **Constant**: a regular law, only one value is needed.
b. **Linear**: a linear progression law between the **Start** and **End** indicated values
c. **S type**: an S-shaped law between the two indicated values
d. **Advanced**: allowing to select a Law element as defined in Creating Laws.

Check the **Inverse law** button to reverse the law as defined using the above options.

You can also apply laws created with the *Knowledge Advisor* workbench to swept surfaces.

The law can be negative, providing the curves are curvature continuous.

Removing Twisted Areas

During creation or edition, you can now generate swept surfaces that have a twisted area by delimiting the portions of the swept surface to be kept. The generated surface is therefore composed of several unconnected parts.

This capability is available with all types of swept surfaces, except for the **With tangency surface** and **With two tangency surfaces** subtypes of the linear profile, and the **One guide and tangency surface** and **With two surfaces** subtypes of the circular profile.

Open the Sweep-Twist.CATPart document.

Let's take an example by creating a swept surface with an implicit linear profile.

1. Click the **Sweep** icon .

   The Swept Surface Definition dialog box appears.

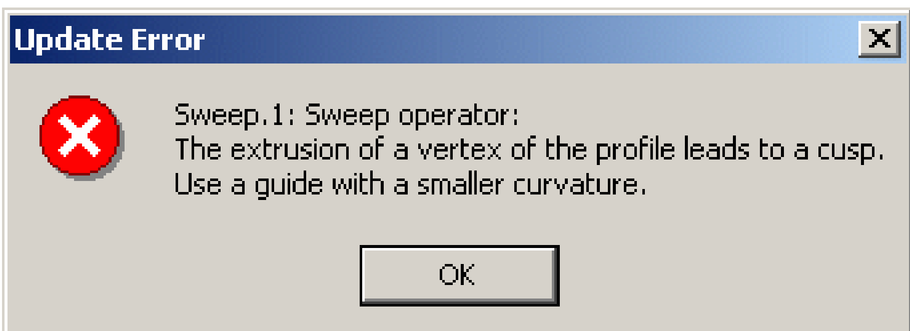
2. Click the Line profile icon and choose the **With Reference surface** subtype.

3. Select Curve.1 as the Guide Curve 1.

4. Select the xy plane as the reference surface.
5. Define a Length 1 of 30 mm and a Length 2 of 10 mm.

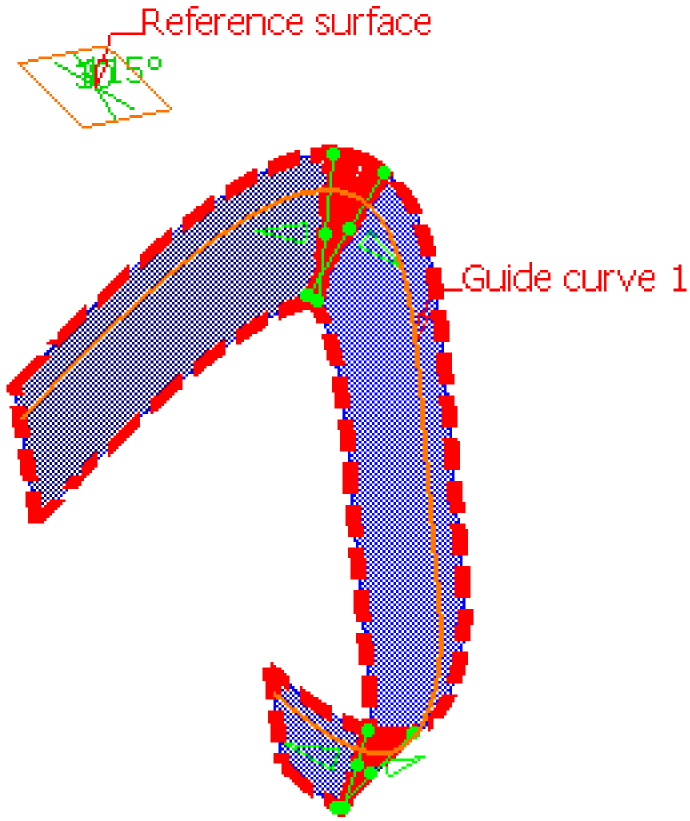
6. Click **Preview**.

An error message displays asking you to use a guide with a smaller curvature.



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

Two manipulators ("cutters") appear for each untwisted zone. Their default positions are the maximal zone delimiters out of which they cannot be dragged. This maximal zone corresponds to the larger untwisted portion of the swept surface.

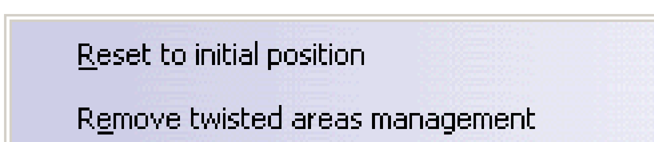


8. Use these manipulators to delimit the portions of the swept surface you want to keep. These cutters are stored in the model as points on curve with ratio parameters when the guide curve is not closed.

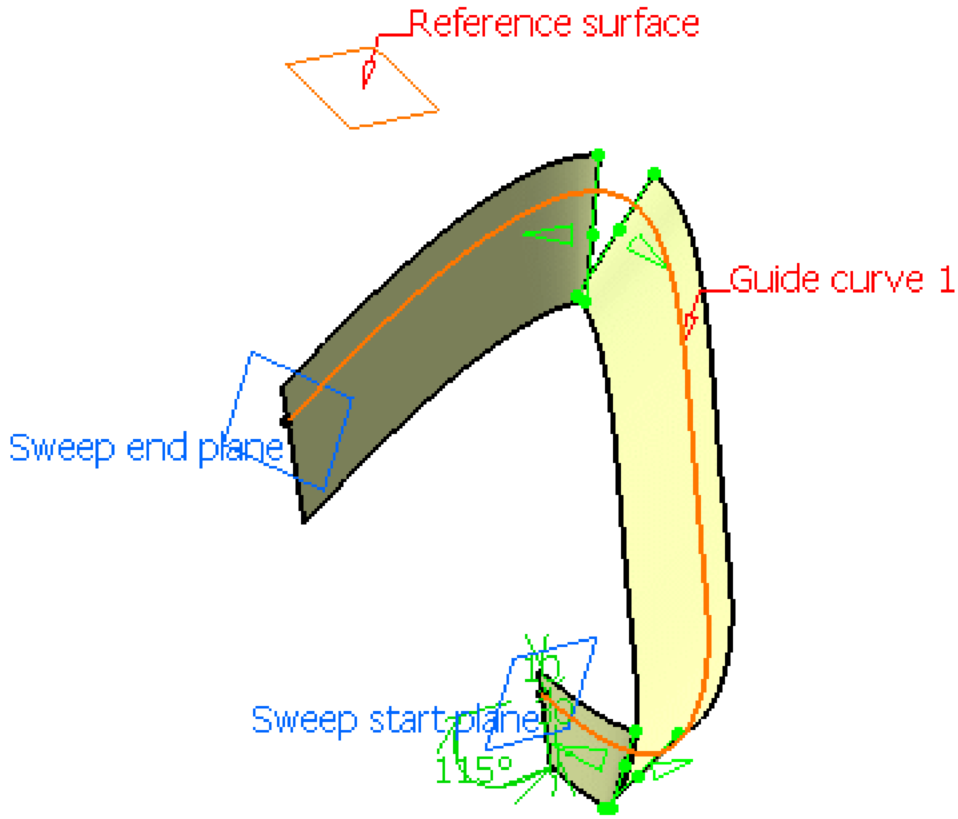
We advise you to cut a bit less than the maximal zone to delimit a safety area around the twisted portion.

A contextual menu is available on the manipulators:

- **Reset to initial position**: sets the manipulators back to their default positions, thqt is the position defined as the maximal zone.

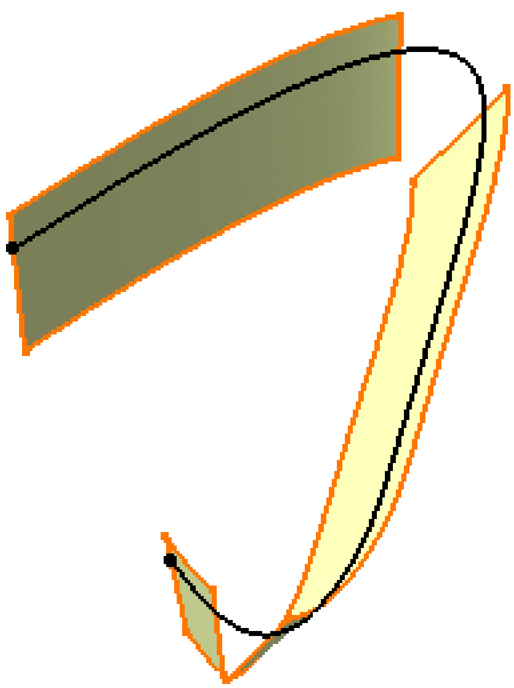


- **Remove twisted areas management**: removes the manipulators and performs the swept surface generation again.



9. Click **Preview** again in the Swept Surface Definition dialog box.

The swept surface is generated.



10. Click **OK** to create the swept surface.

- If you modify the length value after clicking **Preview**, and the swept surface to be generated has no twisted area, the generated swept surface will still be cut. Use the **Remove twisted areas management** option to start the operation again.

- As the generated surface is composed of several unconnected part, the Multi-result management dialog box opens. For further information, refer to the Managing Multi-Result Operations chapter.

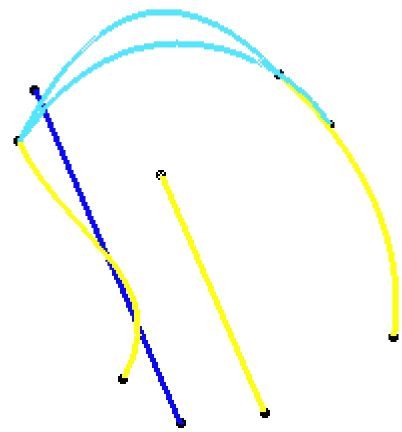
Creating Swept Surfaces Using an Explicit Profile

This task shows you how to create swept surfaces that use an explicit profile. These profiles must not be T- or H-shaped profiles.

The following sub-types are available:

- With reference surface
- With two guide curves
- With pulling direction

You can use the wireframe elements shown in this figure.



Open the Sweep1.CATPart document.

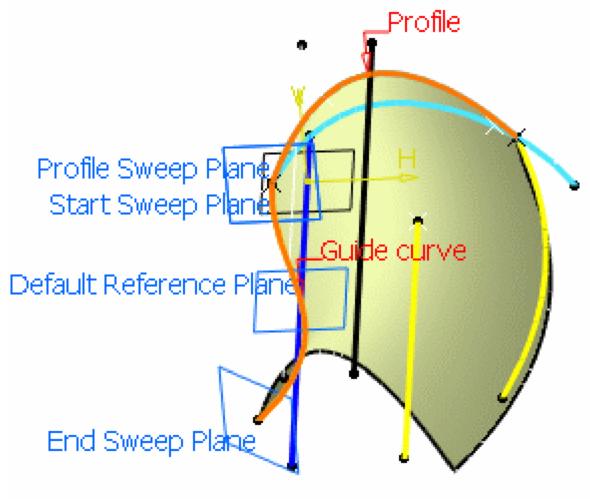
1. Click the **Sweep** icon .

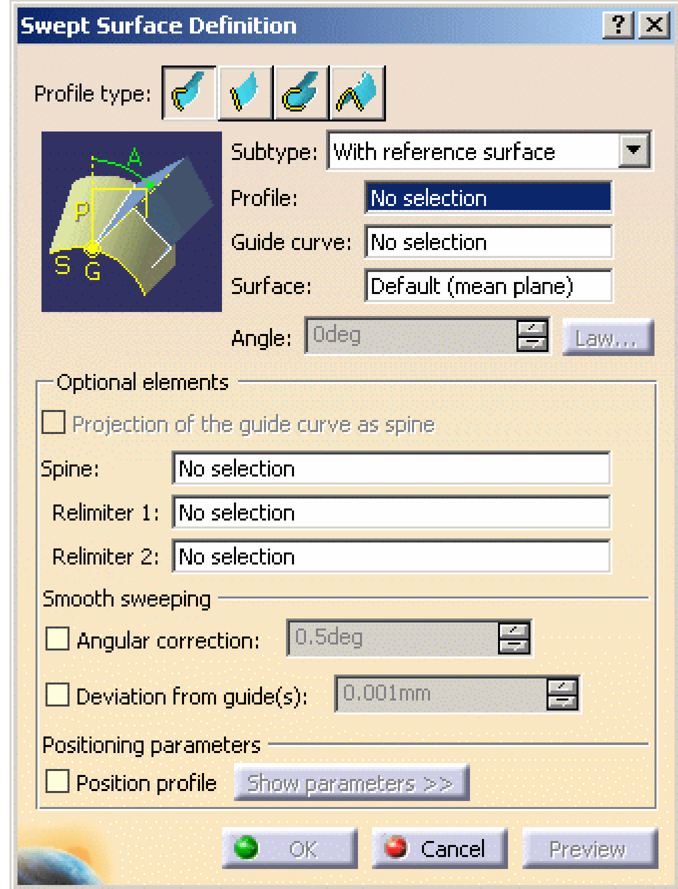
The Swept Surface Definition dialog box appears.

2. Click the **Explicit** profile icon, then use the drop-down list to choose the subtype.

With reference surface

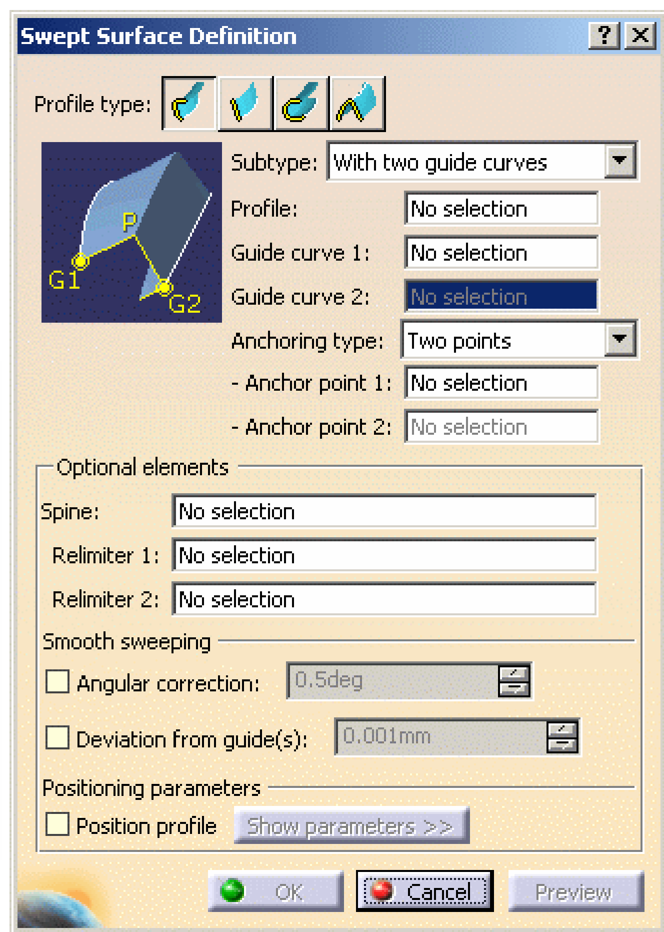
- Select the **Profile** to be swept out (DemoProfile1).
- Select a **Guide curve** (DemoGuide1).
- Select a surface (by default, the reference surface is the mean plane of the spine) in order to control the position of the profile during the sweep.
  Note that in this case, the guiding curve must lie completely on this reference surface, except if it is a plane. You can impose an **Angle** on this surface.

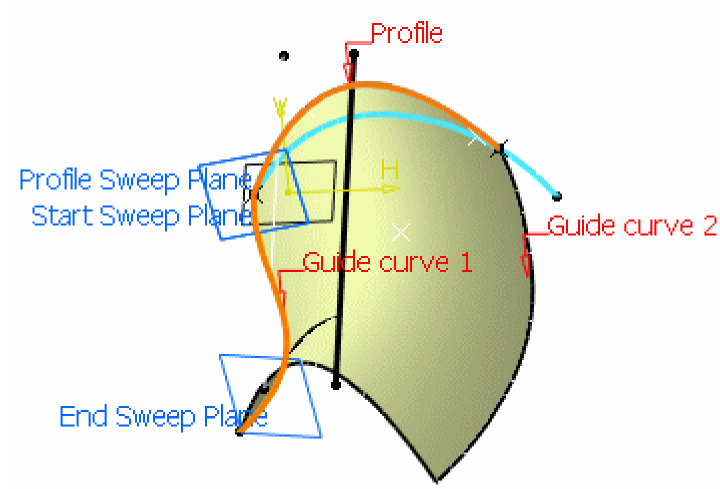




With two guide curves

- Select the **Profile** to be swept out (DemoProfile1).
- Select a first **Guide curve** (DemoGuide1).
- Select a second **Guide curve** (DemoGuide2).

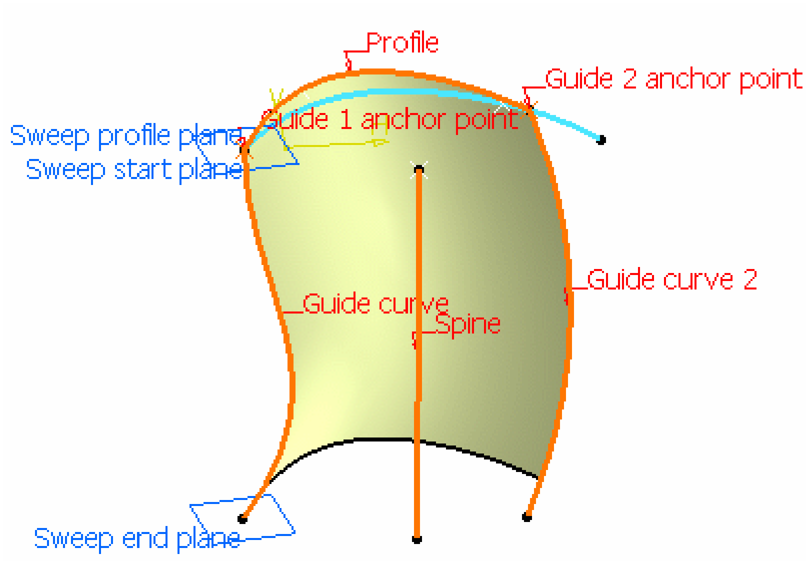




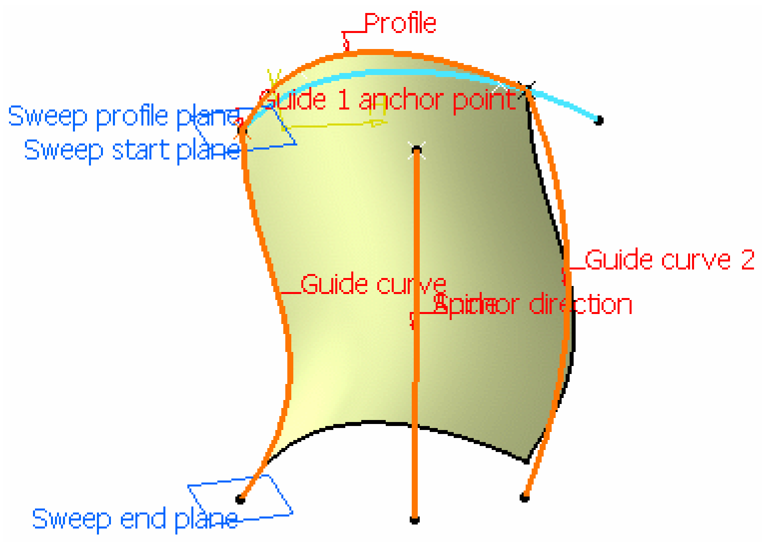
You can also specify anchor points for each guide. These anchor points are intersection points between the guides and the profile's plane or the profile itself, through which the guiding curves will pass.

There are two anchoring types:

- Two points: select anchor points on the profile which will be matched respectively to Guide Curve 1 and 2.
  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 the profile which will be matched onto Guide Curve 1 and an anchor direction. In each sweeping plane, the profile is rotated around the anchor point so that the anchor direction (linked to this profile) is aligned with the two guide curves, from Guide Curve 1 to Guide Curve 2.

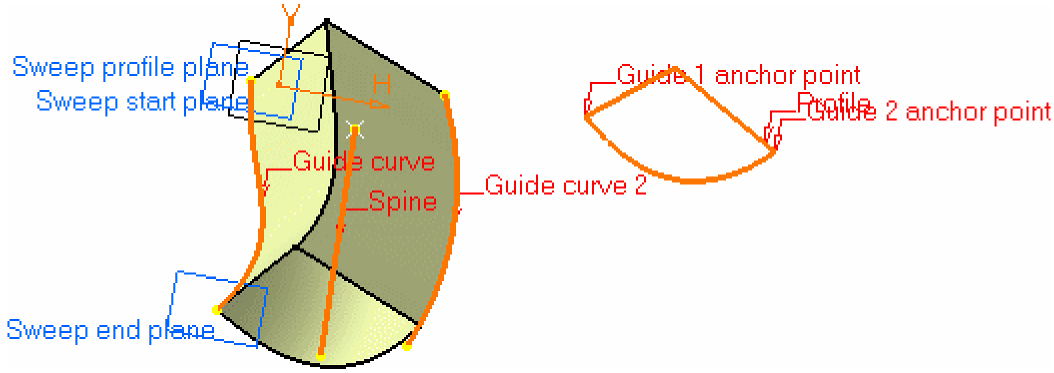


Sweep without positioning
Two points anchoring type



Sweep without positioning
Point and direction anchoring type

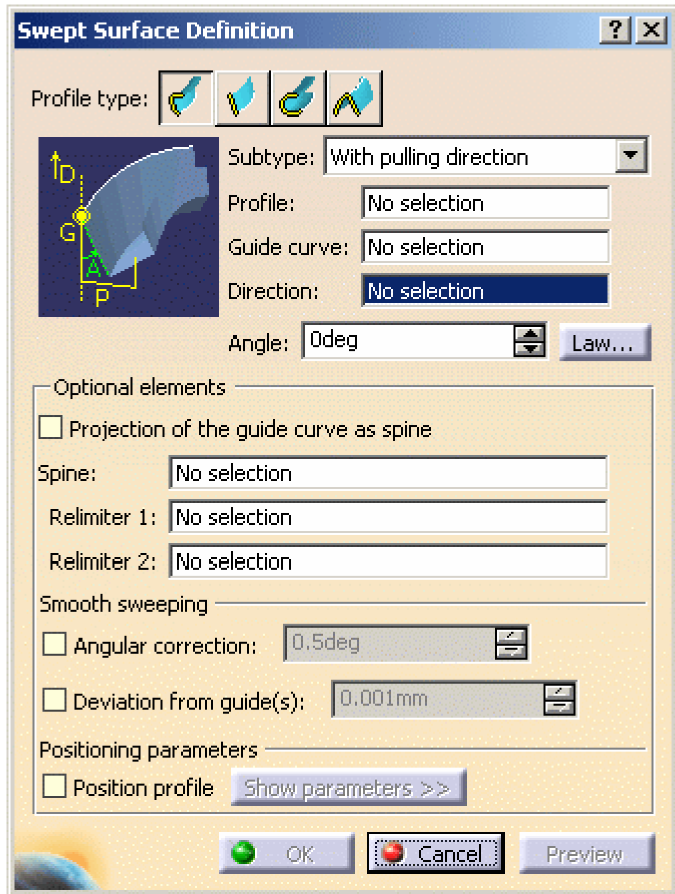
If the profile is manually positioned defining anchor points will position the profile between the guides, matching the anchor points with guide intersection points, prior to performing the sweeping operation.



With pulling direction

The With pulling Direction subtype is equivalent to the With reference surface subtype with a reference plane normal to the pulling direction.

- Select the **Profile** to be swept out (DemoProfile1).
- Select a first **Guide curve** (DemoGuide1).
- Select a **Direction**.



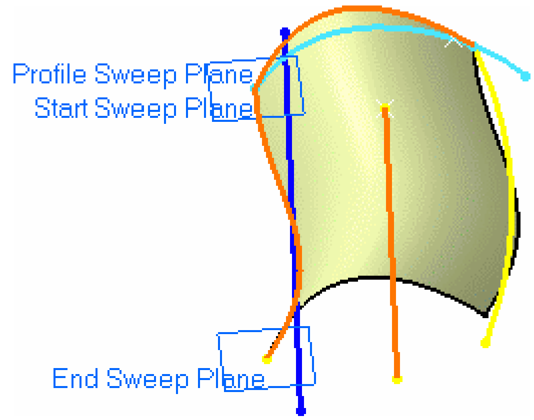
Check the **Projection of the guide curve as spine** option so that the projected spine is the projection of the guide curve onto the reference plane.

- This option is not available with the **Two guides** sub-type.
- It is available with the **Reference surface** sub-type if the reference surface is a plane.
- It is available with the **Pulling direction** sub-type.

3. If needed, select a **Spine**.

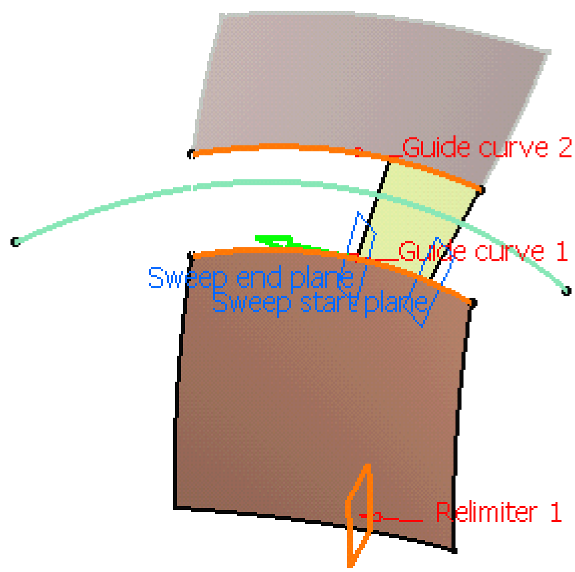
If no spine is selected, the guide curve is implicitly used as the spine.

Here is an example with a linear spline.



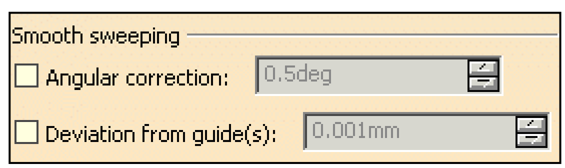
You can define spine relimiters (points or planes) in order to longitudinally reduce the domain of the sweep, if the swept surface is longer than necessary for example.

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 relimit the default spine, thus avoiding to split it to create the sweep.
- You can stack the creation of the elements by using the contextual menu available in either field.

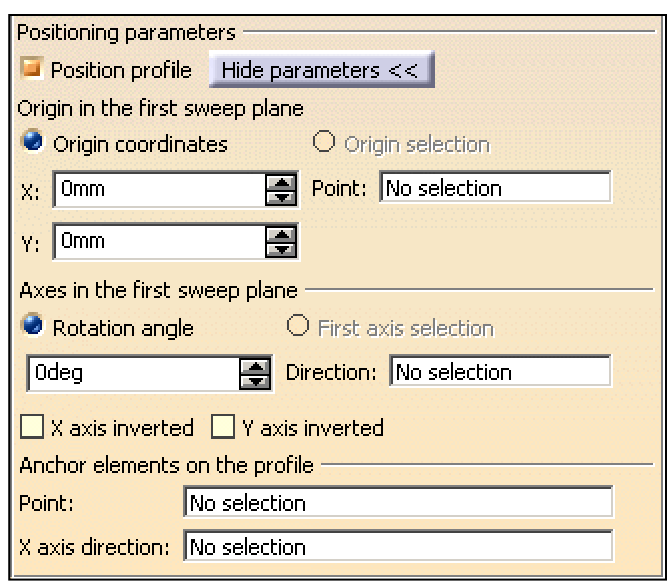
- In the Smooth sweeping section, you can check:
  - the **Angular correction** option to smooth the sweeping motion along the reference surface. This may be necessary when small discontinuities are detected with regards to the spine tangency or the reference surface's normal. The smoothing is done for 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).



4. If you want to manually position the profile, click the **Position profile** check button.

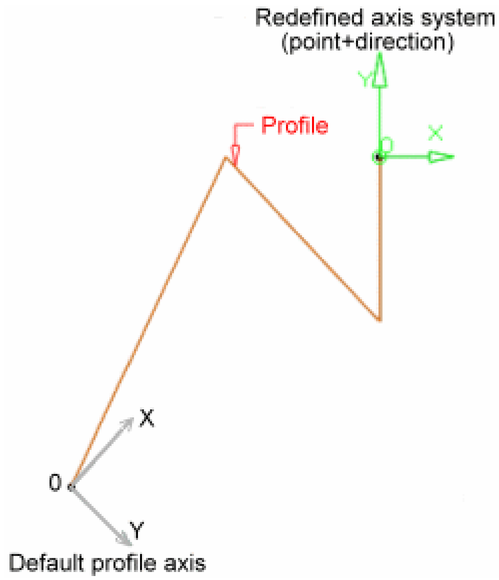
You can then directly manipulate the profile using the graphic manipulators in the geometry, or access positioning parameters clicking on the **Show Parameters>>** button.

These parameters allow you to position the profile in the first sweep plane.

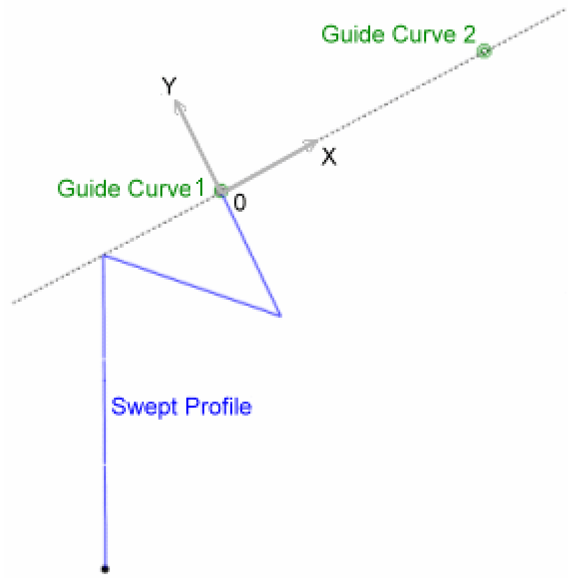


- Specify a positioning point in the first sweep plane by either entering coordinates or selecting a point.
- Specify the x-axis of the positioning axis system by either selecting a line or specifying a rotation angle.
- Select the **X-axis inverted** check box to invert the x-axis orientation (while keeping the y-axis unchanged).
- Select the **Y-axis inverted** check box to invert the y-axis orientation (while keeping the x-axis unchanged).
- Specify an anchor point on the profile by selecting a point. This anchor point is the origin of the axis system that is associated with the profile.
- Specify an axis direction on the profile by selection a direction. If no anchor direction was previously defined, the x-axis of the positioning axis system is used to join the extremities of the profile. The x-axis is aligned with the reference surface.

Reference Surface

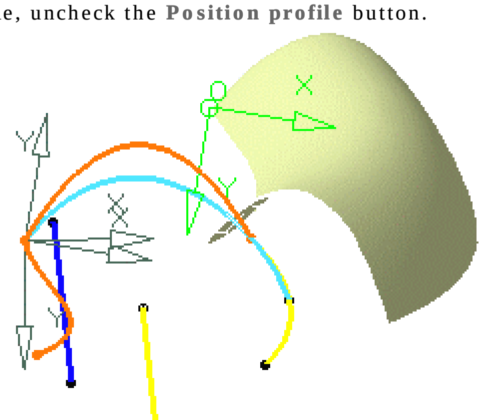


Two Guides



P2 This option is P2 only.

If you want to go back to the original profile, uncheck the **Position profile** button.



5. Click **OK** to create the swept surface.

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

P1 It is not mandatory that the profile be a sketch.

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

Creating Swept Surfaces Using a Linear Profile

This task shows how to create swept surfaces that use an implicit linear profile.

The following subtypes are available:

- Two limits
- Limit and middle
- With reference curve
- With reference surface
- With tangency surface
- With draft direction
- With two tangency surfaces

Open the Sweep1.CATPart document.

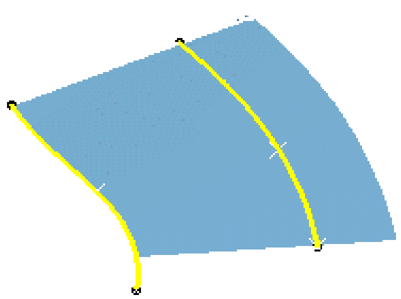
1. Click the **Sweep** icon.

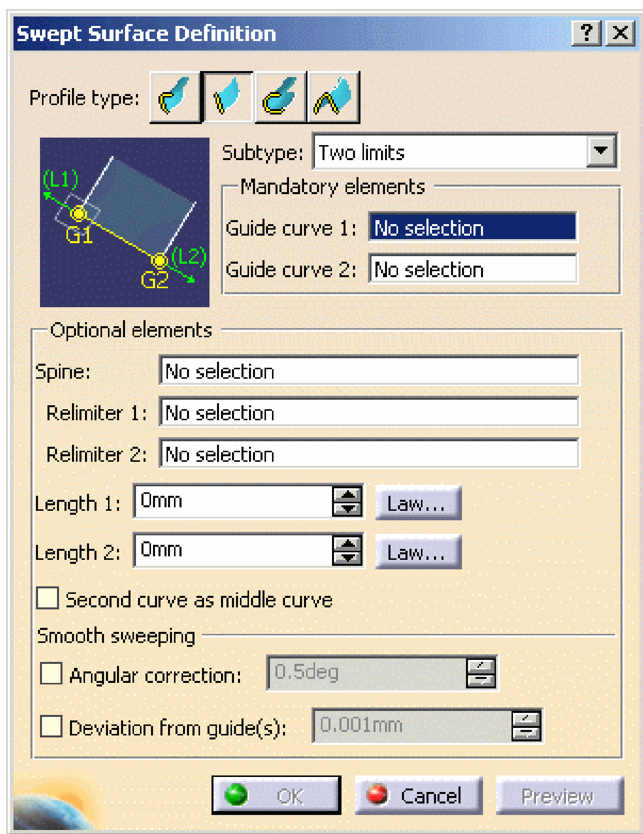
   The Swept Surface Definition dialog box appears.

2. Click the Line profile icon, then use the drop-down list to choose the subtype.

Two limits:

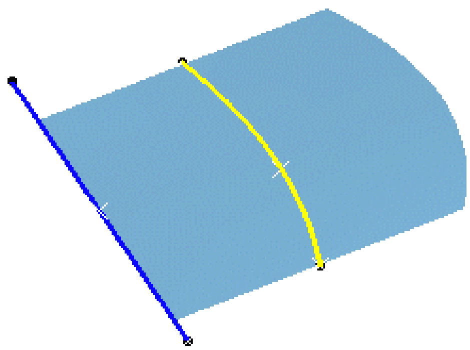
- Select two guide curves.
- You can enter one or two length values to define the width of the swept surface.

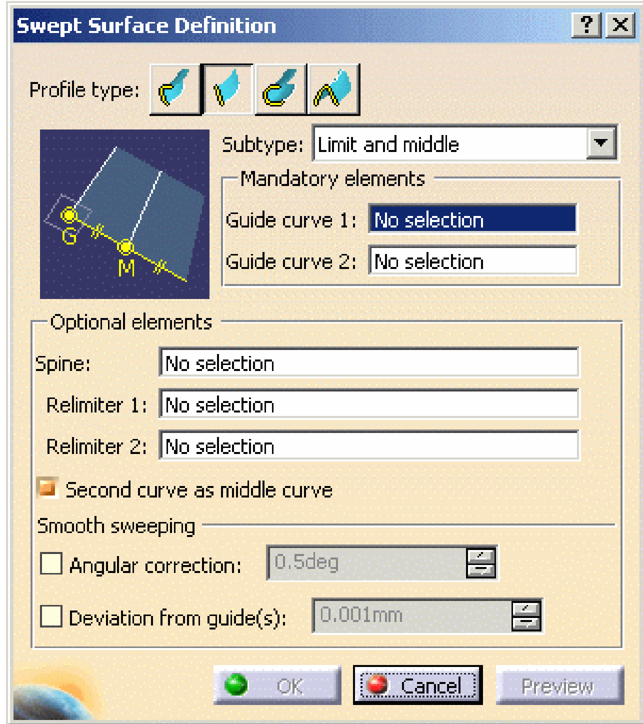




Limit and middle:

- Select two guide curves.
- Select the **Limit and middle** option from the list to use the second guide curve as middle curve.

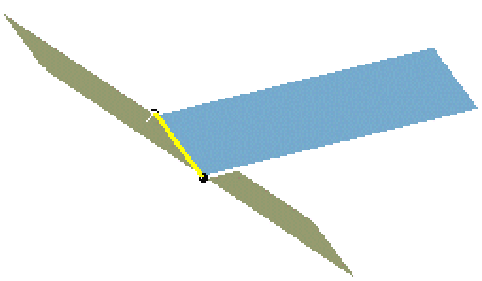


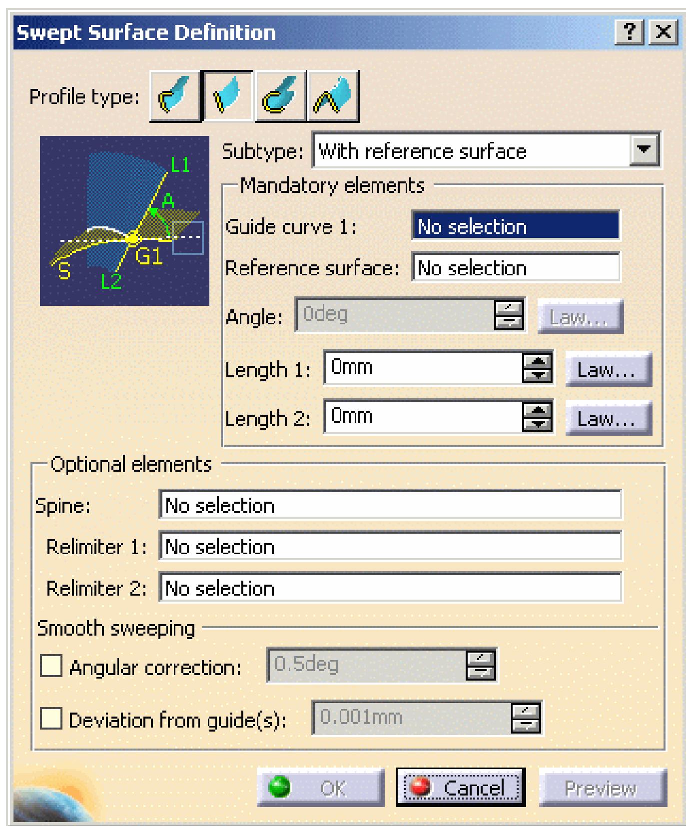


Checking the **Second curve as middle curve** button automatically selects this mode.

With reference surface:

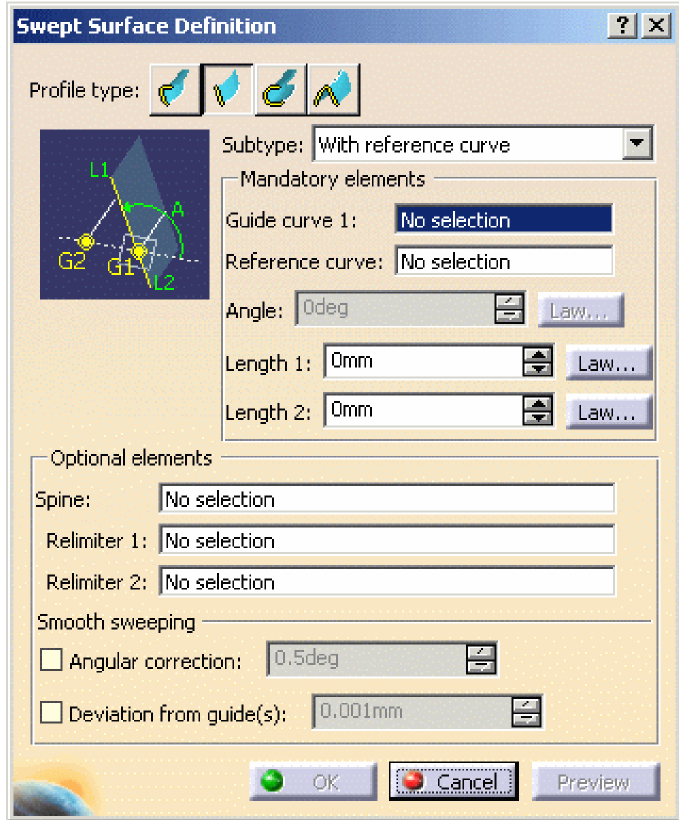
- Select a guide curve, a reference surface, and key in an angle value.
  The guiding curve must lie completely on this reference surface, except if the latter is a plane.
- You can enter one or two length values to define the width of the swept surface.

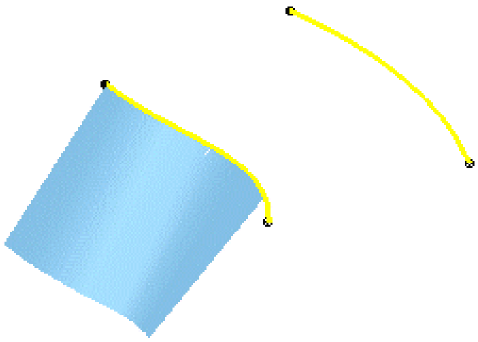




With reference curve:

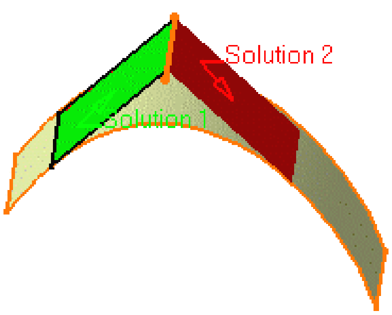
- Select a guide curve, a reference curve, and key in an angle value.
- You can enter one or two length values to define the width of the swept surface.

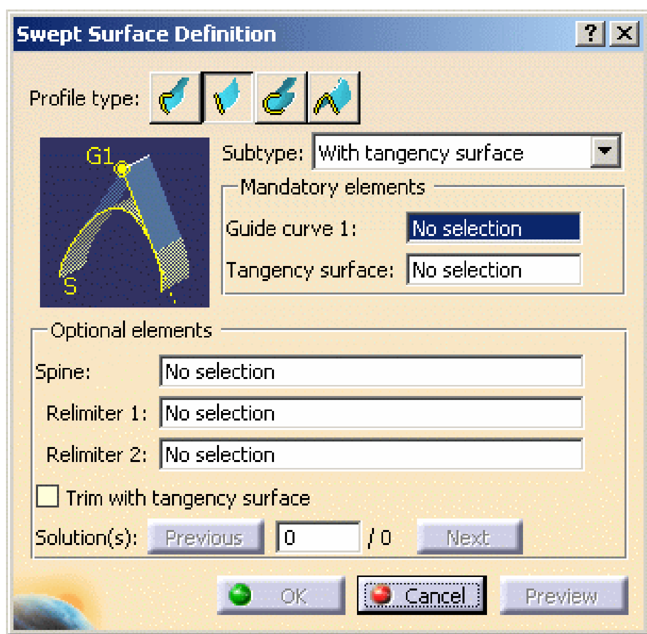




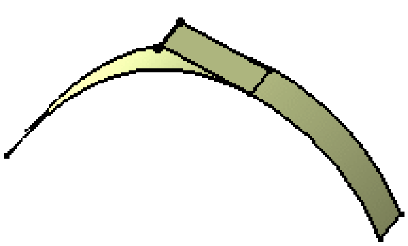
With tangency surface:

- Select a guide curve, and a reference surface to which the sweep is to be tangent.
- Depending on the geometry, there may be one or two solutions from which to choose, either by clicking on the solution displayed in red (inactive) or using the **Next** button.

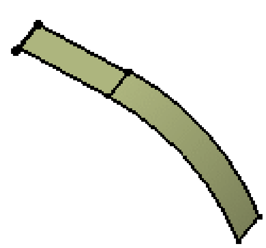




Check the **Trim with tangency surface** to perform a trim between the swept surface and the tangency surface. The part of the tangency surface that is kept is chosen so that the final result is tangent.



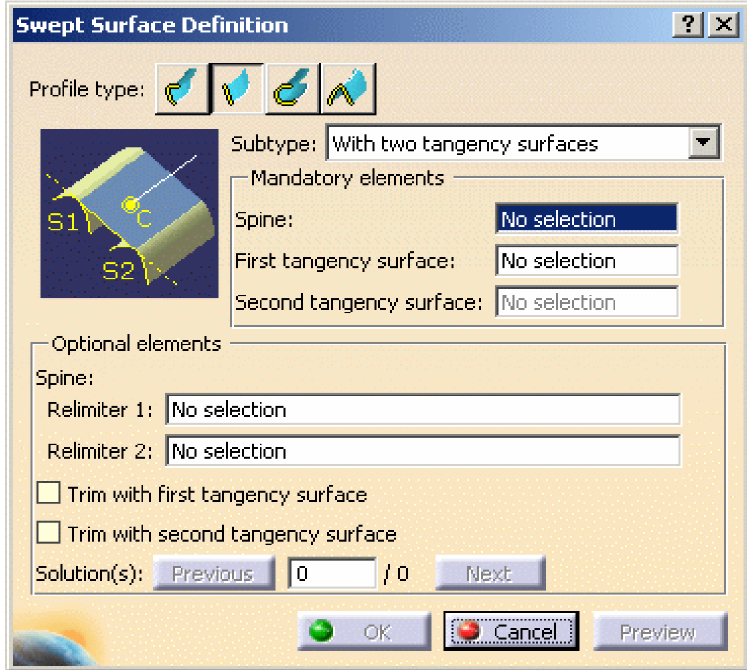
Choosing Solution 2



Solution 2 with Trim option

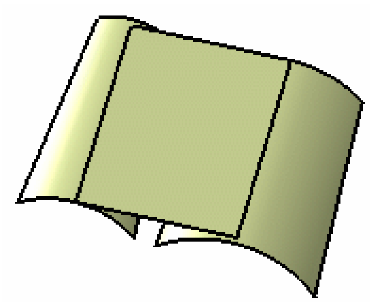
With two tangency surfaces:

Open the Sweep5.CATPart document.

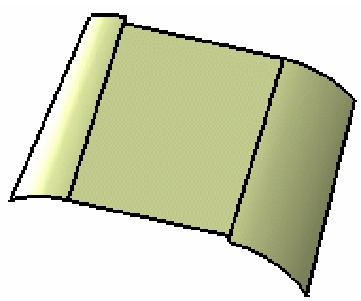


- Select a spine, and two tangency surfaces.

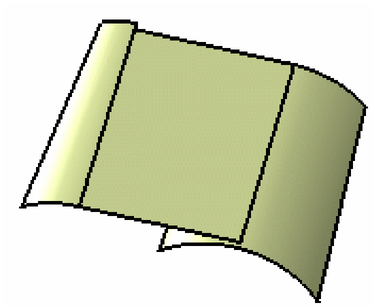
Check the **Trim with tangency surface** to perform a trim between the swept surface and the tangency surface. The part of the tangency surface that is kept is chosen so that the final result is tangent.



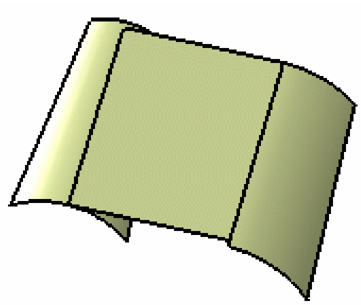
Swept surface without trim



Trim with both surfaces



Trim with first tangency surface



Trim with second tangency surface

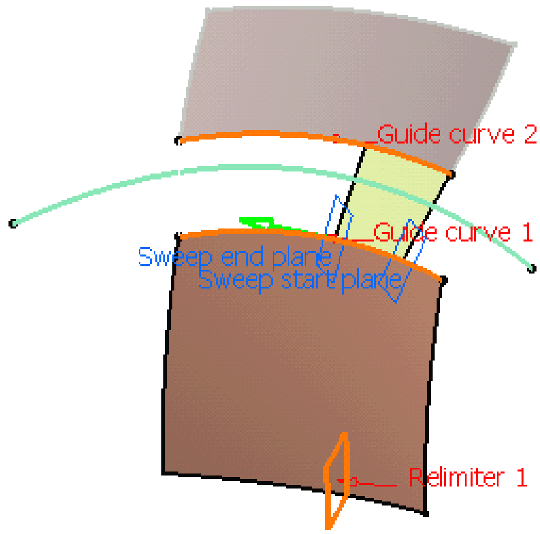
In any of the above cases, you can select a spine using the **Spine** field if you want to specify a spine different from the first guide curve. If no spine is selected, the guide curve is implicitly used as the spine.

Defining Relimiters

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.

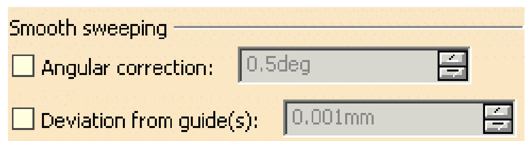
Besides is an example with a plane as Relimiter 1.
When there is only one relimiter, you are able to choose the direction of the sweep by clicking the green arrow.

- 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 relimit the default spine, thus avoiding to split it to create the sweep.
- You can stack the creation of the elements by using the contextual menu available in either field



This option is available with the Draft direction sweep type.

- 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). A curve smooth is performed using correction default parameters in tangency and curvature.
    This option is not available for with tangency surface subtype.



3. Click **OK** to create the swept surface.

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

Click the **Law** button if you want a specific law to be applied rather that the absolute angle value. See Defining Laws for Swept Surfaces.

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

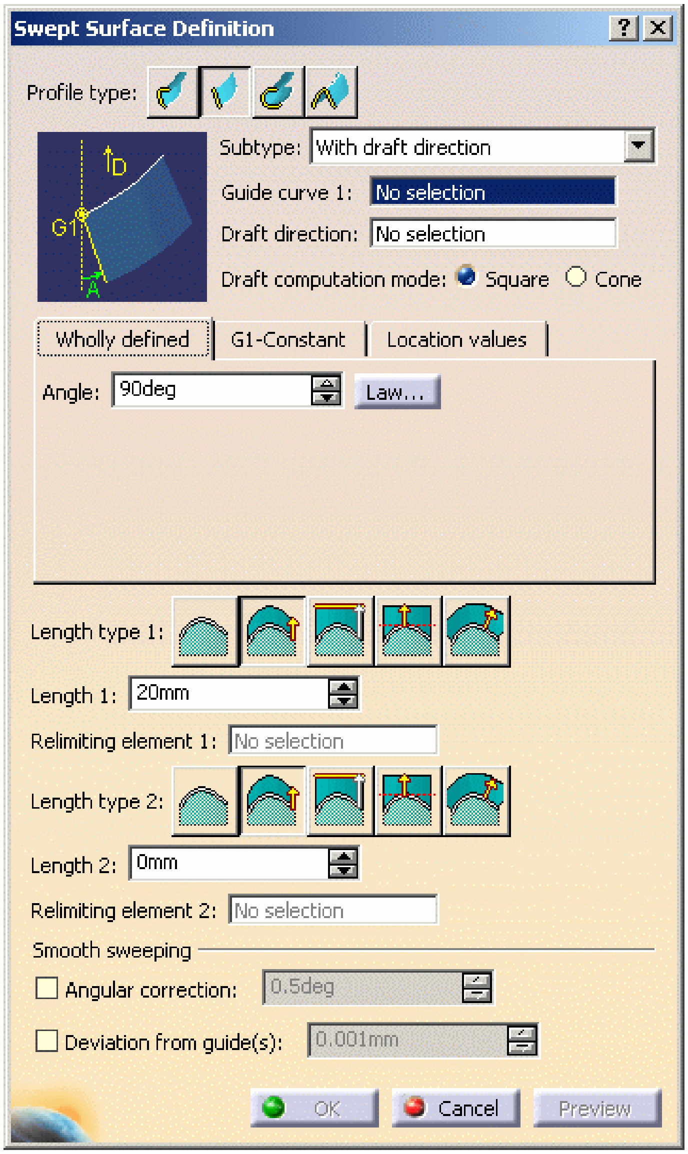
With draft direction:

Open the Sweep6.CATPart document.

- Select a guide curve and a draft direction (a line, a plane or components),
  - Select the draft computation mode:
    - **Square**: equivalent to implicit linear profile swept surface with reference surface, using a plane normal to the draft direction as reference surface, and the projection of the guide curve onto this plane as spine
    - **Cone**: envelop of cones defined along a given curve. In order to have swept start and end planes similar

as the square mode, the guide curve needs to be extrapolated and the resulting surface split as explained in the following figure.

- Choose the angular definition:
  - **Wholly defined**: the angular value varies during the whole sweeping operation
  - **G1-Constant**: a different draft value for every G1 section can be set; in this case, a relimiting plane is requested when defining lengths
  - **Location values**: on given points on the curve, angular values can be defined.



In the **Wholly defined** tab, you can click the **Law...** button to display the Law Definition dialog box. The 2D viewer enables you to previsualize the law evolution before applying it.

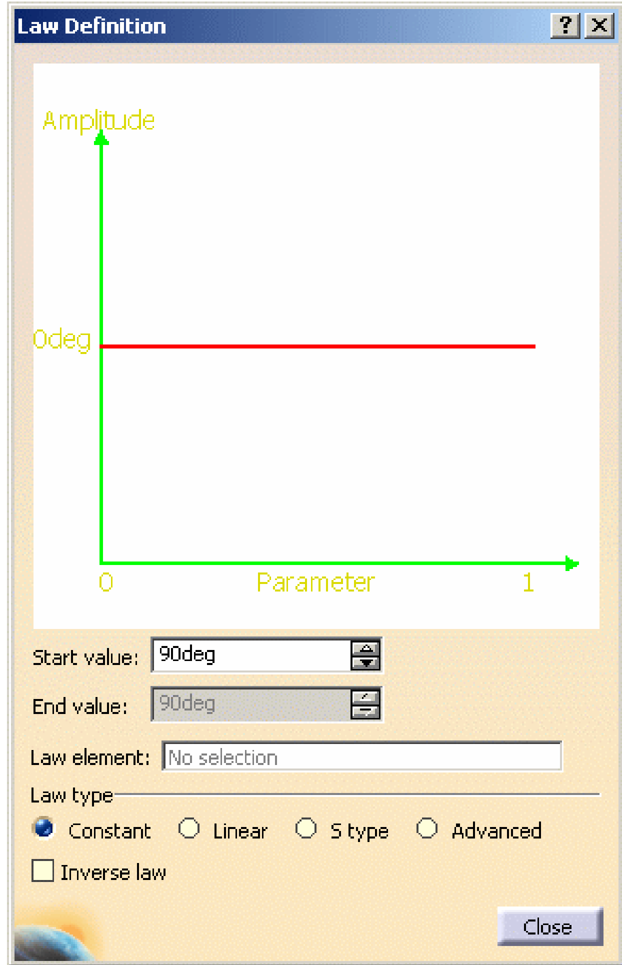
1. Enter **Start** and **End values**.
2. Choose the law type to be applied to the pitch.
   Four law types are available:

a. **Constant**: a regular law, only one value is needed.
b. **Linear**: a linear progression law between the **Start** and **End** indicated values
c. **S type**: an S-shaped law between the two indicated values
d. **Advanced**: allowing to select a Law element as defined in Creating Laws.

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.

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)).

3. Check the **Inverse law** button to reverse the law as defined using the above options.
4. Click **Close** to return to the Swept Surface Definition dialog box.

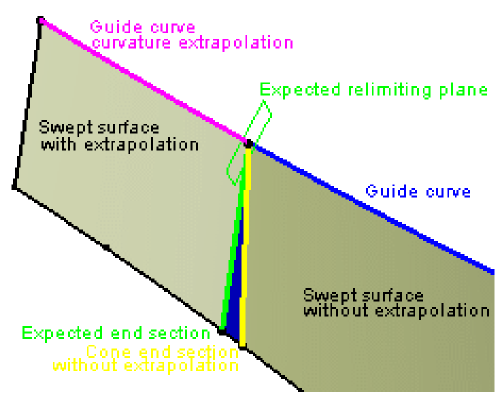


This option is available with both **Square** and **Cone** computation mode.

- Choose the length types:
  - **From curve**: the swept surface starts from the curve
  - **Standard**: the length is computed in sweeping planes
  - **From/Up to**: the length is computed by intersecting a plane or a surface; a point can be selected: a plane parallel to the draft plane would be computed
  - **From extremum**: the lengths are defined along the draft direction from an extremum plane; L1 corresponds to the "maximum plane" in the draft direction, L2 corresponds to the "minimum plane" in the draft direction
  - **Along Surface**: The boundary opposite to the guide curve is computed: it is the exact euclidean parallel curve of the guide curve.

The location value tab is only available for a square computation mode and will work only on G1 curves.

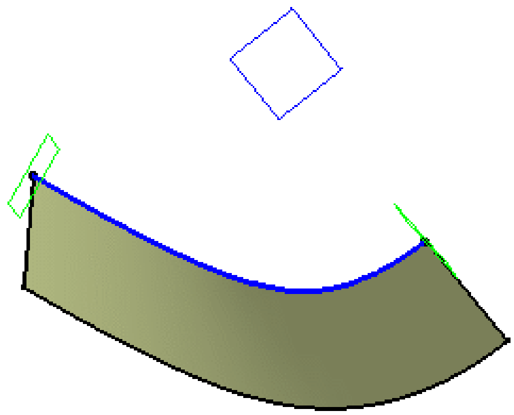
The start (or end) section of the swept surface (in yellow) does not coincide with the expected relimiting plane (in green). As a consequence, the blue portion needed is missing.



Here are the steps performed to create the swept surface:

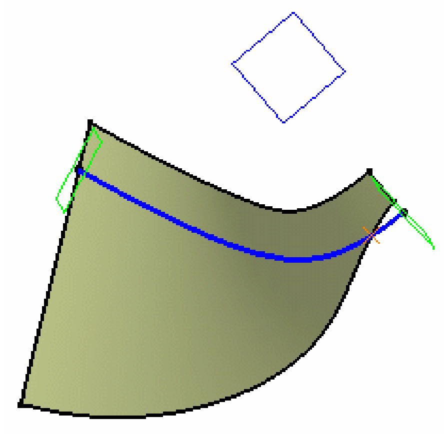
1. The guide curve is extrapolated in curvature (pink curve)
2. The result is split by the green plane to obtain the green end section.

As an information purpose, we put all the elements explaining the steps above in Open_body 2, so that you understand how the sweet surface is created.



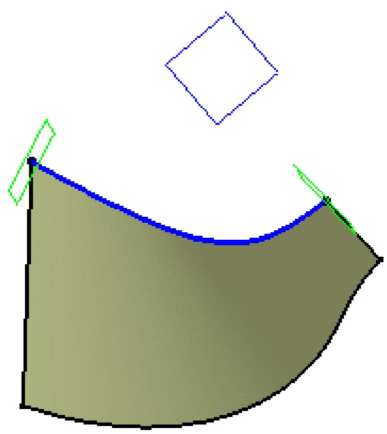
1. In the example above, we selected the following values

Curve.1 as guide curve
Plane.1 as draft direction
Square as computation mode
20deg as Wholly constant angle
Standard Length type
50mm as Length 1



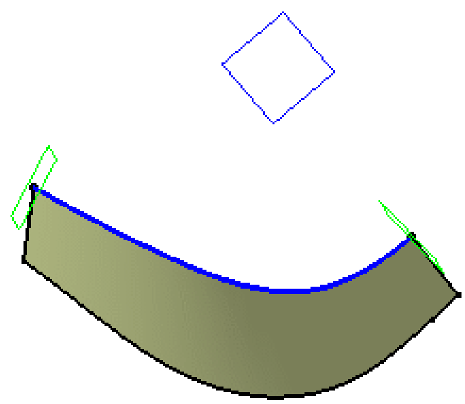
2. In the example above, we selected the following values

Curve.1 as guide curve
Plane.1 as draft direction
Square as computation mode
35deg as Wholly constant angle
From / Up To
Point 1 as Relimiting element 1
20mm as Length 2



3. In the example above, we selected the following values

Curve.1 as guide curve
Plane.1 as draft direction
Cone as computation mode
25deg as Wholly constant angle
From Extremum type
50mm as Length 1



4. In the example above, we selected the following values

Curve.1 as guide curve
Plane.1 as draft direction
Square as computation mode
20deg as Wholly constant angle
Along surface type
30mm as Length 1

Creating Swept Surfaces Using a Circular Profile

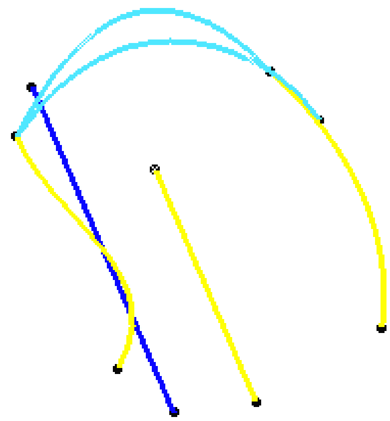
This command is only available with the Generative Shape Design 2 product.

This task shows how to create swept surfaces that use an implicit circular profile.

The following subtypes are available:

- Three guides
- Two guides and radius
- Center and two angles
- Center and radius
- Two guides and tangency surface
- One guide and tangency surface

You can use the wireframe elements shown in this figure.



Open the Sweep1.CATPart document.

1. Click the **Sweep** icon.

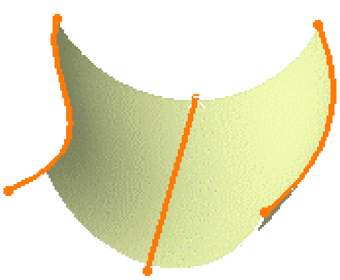
The Swept Surface Definition dialog box appears.

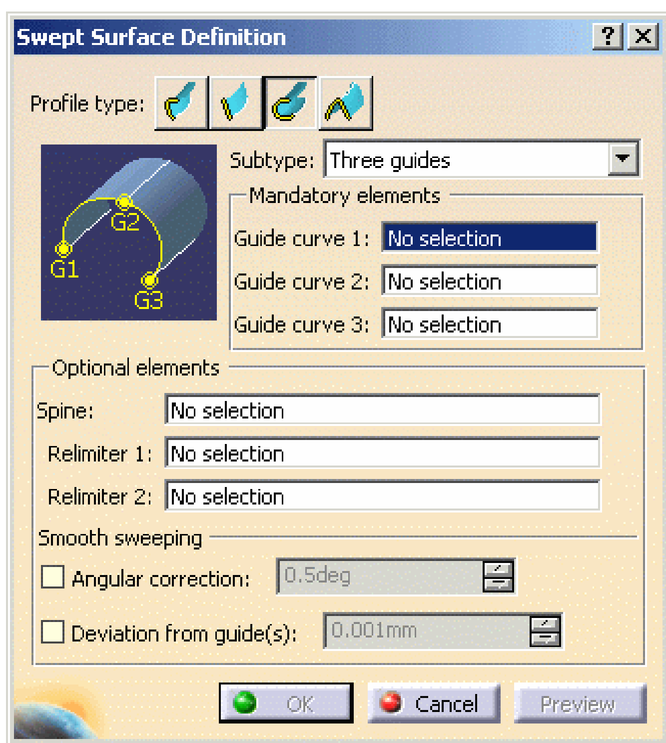
2. Click the **Circle** icon, then use the drop-down list to choose the subtype.

The two following cases are possible using guide curves.

Three guides:

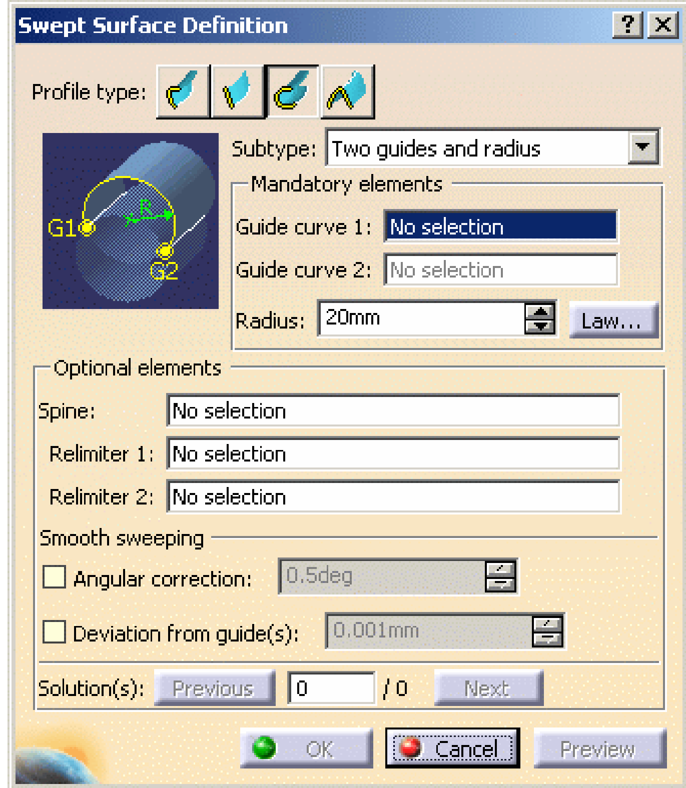
- Select three guide curves.

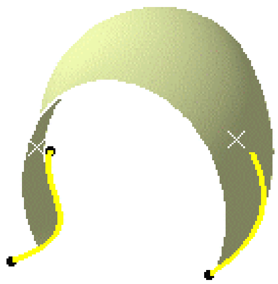




Two guides and radius:

- Select two guide curves and enter a **Radius** value.
  You can then choose between four possible solutions by clicking the **Other Solution** button.



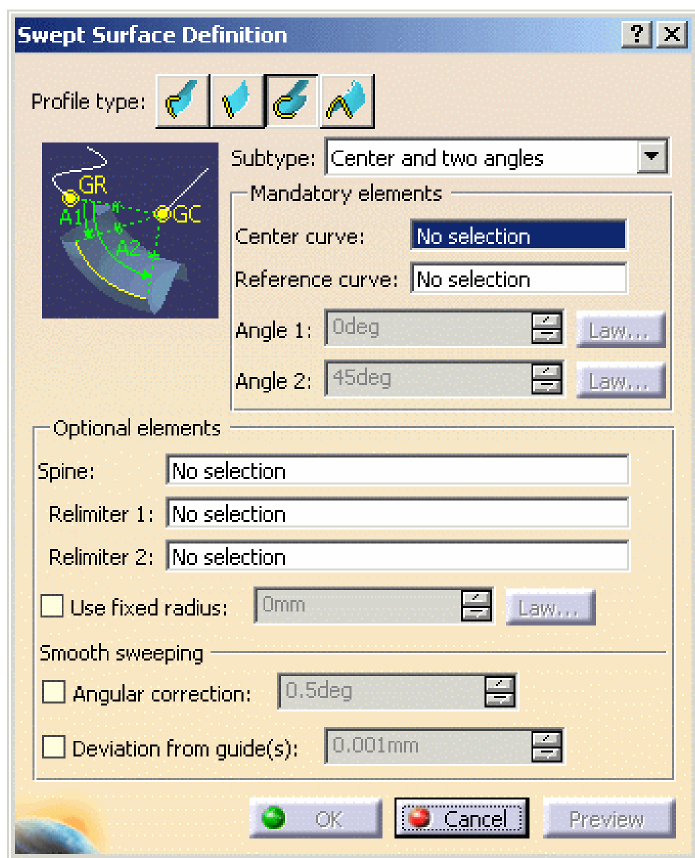


In the example above, the radius value is 45.

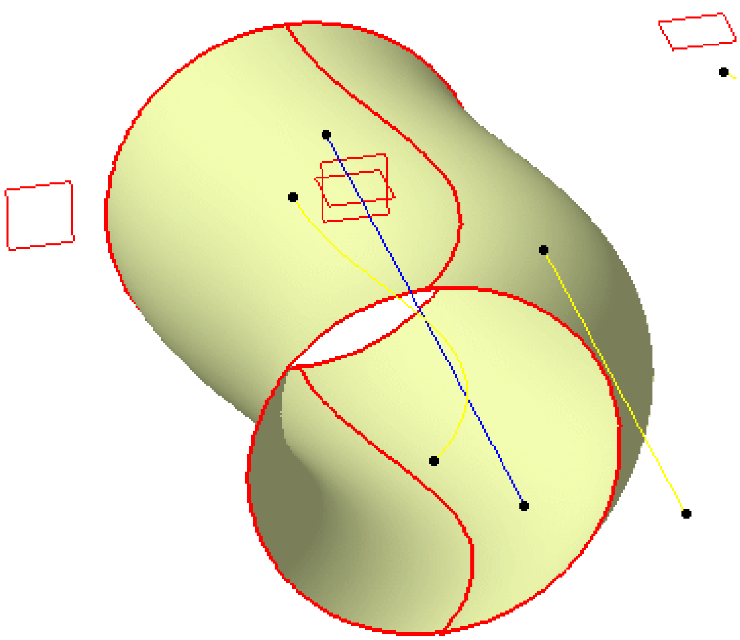
The two following cases are possible using a center curve.

Center and two angles:

- Select a **Center Curve** and a **Reference angle** curve.
  You can relimit the swept surface by entering two angle values.

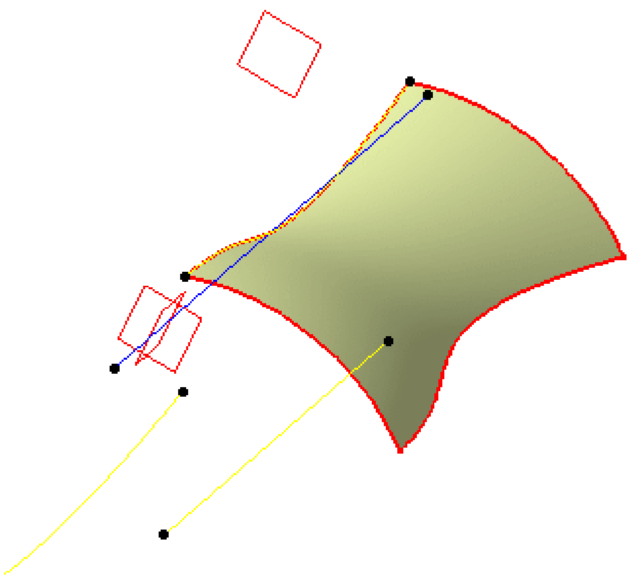


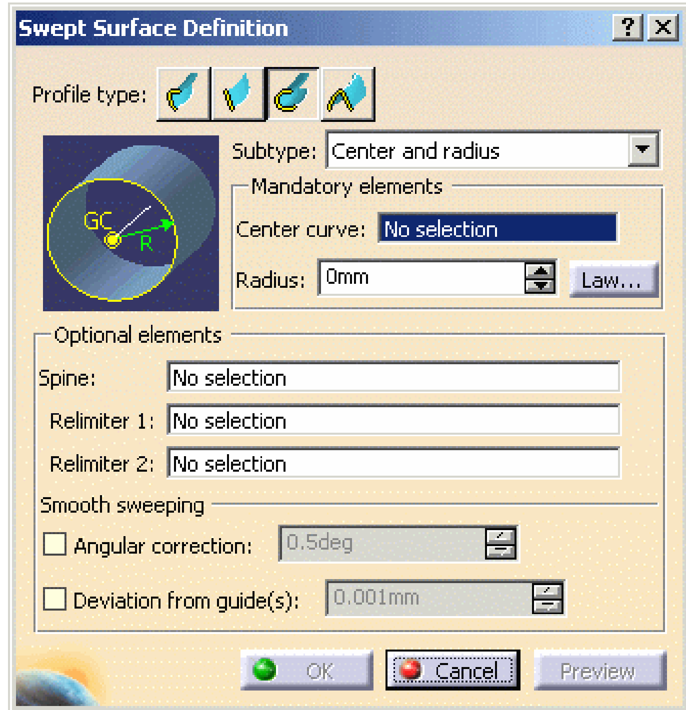
In the example above, we selected a spine



Center and radius:

- Select a **Center Curve** and enter a **Radius** value.





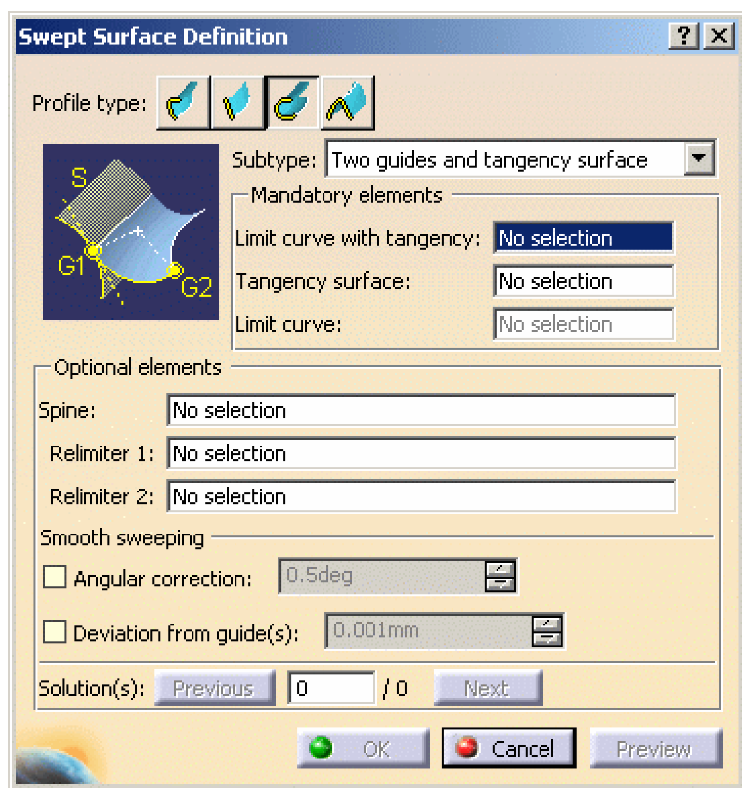
In the example above, we selected the following values:

Center curve: DemoGuide 3
Reference angle: DemoGuide 1
Angle 1: 0 deg
Angle 2: 60 deg

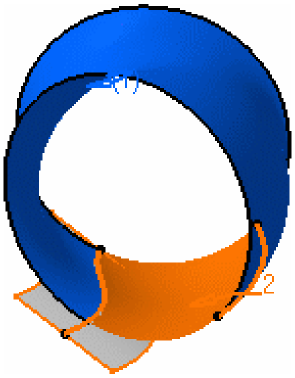
The two following cases are possible using a reference surface to which the swept surface is to be tangent:

Two guides and tangency surface:

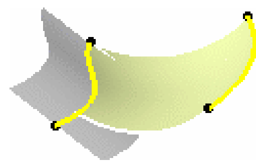
- Select two guide curves, and a reference surface to which the sweep is to be tangent.



- Depending on the geometry, there may be one or two solutions from which to choose. The solution displayed in red shows the active sweep.



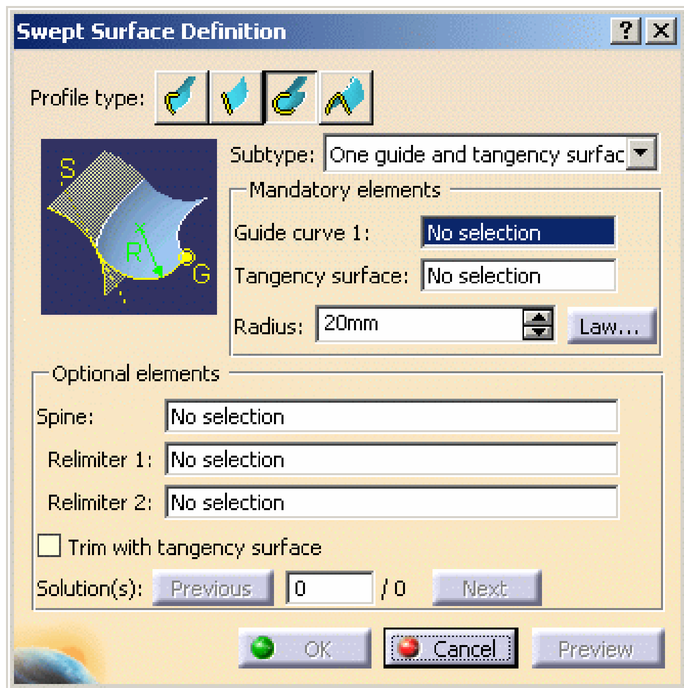
Choosing a solution



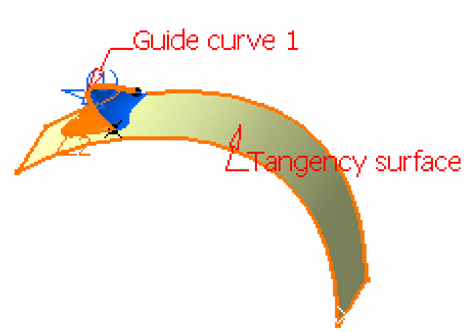
Resulting Sweep

One guide and tangency surface:

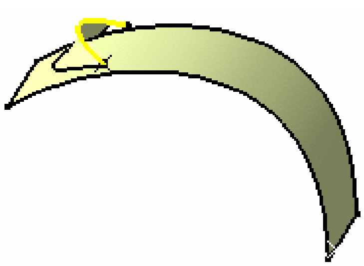
- Select a guide curves, a reference surface to which the sweep is to be tangent, and enter a radius value.



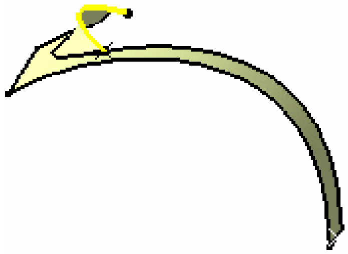
Check the **Trim with tangency surface** to perform a trim between the swept surface and the tangency surface. The part of the tangency surface that is kept is chosen so that the final result is tangent.



Choosing a solution



Resulting Sweep



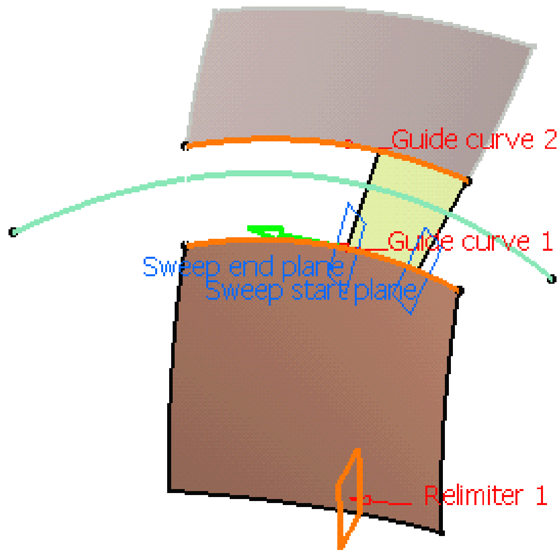
Sweep 2 with Trim option

In any of the above cases, you can select a spine using the **Spine** field if you want to specify a spine different from the first guide curve or center curve.

Defining Relimiters

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 relimit the default spine, thus avoiding to split it to create the sweep.
- 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).
  A curve smooth is performed using correction default parameters in tangency and curvature.
  This option is not available for One guide and tangency surface subtype.

3. Click **OK** to create the swept surface.

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

Click the **Law** button if you want a specific law to be applied rather that the absolute angle value.
See Defining Laws for Swept Surfaces.

Creating Swept Surfaces Using a Conical Profile

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

This task shows how to create swept surfaces that use an implicit conical profile, such as parabolas, hyperbolas or ellipses
These swept surfaces are created based on guide curves and tangency directions. The latter can be defined either by the supporting surface or a curve giving the direction.

The following sub-types are available:

- Two guides
- Three guides
- Four guides
- Five guides

Open the Sweep2.CATPart.

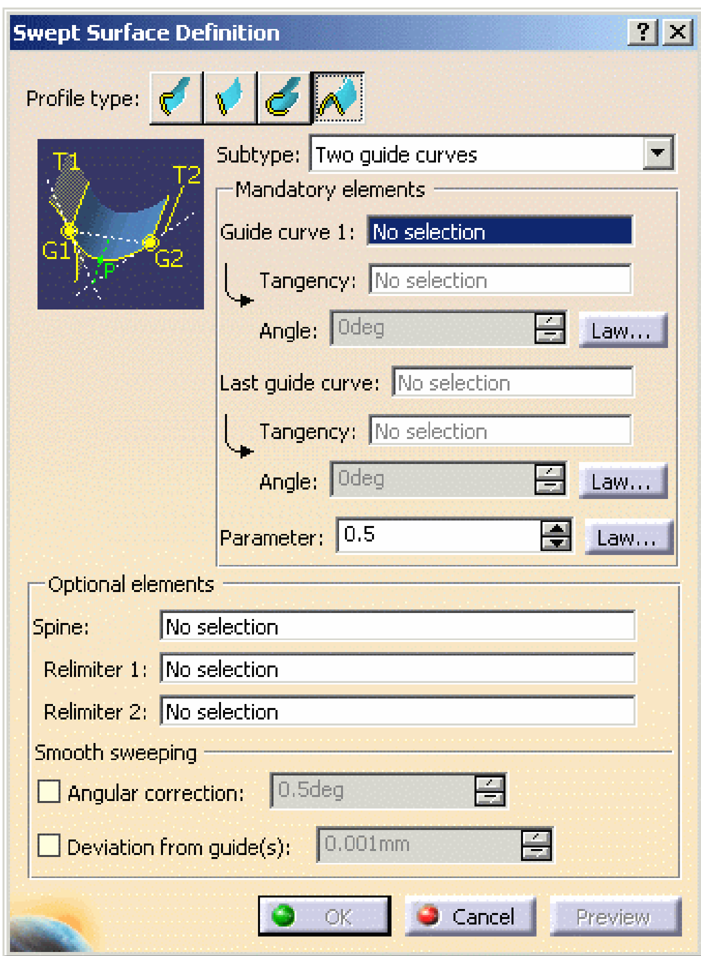
1. Click the **Sweep** icon.

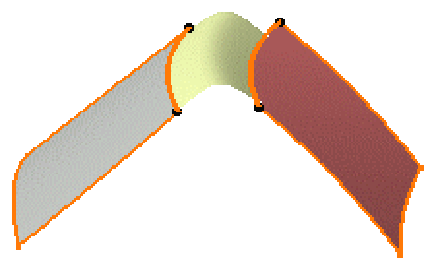
The Swept Surface Definition dialog box appears.

2. Click the Conic icon, then use the drop-down to choose the subtype.

Two guides

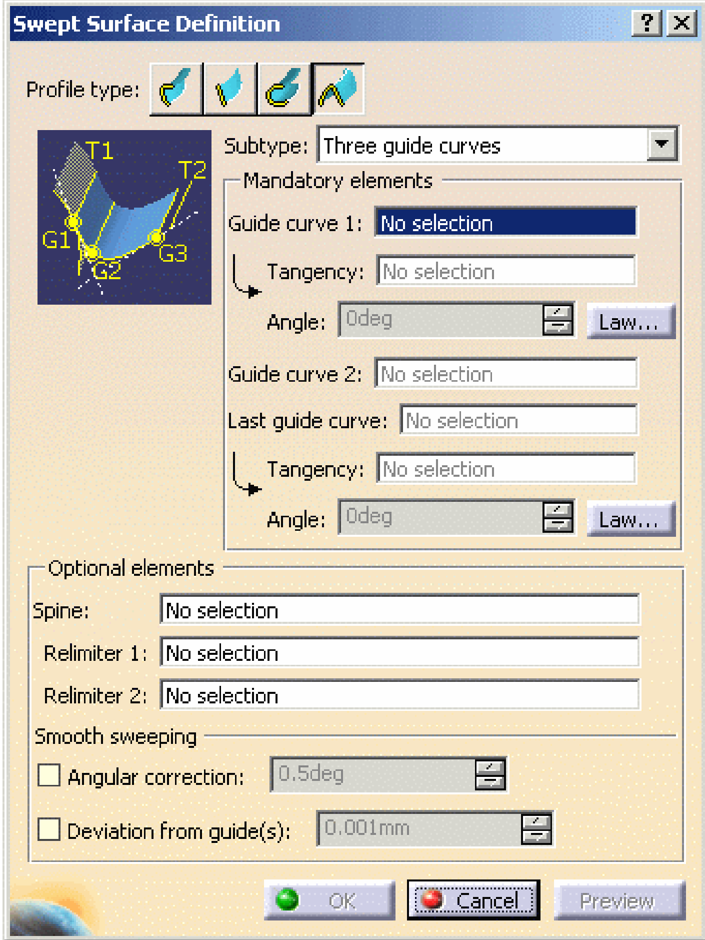
- Select two guide curves and their tangency supports, indicating an angle value in relation to the support, if needed.
- Set the **Parameter** value. It is a ratio ranging from 0 to 1 (excluded), and is used to define a passing point as described in Creating Conic Curves and illustrated in the diagram.

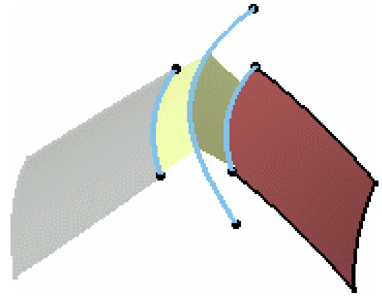




Three guides

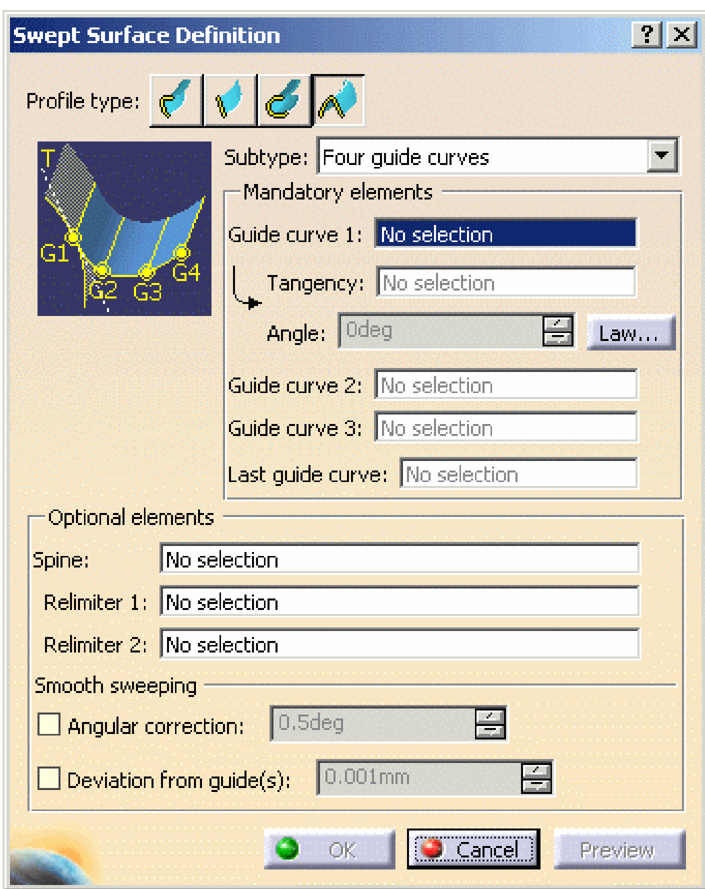
- Select three guide curves, and the tangency supports for the first and last guides. If needed, indicate an angle in relation to the support.

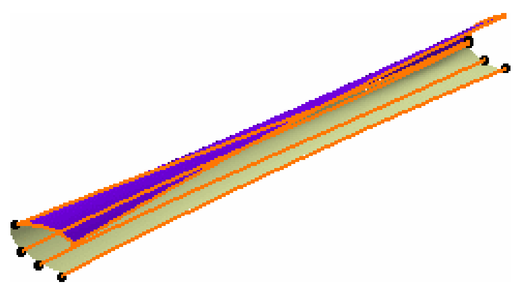




Four guides

- Select four guide curves and the tangency support for the first guide. If needed, indicate an angle in relation to the support.

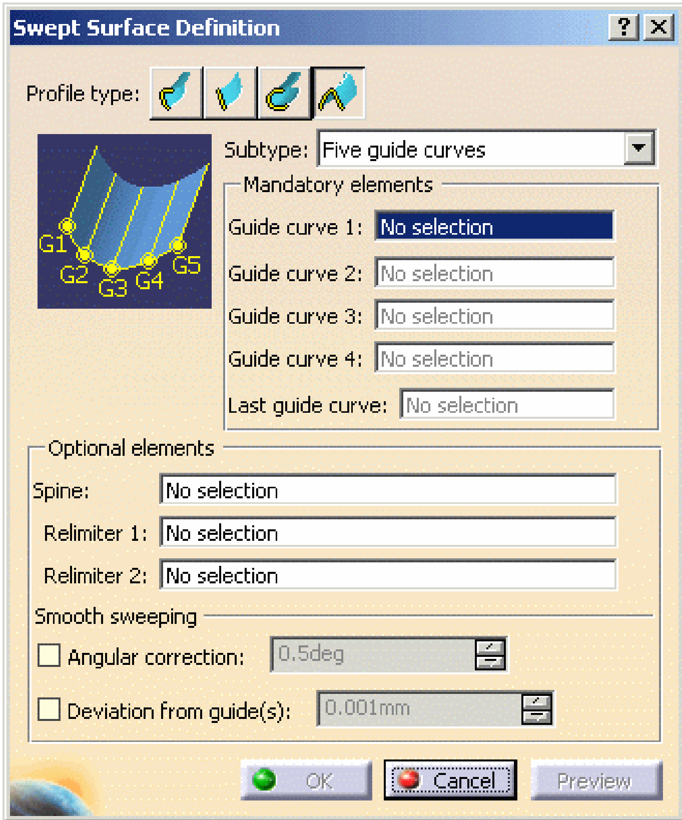


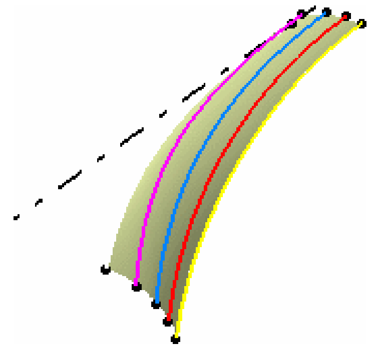


Five guides

Open the Sweep3.CATPart.

- Select five guide curves.

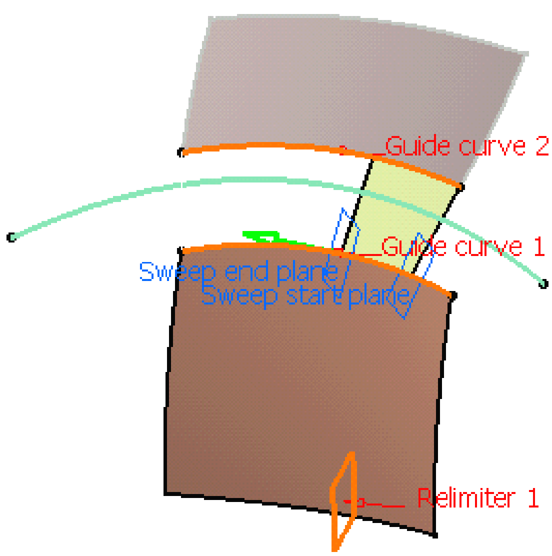




In any of the above cases, you can select a spine using the **Spine** field if you want to specify a spine different from the first guide curve or center curve.

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

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 relimit the default spine, thus avoiding to split it to create the sweep.
- You can stack the creation of the elements by using the contextual menu available in either field.

Provided you are using two, three, or four guides, 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).

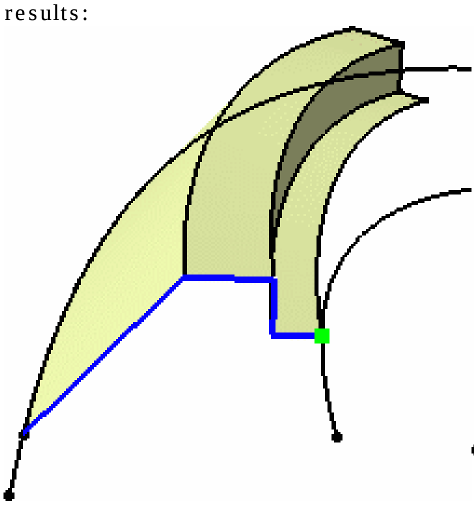
3. Click **OK** to create the swept surface.

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

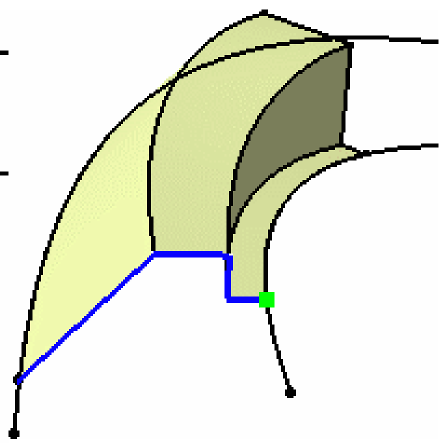
Click the **Law** button if you want a specific law to be applied rather that the absolute angle value. See Defining Laws for Swept Surfaces.

Creating Adaptive Swept Surfaces

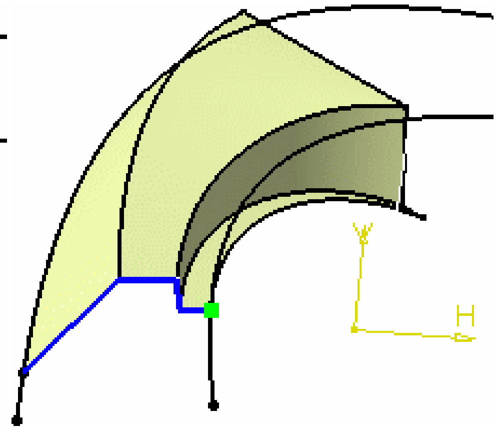
This task shows how to create swept surfaces that use an implicit profile and its constraints along a guiding curve. These swept surfaces are created based on sections along the guiding curve and constraints that can be specified for each of these sections.
The implicit profile is a sketch and as such supports the creation of associative sketch elements over multi-cell surfaces. This allows, when creating the swept surface, to impose a constraint over a multi-cell surface that is use as a construction element.
When designing the profile to be swept, keep in mind that the constraints imposed on the sketched profile affect the resulting swept surface. For example, with the apparently similar sketch (only its construction differs, but there is a coincidence constraint between the sketch extremity and the point on the guiding curve) you can obtain the following results:



Sketch based on the point (no coincidence constraint, but a geometric superimposition)



Sketch based on the point as the intersection of the sketch and the guiding curve



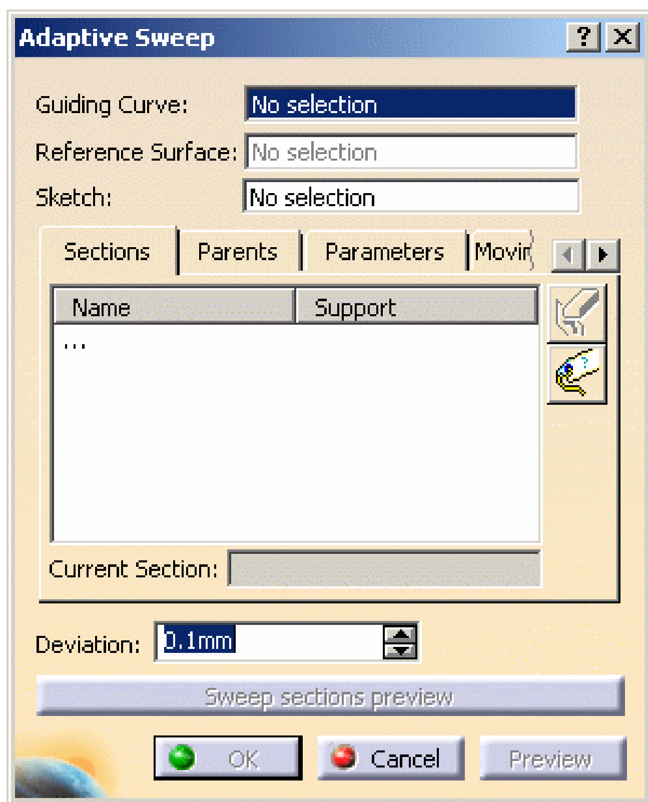
Sketch based on projection of the point in 3D

Similarly, it is best to use angle constraints rather than tangency or perpendicularity constraints, to avoid changes in the sketch orientation as it is swept along the guiding curve. In some cases, with tangency or perpendicularity constraints, the sketch may be inverted and lead to unsatisfactory results.

Open the AdaptiveSweep1.CATPart document.

1. Click the **Adaptive Sweep** icon .

The Adaptive Sweep dialog box appears.



2. Select the **Guiding curve** (Sketch.5 here).

If no guiding curve already exists, use the contextual menu on the **Guiding curve** field to create, either a line, or a boundary.

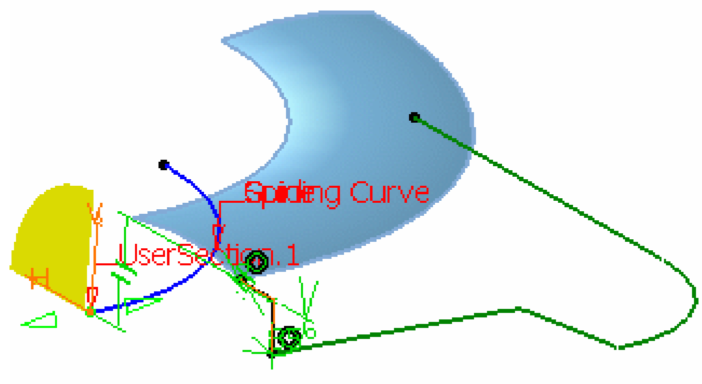
The **Reference surface** is optional. It is the surface on which the guiding curve lies and is used to define the axis system in which the swept surface is created.

In our example, may you wish to define a reference surface, select the xy plane. Otherwise the mean plane is used as default.

If you choose a boundary as the guiding curve, the reference surface automatically is the surface to which the boundary belongs.
You can de-select a reference surface using the Clear Selection contextual menu on the **Reference surface** field.

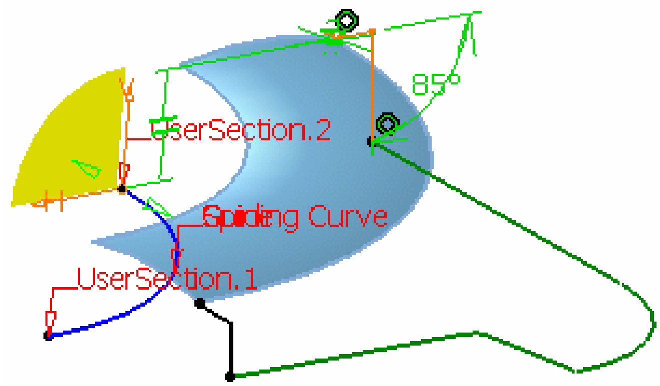
3. Select the **Sketch** to be swept along the guiding curve (Sketch.4 here).

An axis-system is displayed defining the plane in which the first section is created.



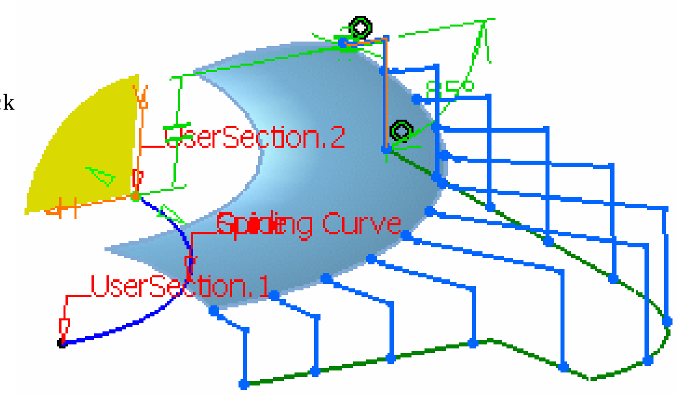
4. Select the end point of the guiding curve to create another section.

The axis-system is displayed at this new section.

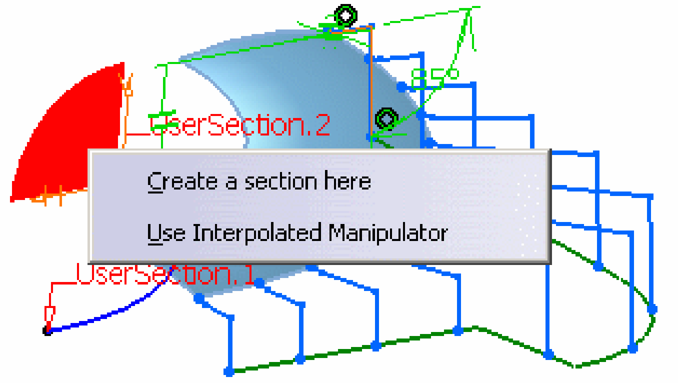


5. Click the **Sweep sections preview** to have a quick wireframe preview of the adaptive sweep surface.

This option lets you see the evolution of the sketch along the guide curve.



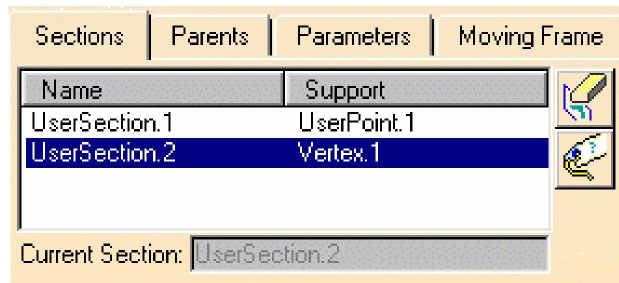
A contextual menu is available on the manipulators:



- **Create a section here**: lets you create a section at the manipulator's place. A new point is dropped on the guide curve with the corresponding ratio. If the guide curve is closed, the created point is a 3D coordinates point.
- **Use Interpolated Manipulator**: the interpolation value between the section parameters is computed. You can move the manipulator along the guide curve to visualize the parameters evolution.

The list in the **Sections** tab is automatically updated with:

- the first section being at the intersection of the selected sketch and guiding curve
- the second section at the selected point on the guiding curve.



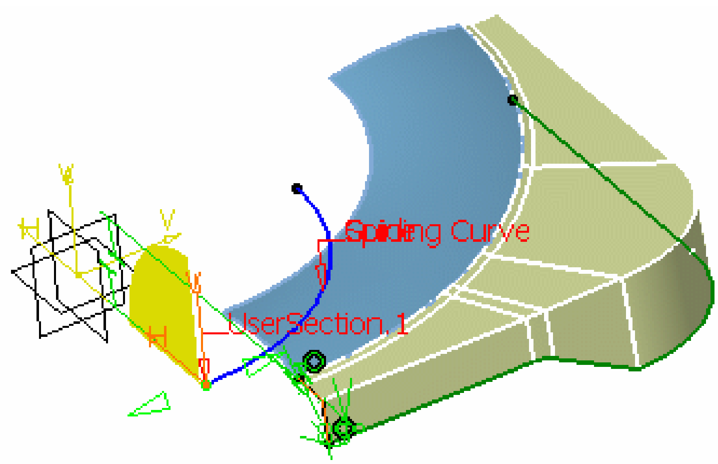
- Use the **Remove current section** icon, or choose the **Remove Section** contextual menu, to delete a section from the swept surface. The first section cannot be deleted.
- Use the **Rename current section** icon, or choose the **Rename Section** contextual menu, to give a new - more explicit - name to any user section.

6. Set the **Deviation** value: it corresponds to a point tolerance.

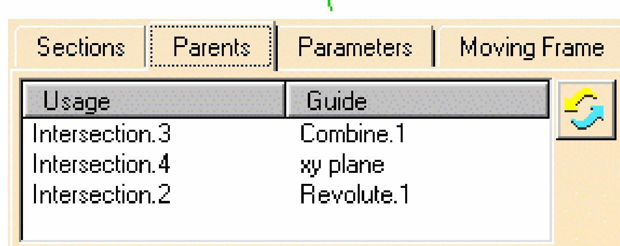
   Decreasing this value increases the precision but leads to slower performances.

   By default the value is 0.1mm (maximum value).

7. Click **Preview** to preview the swept surface:



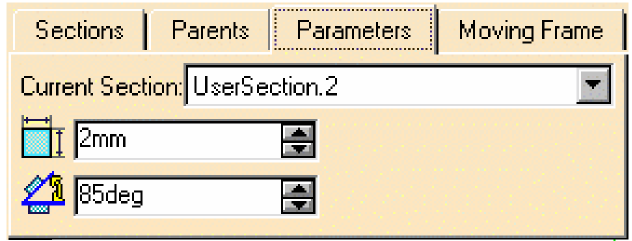
8. Click the **Parent** tab to display the elements making up the sweep.



You can select one of the parents from the list and click the  icon, or choose the **Replace parent** contextual menu to choose a new parent for the swept surface.

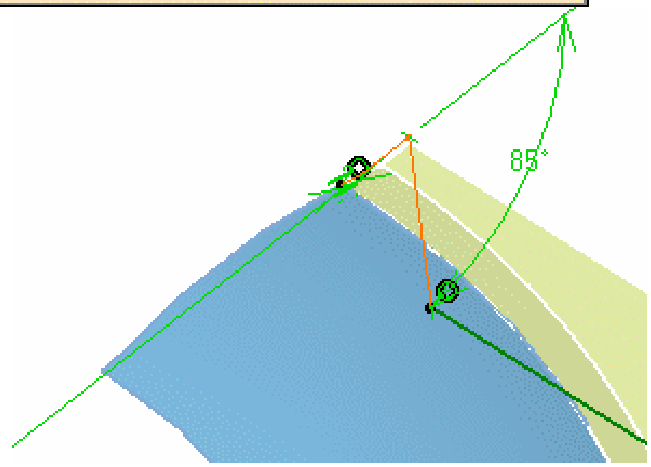
9. Click the **Parameters** tab to display and redefine the constraints on a given section.

   Use the combo to choose Usersection.2.



10. Change the constraint value to 5mm, and click Apply.

The modified sweep is previewed.

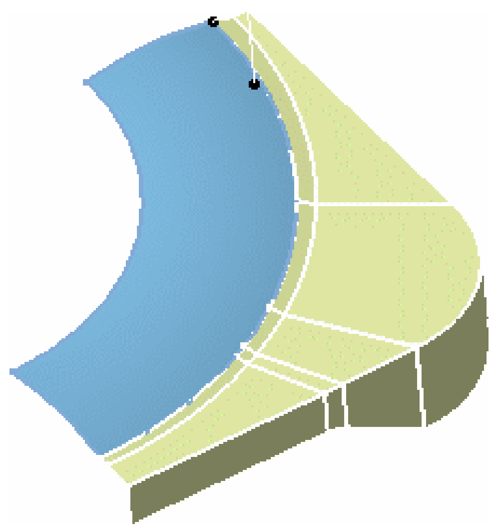


The **Moving Frame** tab is used to display the spine, and possibly select a new one.

Changing to this tab automatically activates the acquisition field of the spine.
By default, the spine is the guiding curve.

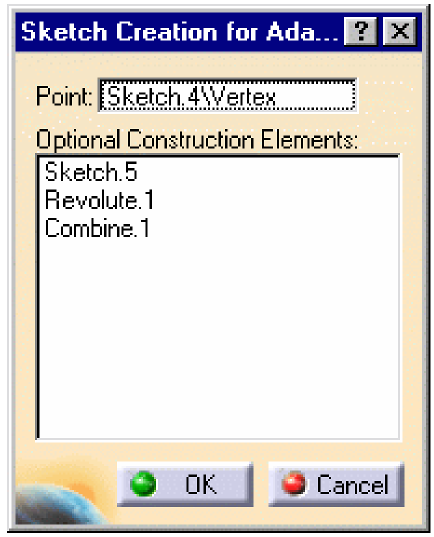
11. Click **OK** to create the swept surface.

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



Once you have selected the guiding curve, you can select an existing sketch or create one using the **Create Sketch** contextual menu on the **Sketch** field to start the sketcher within the adaptive sweep context.

In this case, the Sketch Creation for Adaptive Sweep dialog box is displayed, and allows you to define the construction elements for a new sketch in relation to existing geometry:



1. Select a point, used to position the sketch on the guiding curve, as well as the origin of the sketch.
2. If needed, select construction elements (another guiding curve, support surfaces, and so forth).
3. Click OK.

The selection in the geometry implies a global selection of the 3D elements.

The system automatically loads the Sketcher workbench, and provided the correct option is active, sets the sketch plane parallel to the screen. You can then define a new sketch.

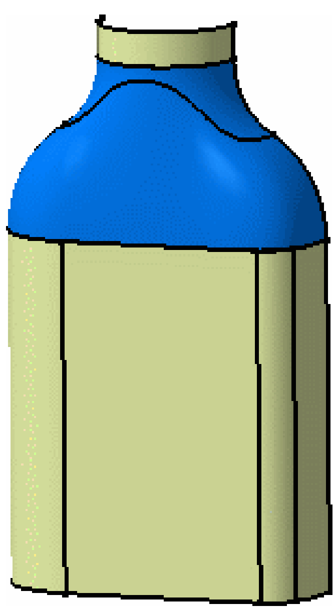
Once you exit the Sketcher, you return to the adaptive sweep command after the sketch selection, as described above in step 3.

This local definition of the sketch is particularly interesting as it allows to redefine the swept surface simply by editing the local sketch (add/remove construction elements, or constraints for example).

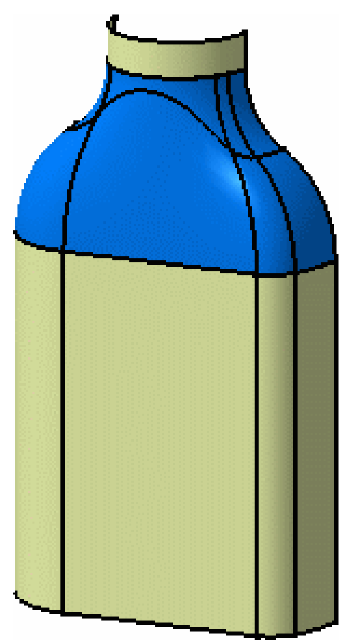
In this case, would you want to exit the Adaptive Sweep command, after having created the sketch using the **Create Sketch** contextual menu, yet retain the sketch itself, simply press the **To cancel the command but keep the sketch** button.

- You also have the possibility to create your sketch using the Sketcher workbench before entering the Sweep command.
  In this case, when you select the 3D construction elements, please be careful to select them directly
- To avoid unsatisfactory surface quality such as gaps between surfaces for example, you can perform one of the following:

1. select a boundary on an adjacent surface as a constraining element when creating the sketch.
   The selection of the boundary allows a better topological splitting, and therefore better quality for the created surface.



Swept surface (blue) without selected boundary



Swept surface (blue) with selected boundary

2. impose more sections along the guiding curve
3. decrease the discretization step value

to better define the sweeping along the guiding curve.

Creating Fill Surfaces

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

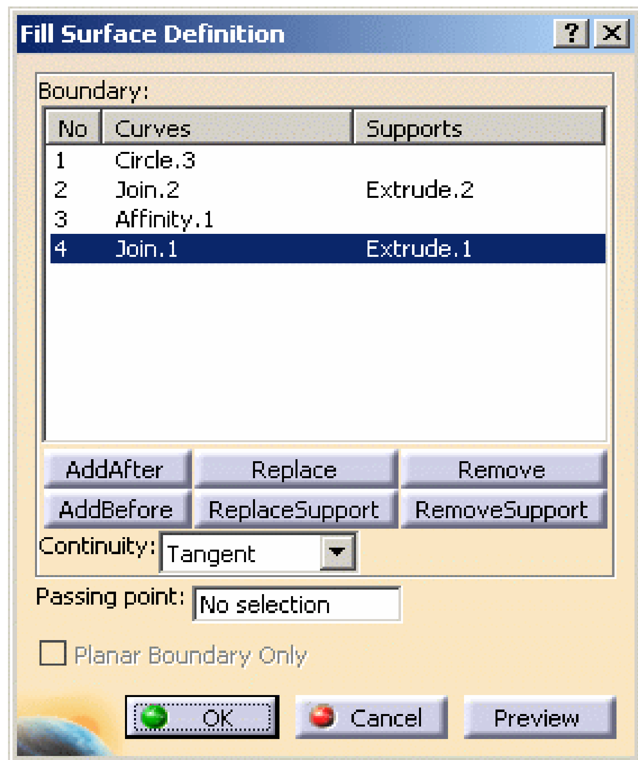
Open the Fill1.CATPart document.

1. Click the **Fill** icon.

The Fill Surface Definition dialog box appears.

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

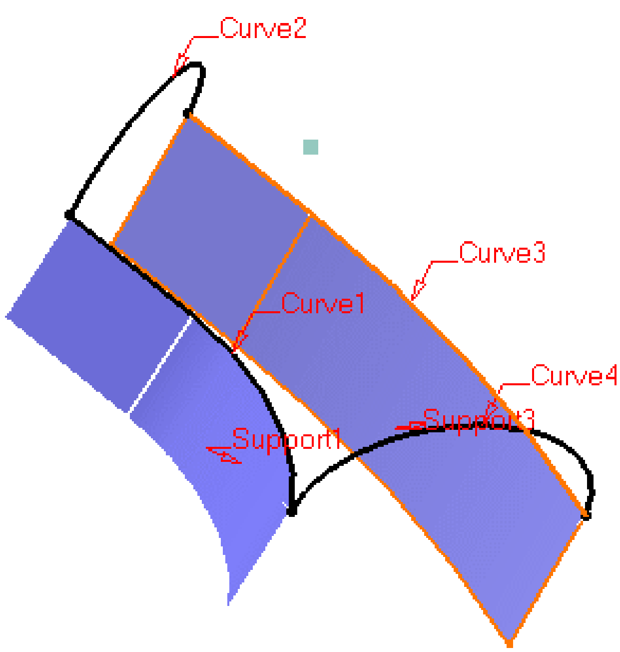
You can select a support surface for each curve or edge. In this case continuity will be assured between the fill surface and selected support surfaces.



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

- **Point**
- **Tangent**, or
- **Curvature** continuity.

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.

5. Click in the **Passing point** field, and select a point.

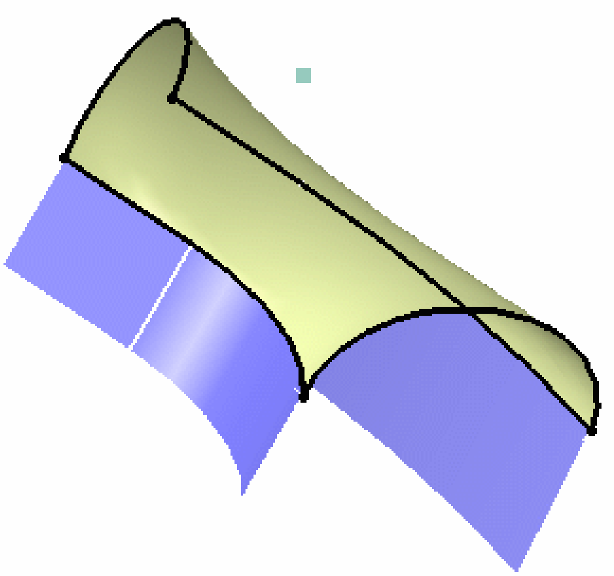
This point is a point through which the filling surface must pass, thus adding a constraint to its creation.
However, you may need to alleviate the number of constraints by removing the supports.

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

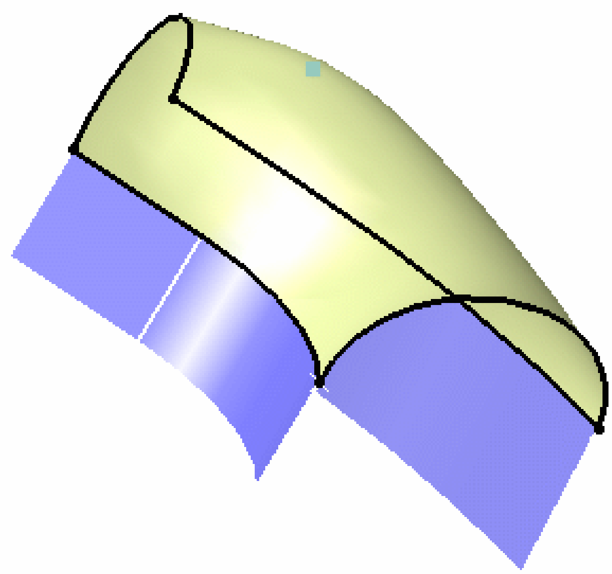
The Planar Boundary Only check box allows you to fill only planar boundaries, when the boundary is defined by one curve on one surface.

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

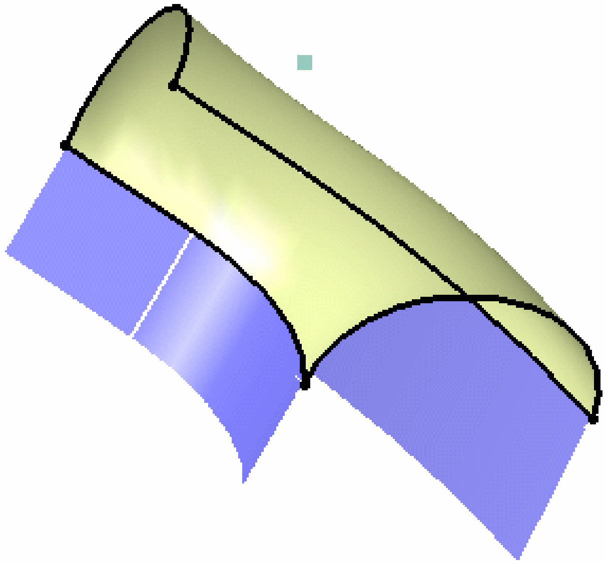
The surface (identified as Fill.xxx) is added to the specification tree.



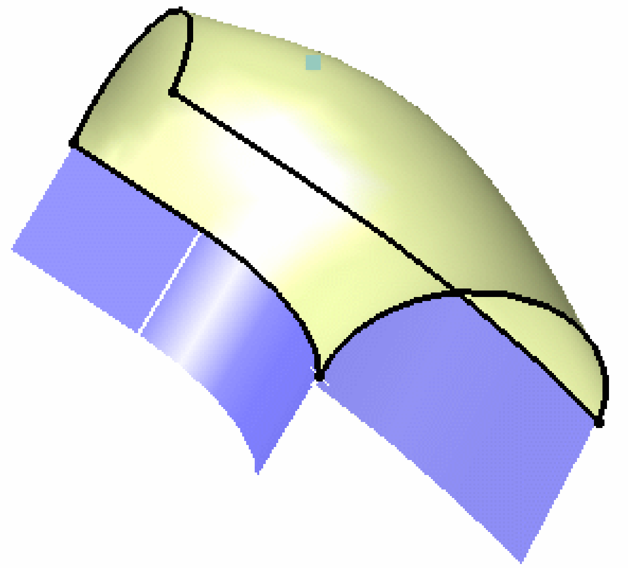
Filling surface without specified supports



Filling surface with a passing point and no specified supports

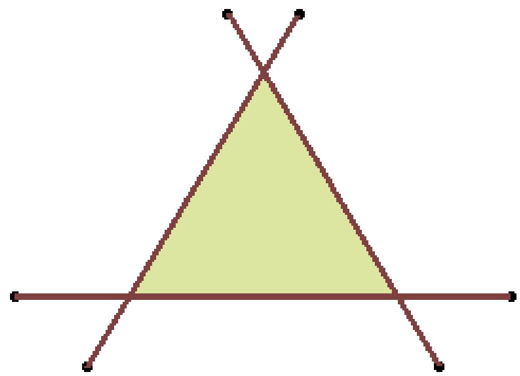


Filling surface with specified supports (Extrude 1 and Extrude 2) and a tangency continuity



Filling surface with a passing point, specified supports (Extrude 1 and Extrude 2) and a tangency continuity

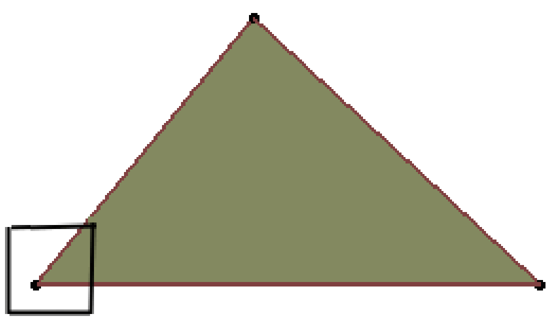
- The selected curves or surface edges can intersect. Therefore a relimitation of the intersecting boundaries is performed to allow the creation of the fill surface.

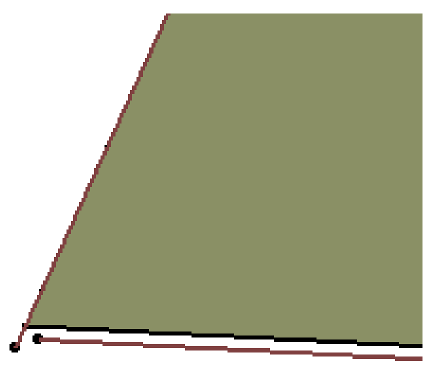


Two consecutive boundaries must have only one intersection.
Selected curves cannot be closed curves.

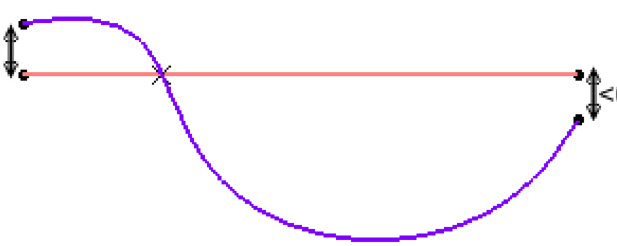
- The selected curves or surfaces edges can have non-coincident boundaries. Therefore, an extrapolation is performed to allow the creation of the fill surface.

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



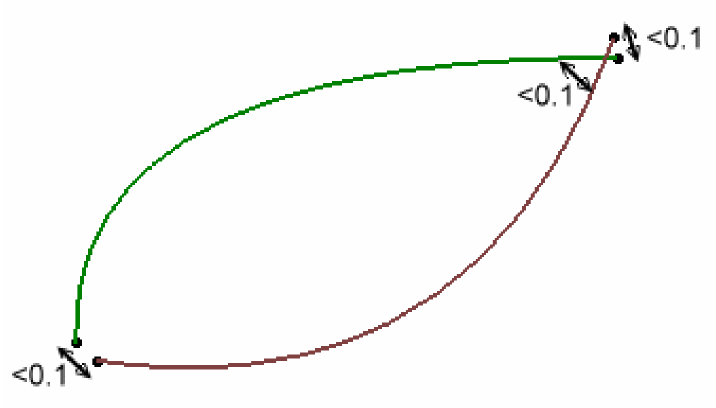


A two-side fill surface cannot be created in the following ambiguous cases:



- one intersection and two distances below 0.1 mm

- no true intersection (therefore there may be several distances below 0.1 mm)



Creating Multi-sections Surfaces

This task shows how to create a multi-sections surface and includes the following functionalities:

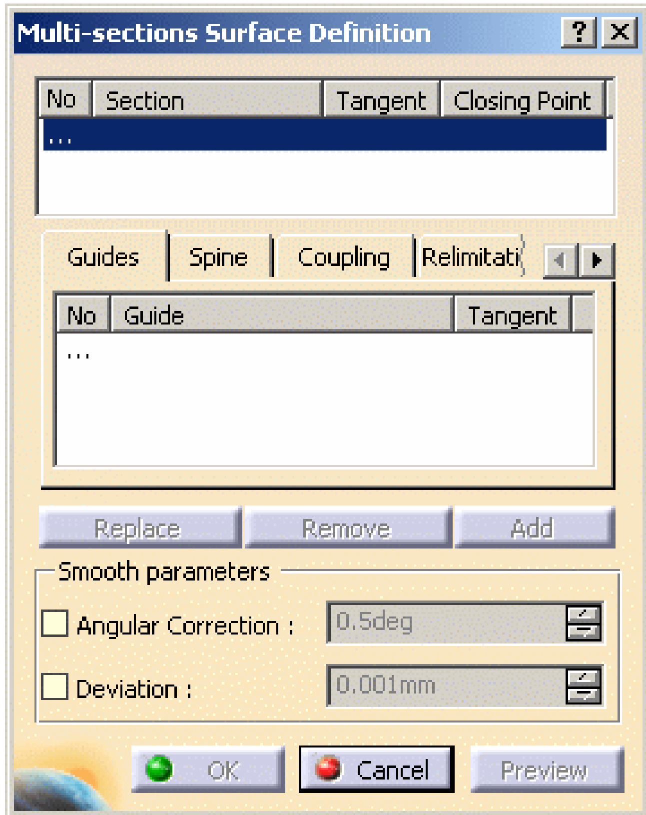
- Relimitation
- Planar Surface Detection
- Coupling

You can generate a multi-sections surface by sweeping two or more section curves along an automatically 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-sections Surface Definition dialog box appears.



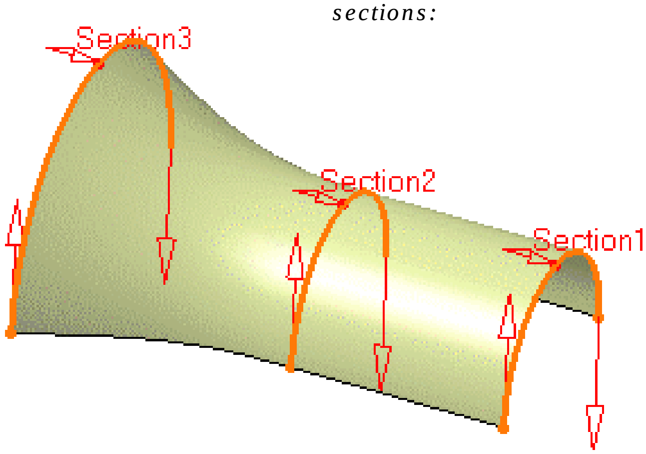
2. Select two or more planar section curves.

The curves must be continuous in point.

You can select tangent surfaces for the start and end section curves. These tangent surfaces must not be parallel to the sections.

A closing point can be selected for a closed section curves.

Example of a multi-sections surface defined by three planar sections:



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

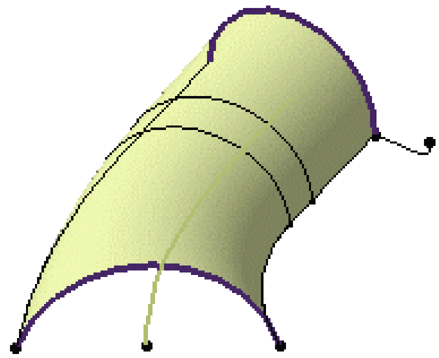
Guide curves must intersect each section

Example of a multi-sections surface defined by 2 planar sections and 2 guide curves:

curve and must be continuous in point.

The first guide curve will be a boundary of the multi-sections surface if it intersects the first extremity of each sections curve.

Similarly, the last guide curve will be a boundary of the multi-sections surface if it intersects the last extremity of each section curve.



You can make a multi-sections surface tangent to an adjacent surface by selecting an end section that lies on the adjacent surface. In this case, the guides must also be tangent to the surface.

In Figure 2 a multi-sections surface tangent to the existing surface has been created:

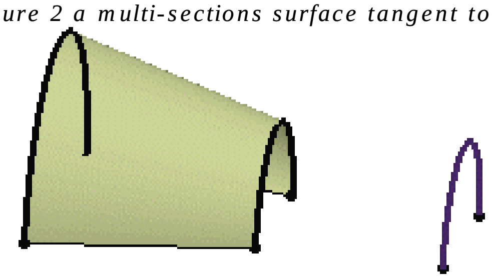


Figure 1

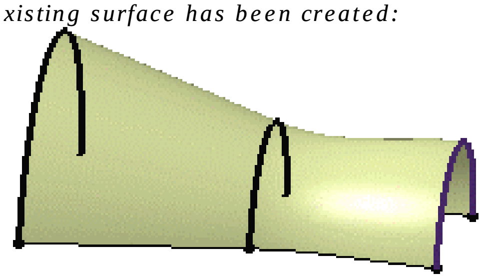


Figure 2

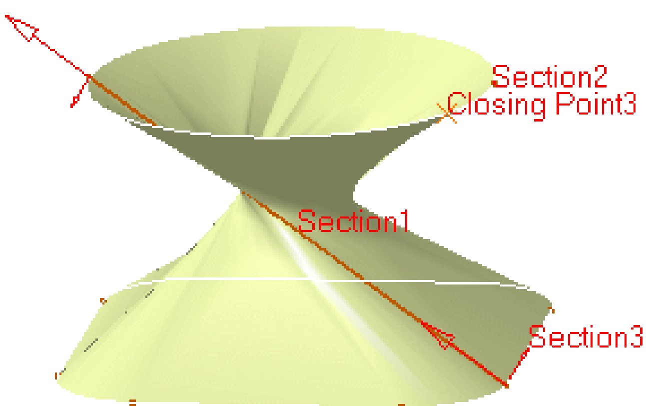
You can also impose tangency conditions by specifying a direction for the tangent vector (selecting a plane to take its normal, for example). This is useful for creating parts that are symmetrical with respect to a plane. Tangency conditions can be imposed on the two symmetrical halves.
Similarly, you 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.

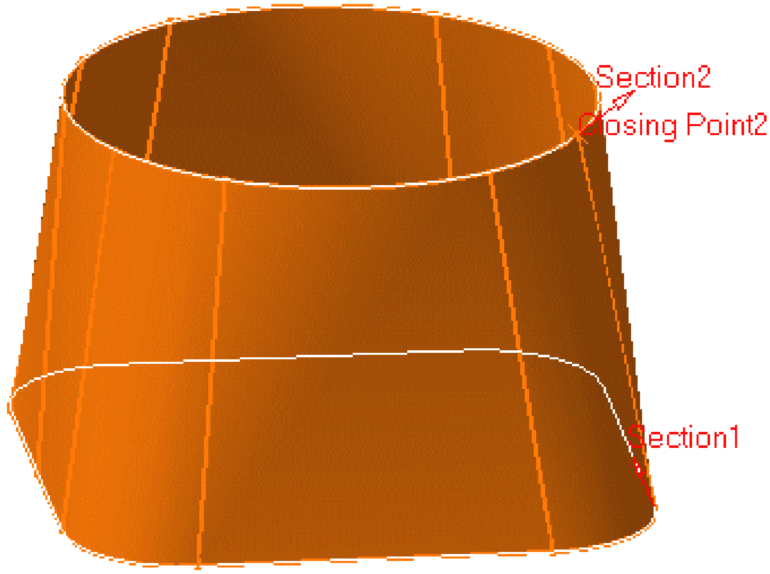
4. In the **Spine** tab page, select the **Spine** check box to use a spine that is automatically computed by the program or select a curve to impose that curve as the spine.

Note that the spine curve must be normal to each section plane and must be continuous in tangency.

You can create multi-sections surface 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.

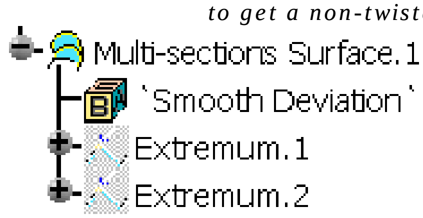




The surface is twisted

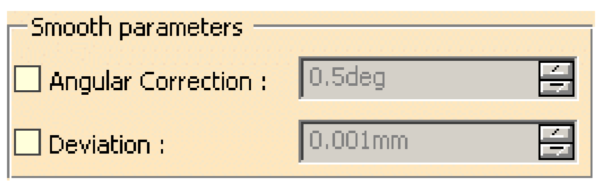
A new closing point has been imposed to get a non-twisted surface

Extremum points are now aggregated under the parent command that created them and put in no show in the specification tree.



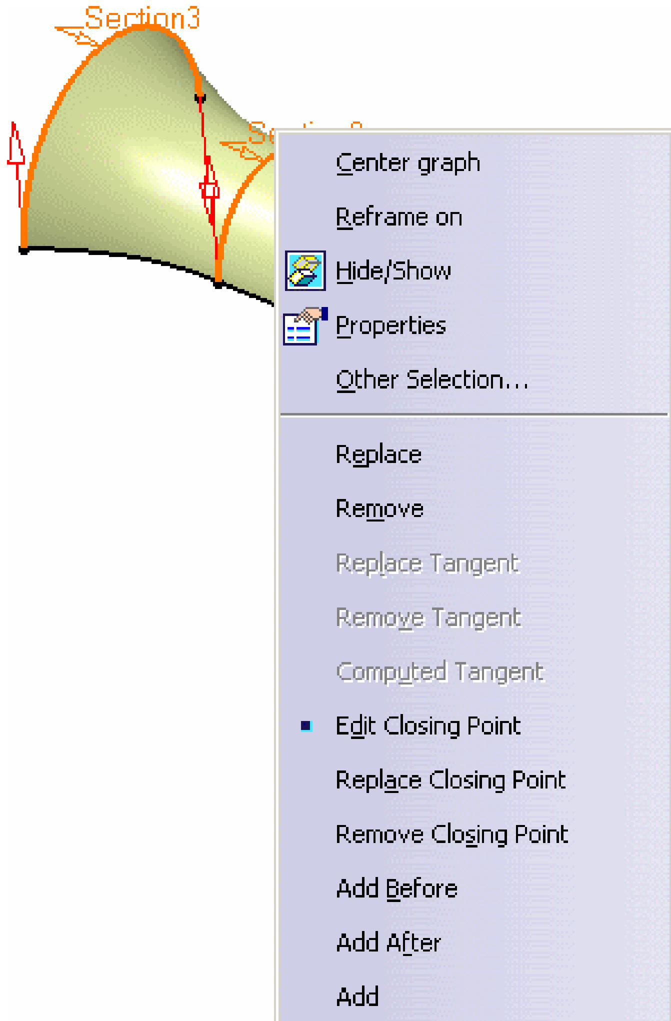
In the Smooth parameters section, you can check:

- 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-sections surface.
- the **Deviation** option to smooth the lofting motion by deviating from the guide curve(s).

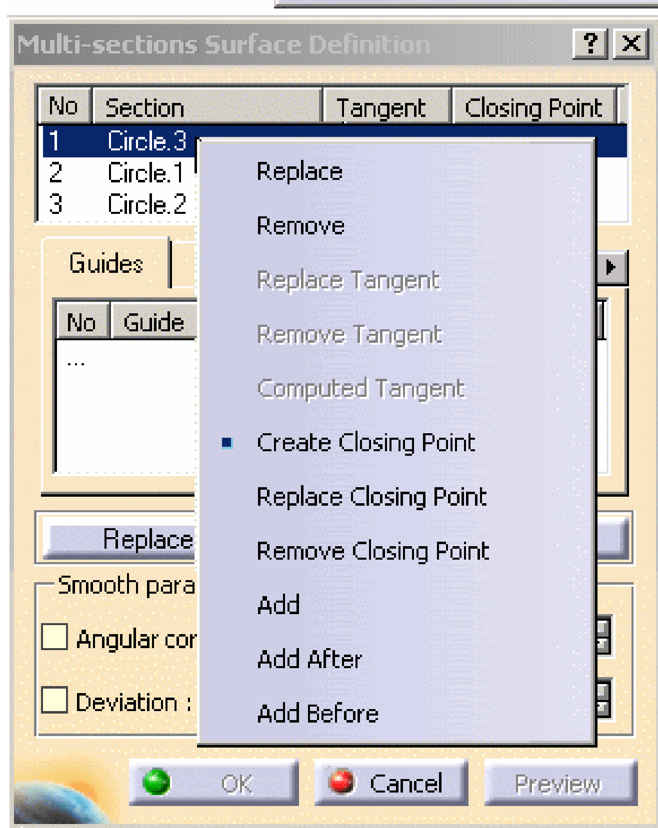


5. It is possible to edit the multi-sections surface reference elements by first selecting a curve in the dialog box list, or by selecting the text on the figure, then choosing a button to either:

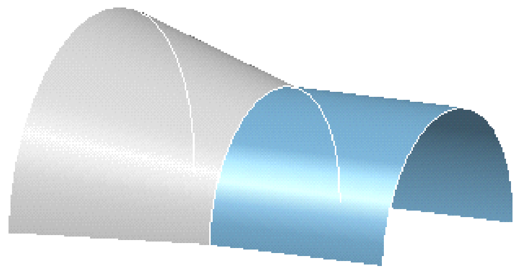
- remove the selected curve
- replace the selected curve by another curve
- add another curve



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.



The following example illustrates the result when the tangency condition is removed between the blue multi-sections surface and the adjacent surface.



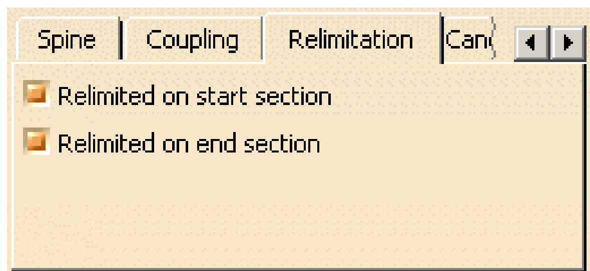
6. Click **OK** to create the multi-sections surface.

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

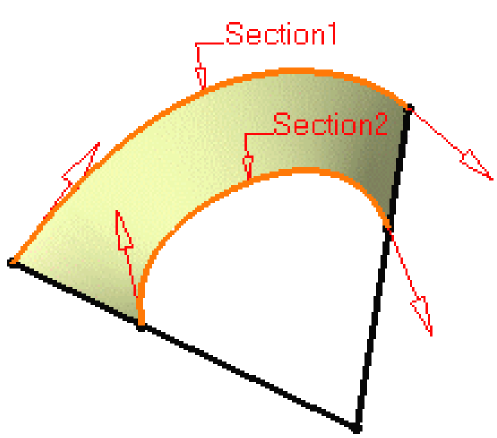
- Sections can be 3D curves with following restrictions:
  - the intersection between one 3D profile and all guides must be coplanar (if three guides or more are defined)
  - in case of a user-defined spine, this spine must be normal to the plane implicitly obtained above.

Relimitation

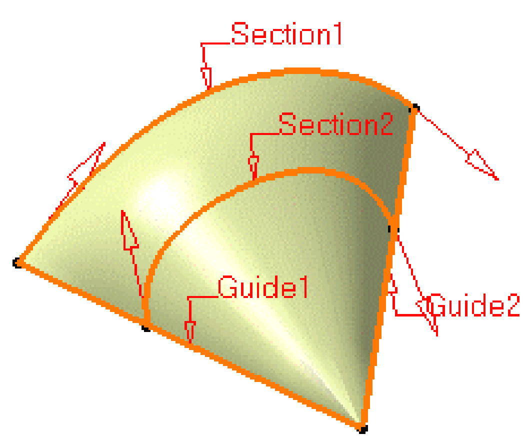
The Relimitation tab lets you specify the relimitation type. (Open the Loft3.CATPart document).
You can choose to limit the multi-sections surface only on the Start section, only on the End section, on both, or on none.



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



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

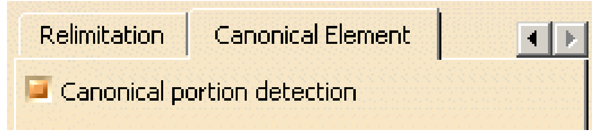


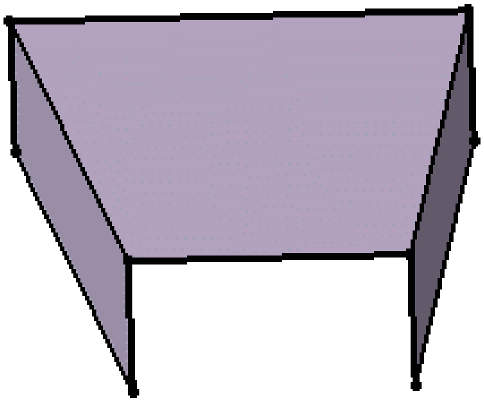
Multi-sections surface relimitation option unchecked on End section only

After the multi-sections surface is relimited, the following constraint needs to be fulfilled: the plane normal to the spine defined at the relimitation point must intersect the guide(s) and the point(s) resulting from this intersection must belong to the section.

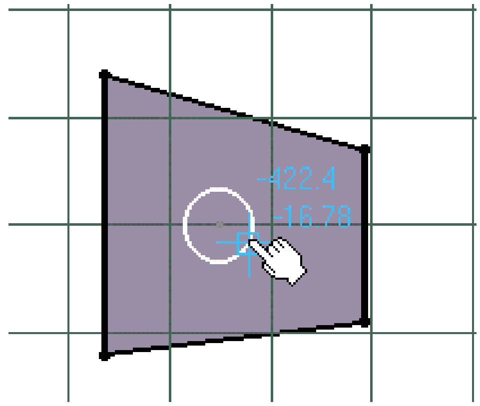
Planar Surface Detection

- Use the **Canonical portion detection** check button in the Canonical Element tab to automatically detect planar surfaces to be used as planes for features needing one in their definition.

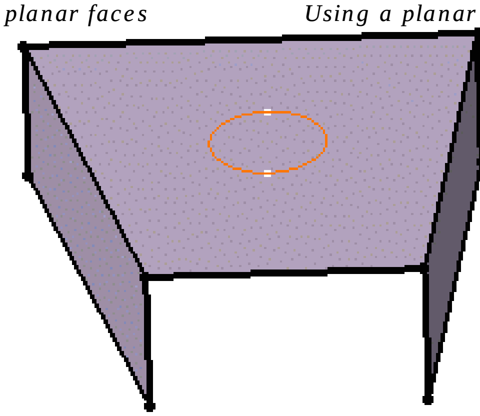




Initial multi-sections surface with planar faces



Using a planar face as reference for a sketch



Resulting sketch

Coupling

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

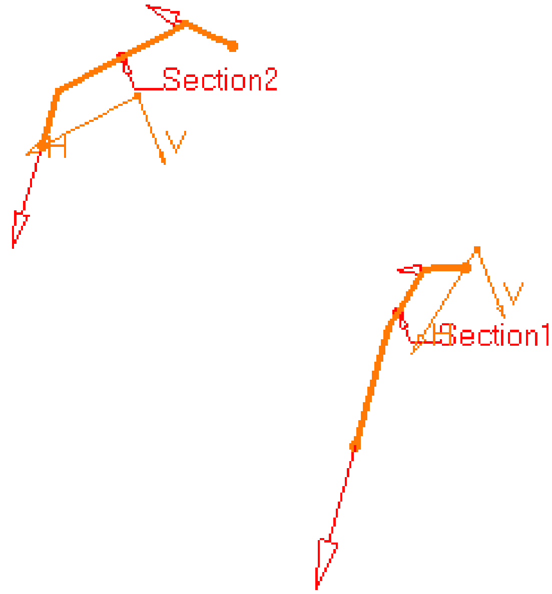
- coupling between two consecutive sections
- coupling between guides

These couplings compute the distribution of isoparameters on the surface.

Open the Loft2.CATPart document.

Coupling between two consecutive sections

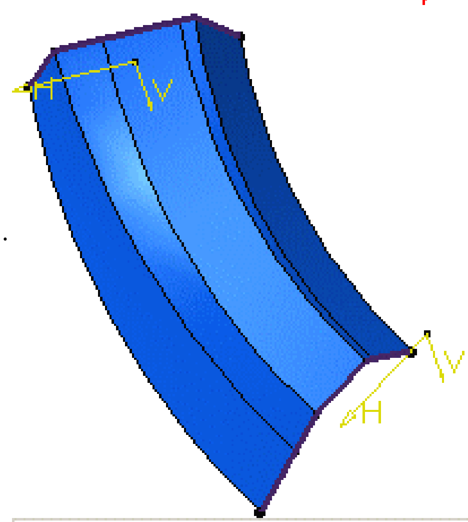
This coupling is based on the curvilinear abscissa.



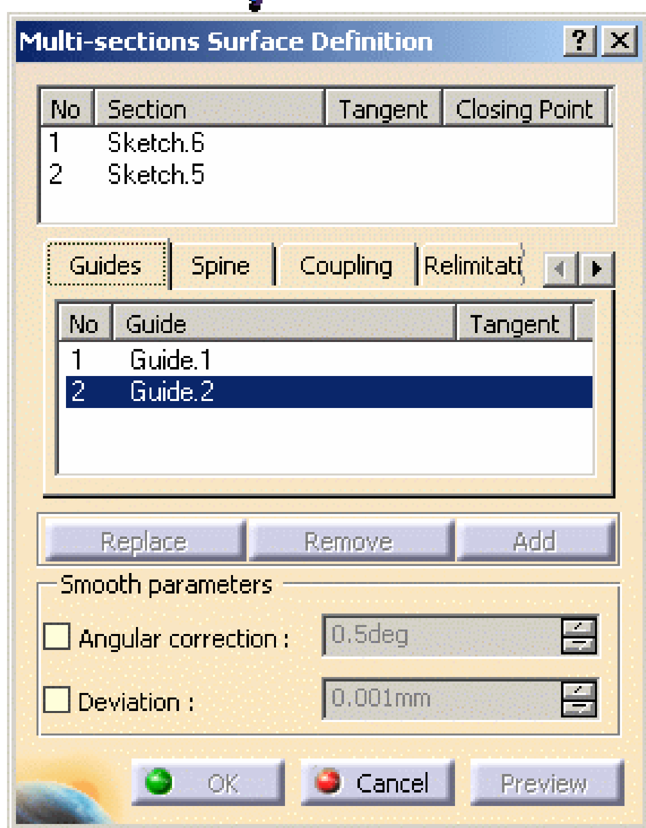
1. Click the **Multi-sections Surface** icon .

The Multi-sections Surface Definition dialog box appears.

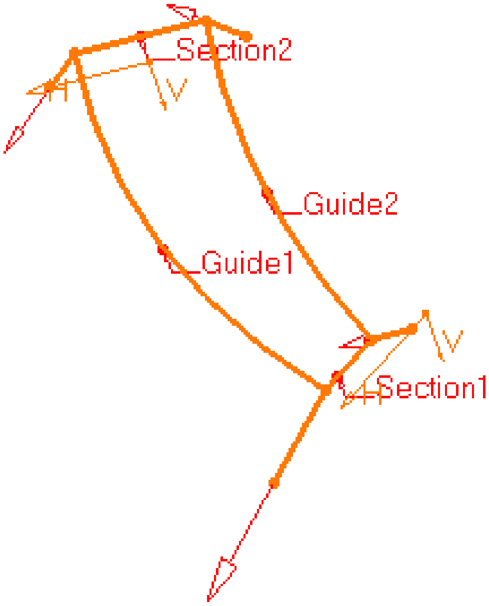
2. Select the two consecutive sections.

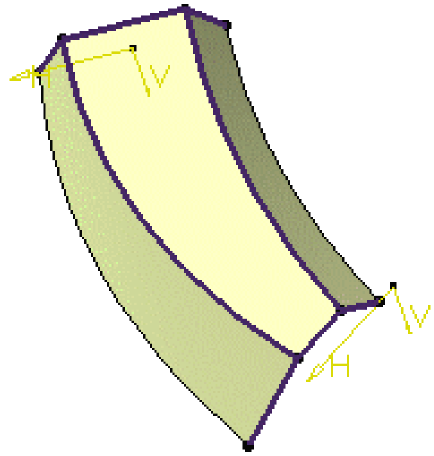


3. Click **OK** to create the multi-sections surface.



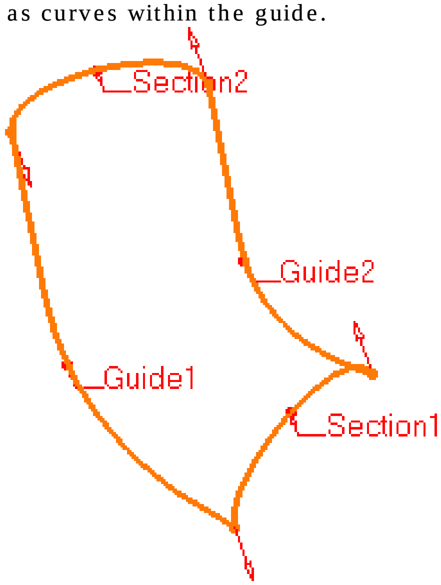
If you want to create a coupling between particular points, you can add guides or define the coupling type.

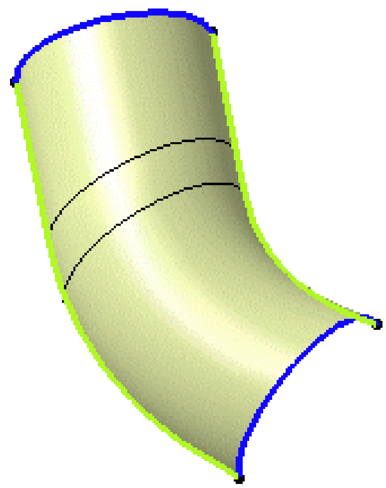




Coupling between guides

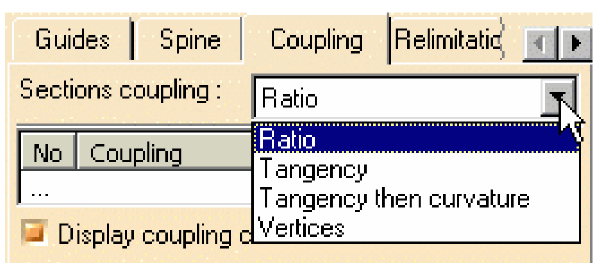
This coupling is performed by the spine.
If a guide is the concatenation of several curves, the resulting multi-sections surface will contain as many surfaces as curves within the guide.

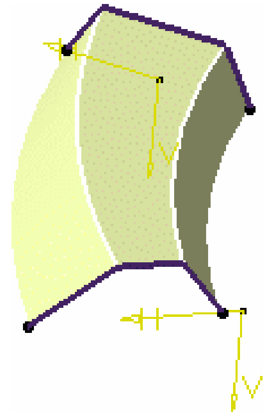




Several coupling types are available, depending on the section configuration:

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



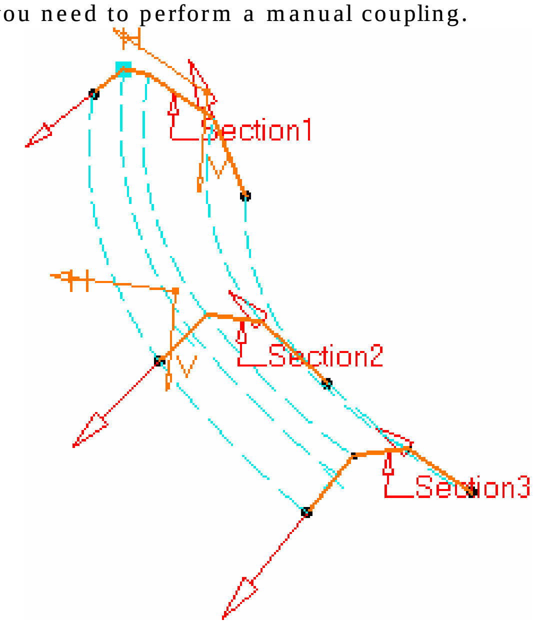


- **Tangency**: the curves are coupled according to their tangency discontinuity points. If they do not have the same number of points, they cannot be coupled using this option.
- **Tangency then curvature**: the curves are coupled according to their tangency continuity first then curvature discontinuity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

Manual Coupling

If the number of vertices differ from one section to another, you need to perform a manual coupling.

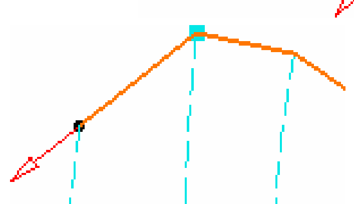
1. Select the sections for the multi-sections surface, and check their orientations.
2. In the **Coupling** tab, choose the **Tangency** option and click **Apply**.



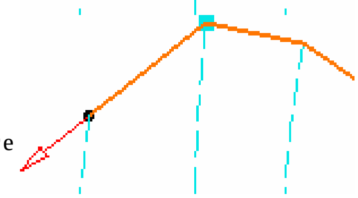
An error message is displayed as the number of discontinuity points on the first section is greater than on the other two sections.

The points that could not be coupled, are displayed in the geometry with specific symbol depending on the selected mode, along with coupling lines:

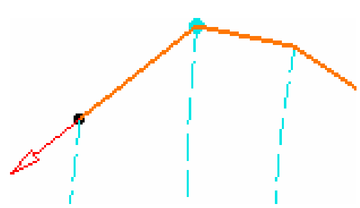
- In **Tangency** mode: uncoupled tangency discontinuity points are represented by a square



- In **Tangency then curvature** mode:
  - uncoupled tangency discontinuity points are represented by a square
  - uncoupled curvatures discontinuity points are represented by a empty circle

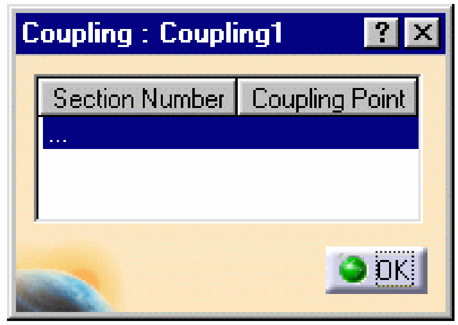


- In **Vertices** mode: uncoupled vertices are represented by a full circle



3. Click in the coupling list, or choose **Add coupling** in the contextual menu, or using the **Add** button, and manually select a point on the first section.

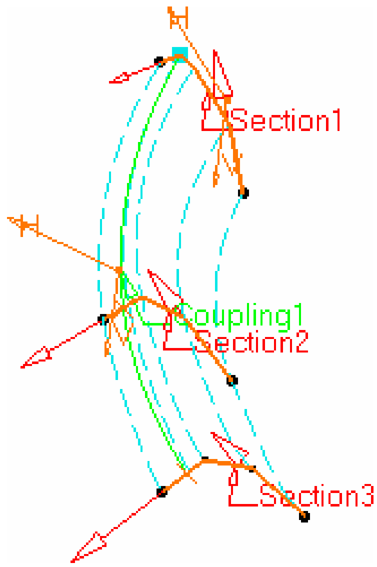
The Coupling dialog box is displayed.



4. Select a corresponding coupling point on each section of the multi-sections surface.

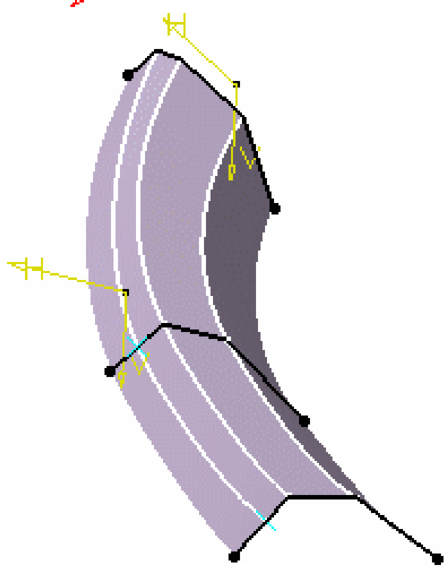
The Coupling dialog box is updated consequently, and the coupling curve is previewed, provided the **Display coupling curves** option is active.

When a coupling point has been defined on each section, this dialog box automatically disappears.



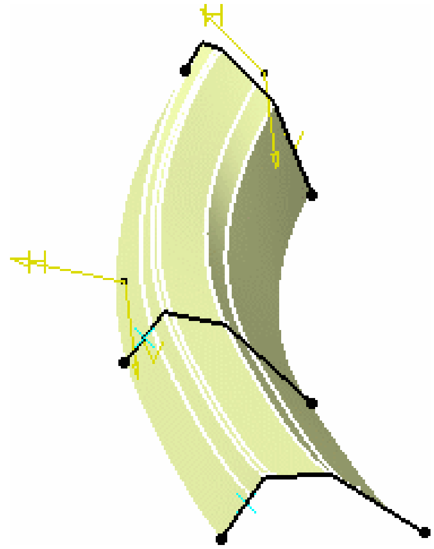
5. Click OK.

The multi-sections surface is created as defined with the coupling specifications.

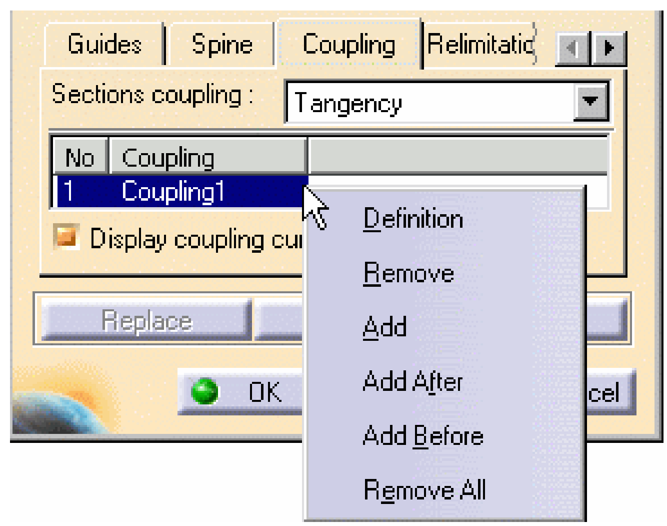


The same multi-sections surface without coupling and with Ratio option would have looked like this:

Note the increased number of generated surfaces.



- You can create coupling point on the fly, using the **Create coupling point** contextual menu item (click on the document background to display the contextual menu) instead of selecting an existing point.
- To edit the coupling, simply double-click the coupling name in the list (Coupling tab) to display the Coupling dialog box. Then you select the point to be edited from the list and create/select a replacing coupling point, then click OK.
- Use the contextual menu on the coupling list to edit defined couplings.



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.

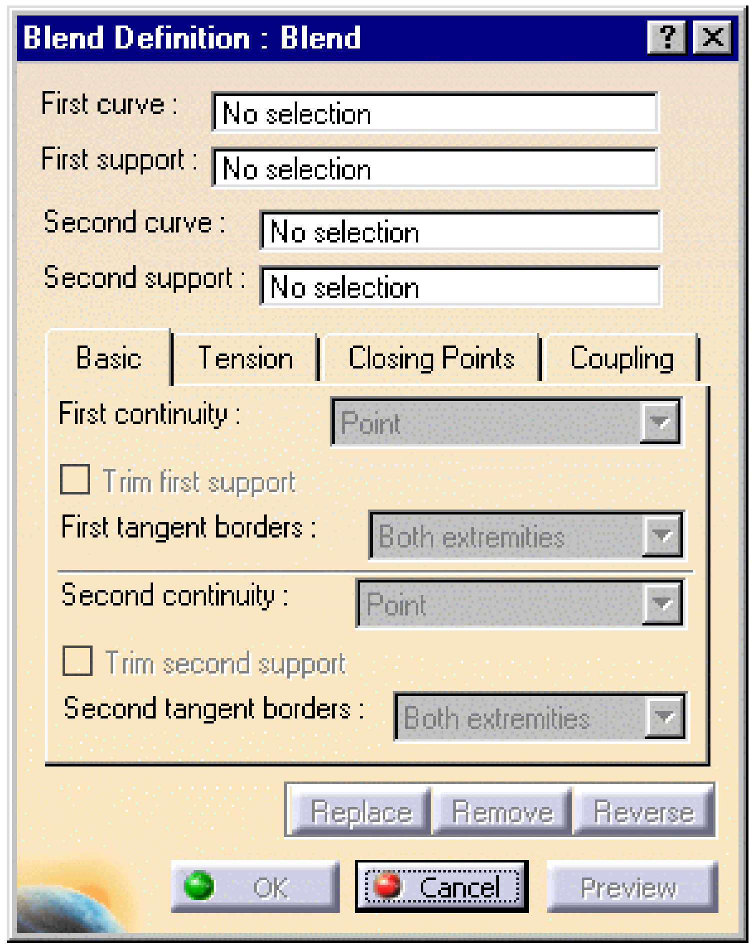
Several cases are worth surveying:

- blend between curves
- blend between closed contours
- coupling blend

Open the Blend1.CATPart document.

1. Click the **Blend** icon .

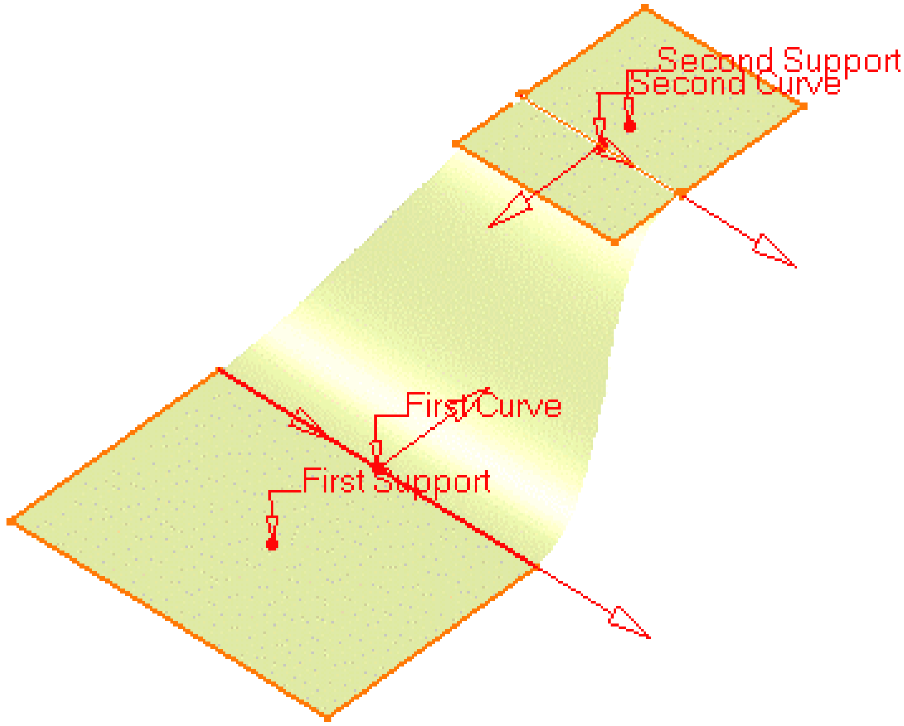
The Blend Surface Definition dialog box appears.



Blend between curves:

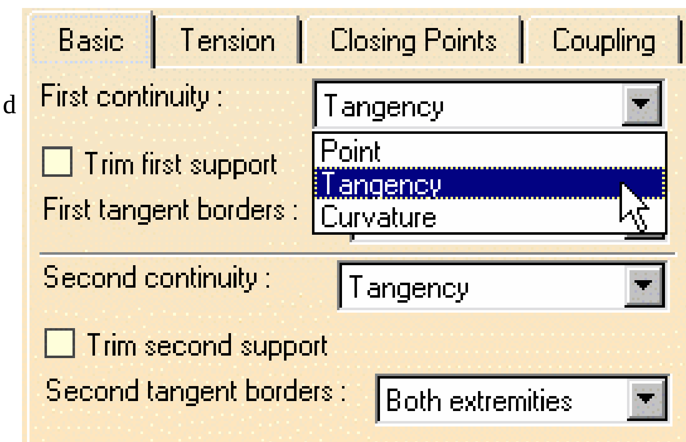
1. Successively select the first curve and its support, then the second curve and its support.

   These can be surface edges, or any curve.

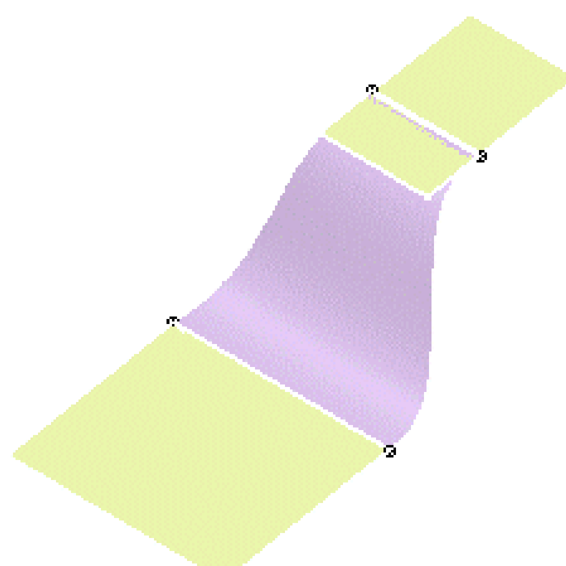


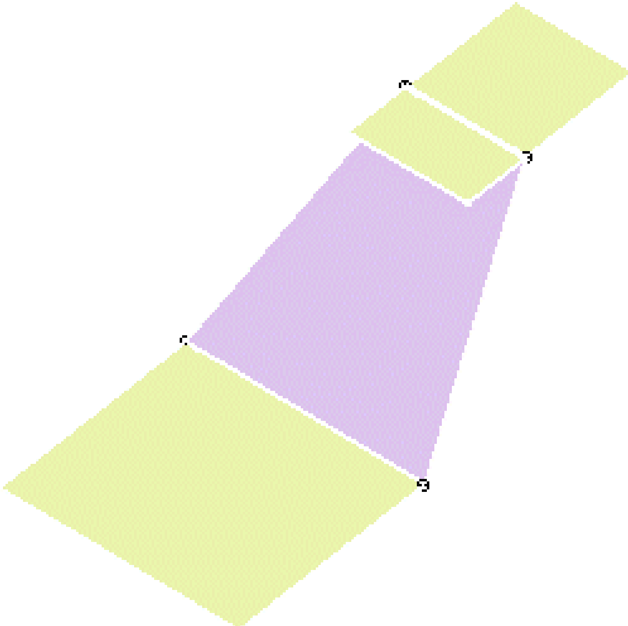
2. Set the continuity type using the **Basic** tab.

   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:

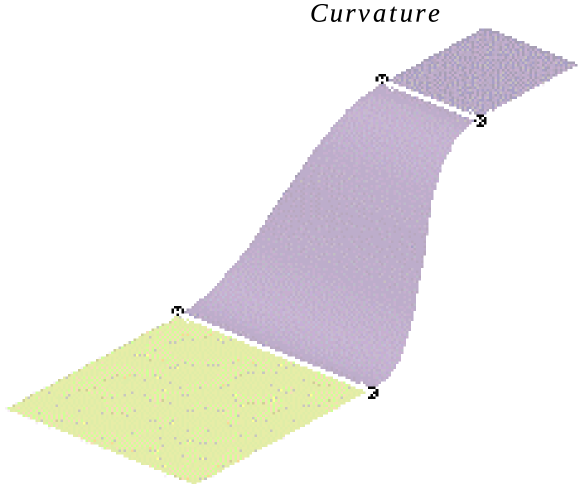




Point continuity on both limits

Curvature

3. 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.

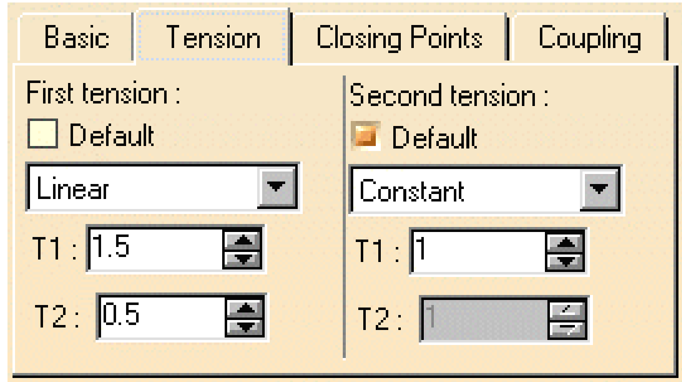
You can also specify whether and where the blend boundaries must be tangent to the supports boundaries:

- **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.

4. 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.

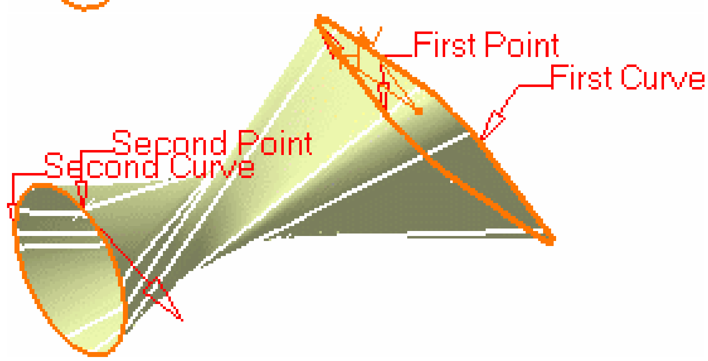
   A third tension type is available: **S Type**. It enables to set a variable tension.

   If you choose any of the tension types in the drop-down list, the Default button is deselected.

5. Click OK.

Blend between closed contours:

1. Successively select two contours.



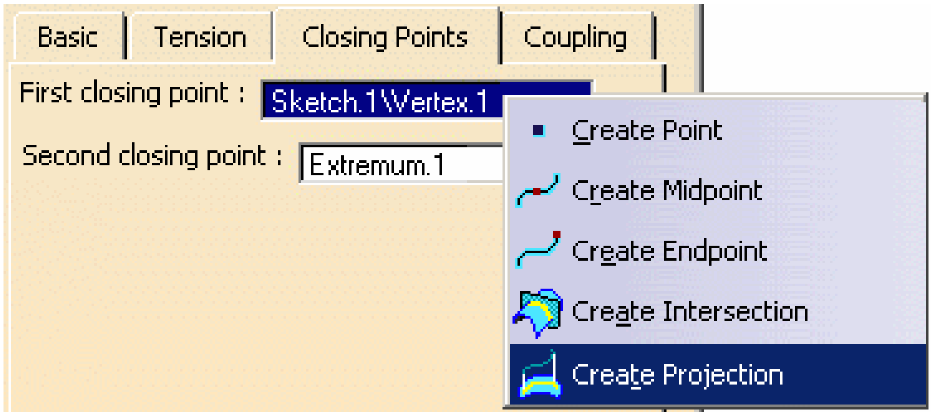
2. Click Preview.

   The surface to be generated is twisted.

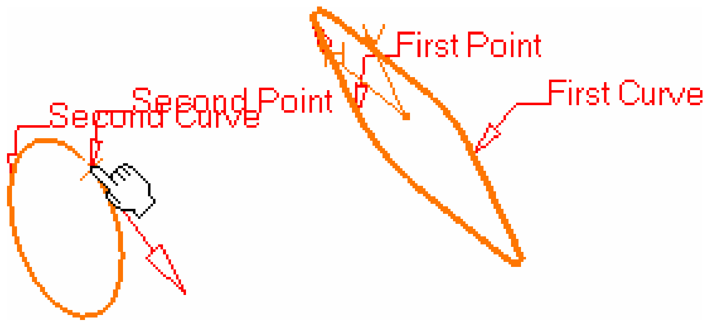
   To avoid this you need to define a closing point.

By default, the system detects and highlights a vertex on each curve that can be used as a closing point, or it creates an extremum point (you can also manually select another one if you wish).

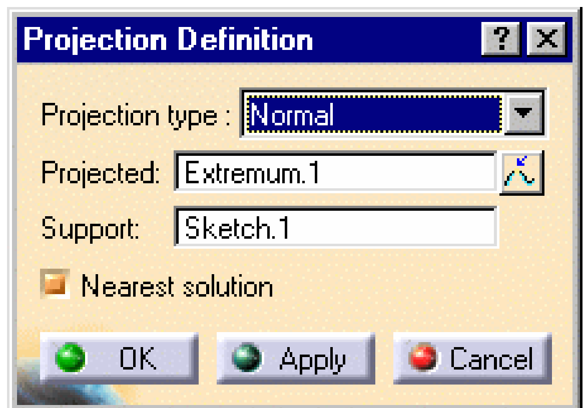
3. Choose the **Closing Point** tab, and using the contextual menu, choose **Create Projection**.



4. The Projection Definition dialog box is displayed.

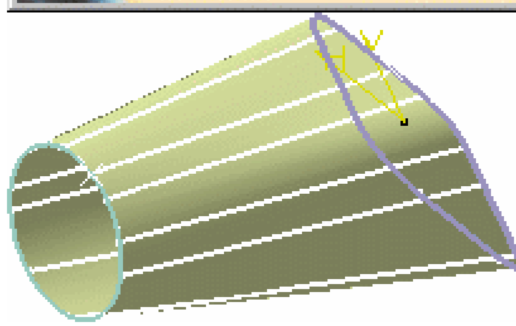


5. Select the closing point on the second contour, then the first curve onto which the point is to be projected.



6. Click OK in the Projection Definition dialog box.

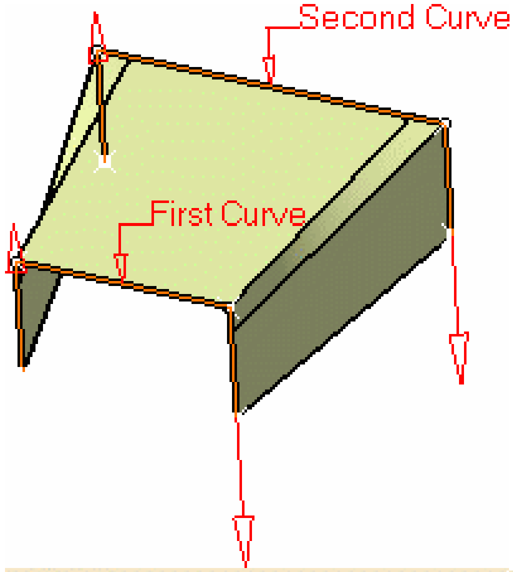
7. Click OK in the Blend Definition dialog box.



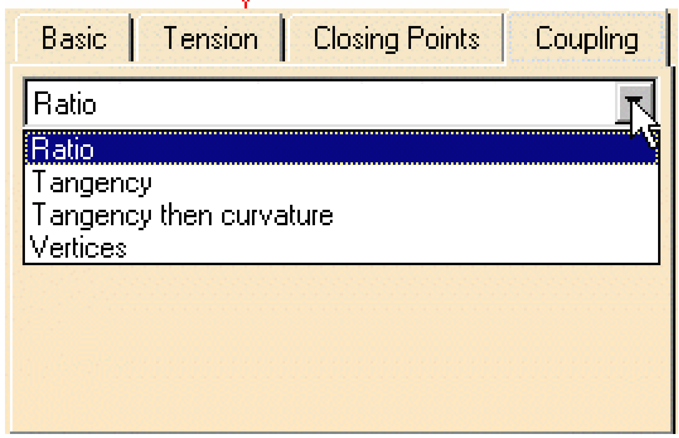
The blend is correctly created.

Coupling blend:

1. Select the elements to be blended and click Preview.



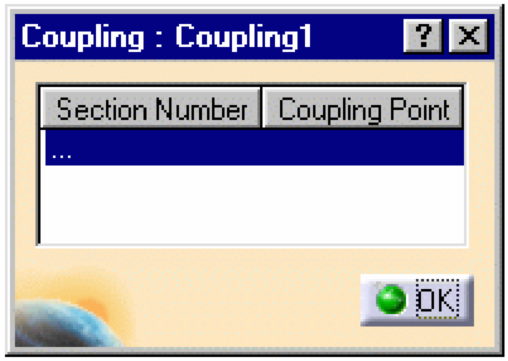
2. Select the **Coupling** tab and define the coupling type.



- **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.

3. Click in the coupling list, or choose **Add coupling** in the contextual menu, or using the **Add** button, and manually select a point on the first section.

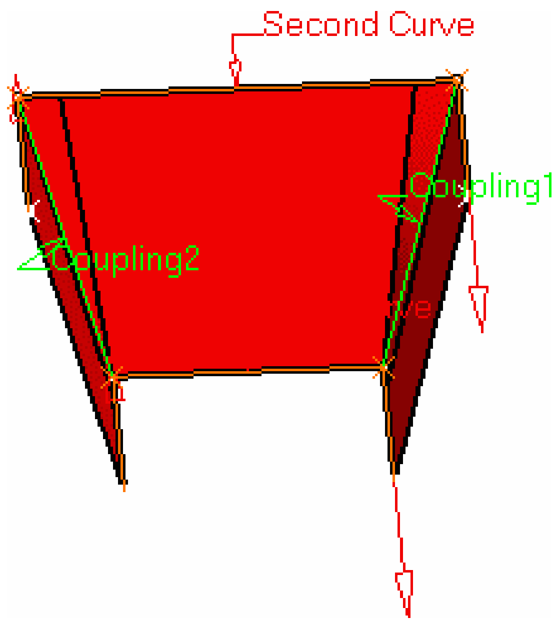
The Coupling dialog box is displayed.



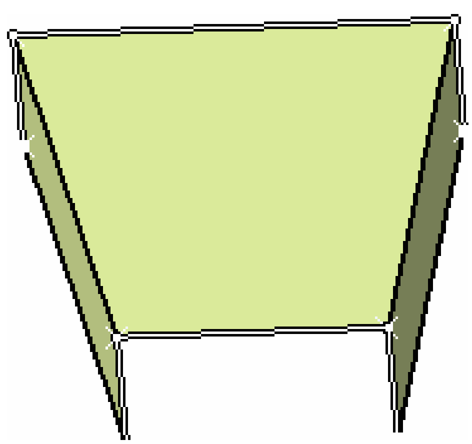
4. Select a corresponding coupling point on each section.

The Coupling dialog box is updated consequently, and the coupling curve is previewed, provided the **Display coupling curves** option is active.

When a coupling point has been defined on each section, this dialog box automatically disappears.



5. 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.

Performing Operations on Shape Geometry

Generative Shape Design allows you to modify your design using techniques such as trimming, extrapolating and filleting.

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.

Smooth a curve: select the curve to be smoothed and set the tangency threshold

Restore 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 edge or the face of a geometric element, and set the propagation type.

Extract multiple edges: select one or more element(s) of a sketch, and click OK.

Create bitangent shape fillets: select two support surfaces, and define required parameters.

Create tritangent shape fillets: select two support surfaces, select the surface to remove, and enter a radius value.

Create edge fillets: select an internal edge of a surface, the surface itself, define the type of fillet and propagation mode, and enter a radius value.

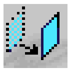
Create variable radius fillets: select an edge to be filleted, specify the fillet extremity type, the propagation mode, select a point on the edge where the radius will vary, and enter the radius value at this point.

Create variable radius fillets using a spine: select edges with no tangency continuity to be filleted, specify the fillet extremity type, the propagation mode, click the circle option, and select a spine.

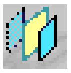
 Create face-face fillets: select a support surface, the two faces to be filleted, specify the relimitation mode, and enter a radius value.

 Create tritangent fillets: select a support surface, specify the relimitation mode, the two faces to be filleted and the one to be removed.

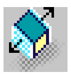
Reshape Corners: click either the **Edge Fillet** icon or the **Variable Radius Fillet** icon, select the edge to be filleted, click **More>>** and define the corner to reshape and the setback distance.

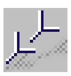
 Translate geometry: select an element, a translation direction (line, plane or vector), specify the translation distance.

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

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

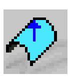
 Transform geometry by scaling: select an element, then a point, plane, or planar surface as reference element, and specify the scaling ratio.

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

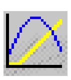
 Transform geometry from an axis to another: select an element to be transformed, specify the axis system characteristics, and the enter the affinity ratio values.

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

 Extrapolate a curve: 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).

 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.

 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.

 Create laws: select a reference line and a curve.

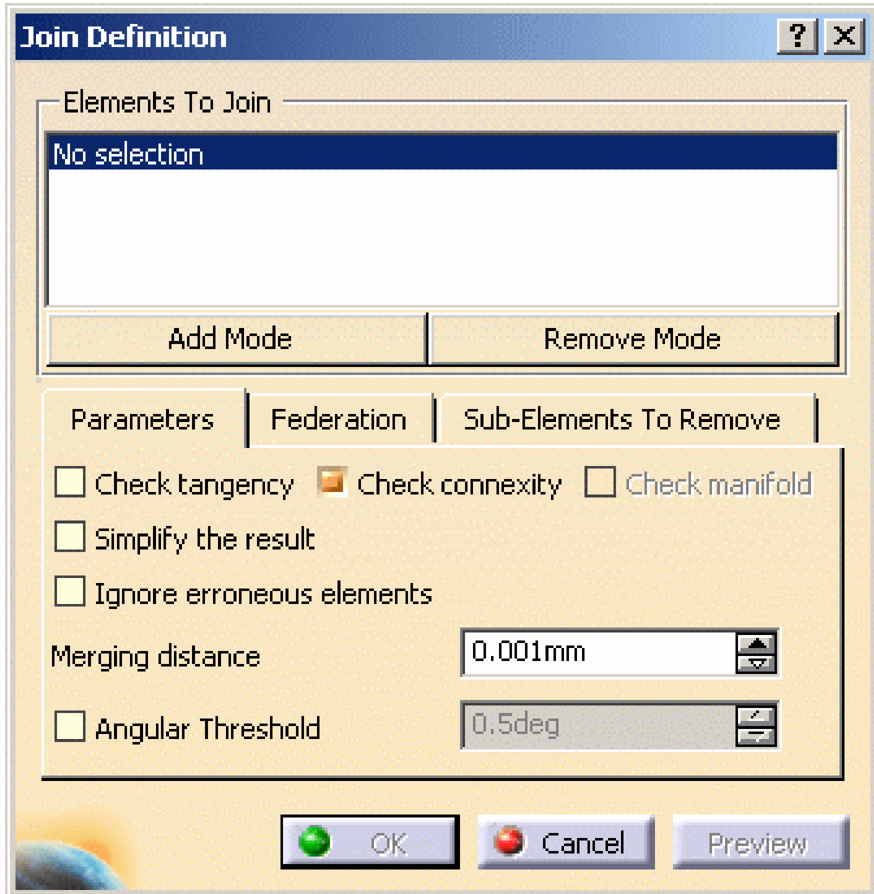
Joining Surfaces or Curves

This task shows how to join surfaces or curves.

Open the Join1.CATPart document.

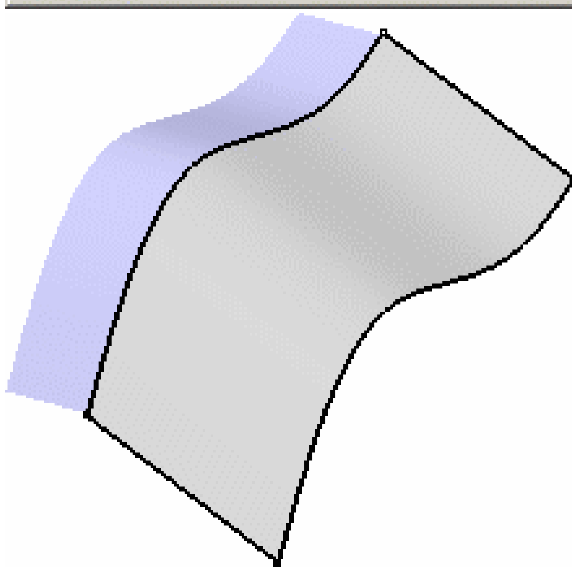
1. Click the **Join** icon.

   The Join Definition dialog box appears.



In Part Design workbench, the **Join** capability is available as a contextual command named '**Create Join**' that you can access from Sketch-based features dialog boxes.

2. Select the surfaces or curves to be joined.

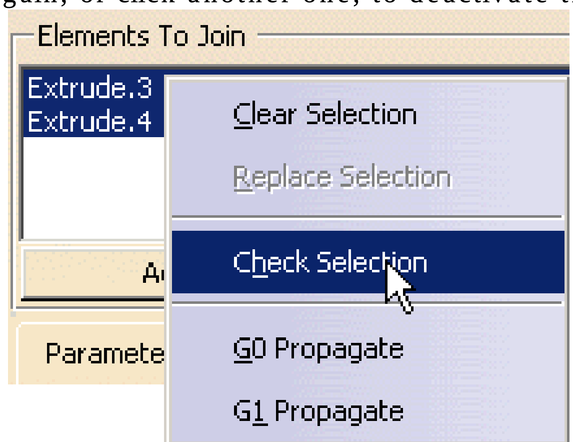


3. You can edit the list of elements to be joined:

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

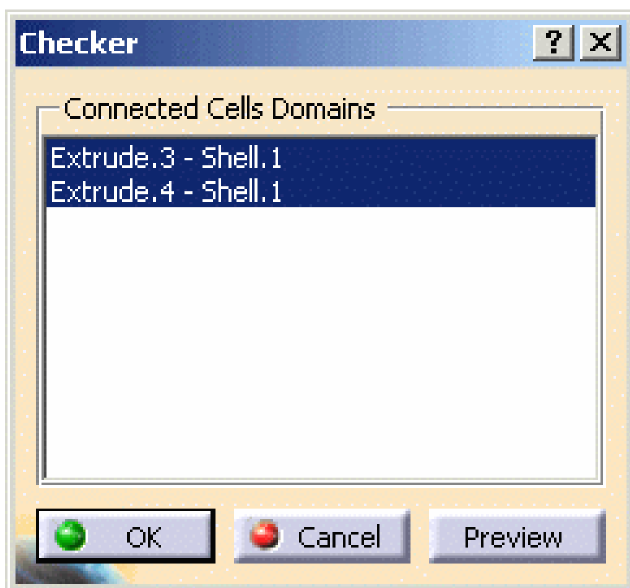
If you double-click the **Add Mode** or **Remove Mode** button, the chosen mode is permanent, i.e. successively selecting elements will add/remove them. However, if you click only once, only the next selected element is added or removed.
You only have to click the button again, or click another one,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

4. Right-click the elements from the list and choose the **Check Selection** 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:

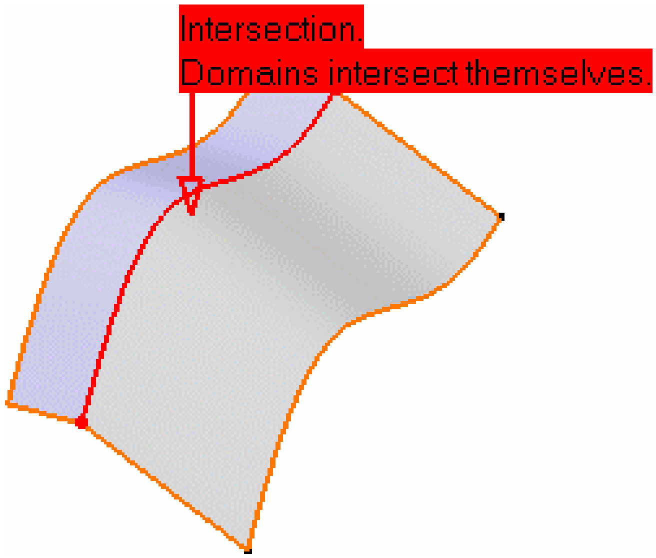
The Checker dialog box is displayed, containing the list of domains (i.e. sets of connected cells) belonging to the selected elements from the **Elements To Join** list.



5. Click Preview.

- 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 again and choose the Propagation options to allow the selection of elements of same dimension.

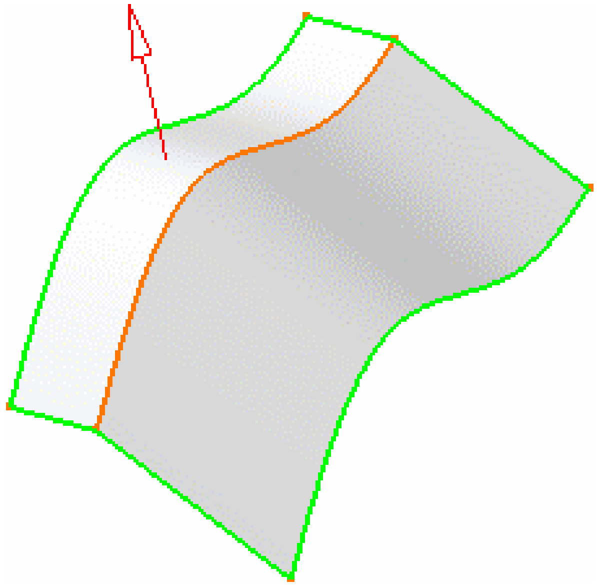
- **G0 Propagate**: the tolerance corresponds to the Merging distance value.
- **G1 Propagate**: the tolerance corresponds to the Angular Threshold value, if defined. Otherwise, it corresponds to the G1 tolerance value as defined in the part.

Each new element found by propagation of the selected element(s) is highlighted and added to the **Elements To Join** list.
Please note that:

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

8. Click Preview in the Join Definition dialog box.

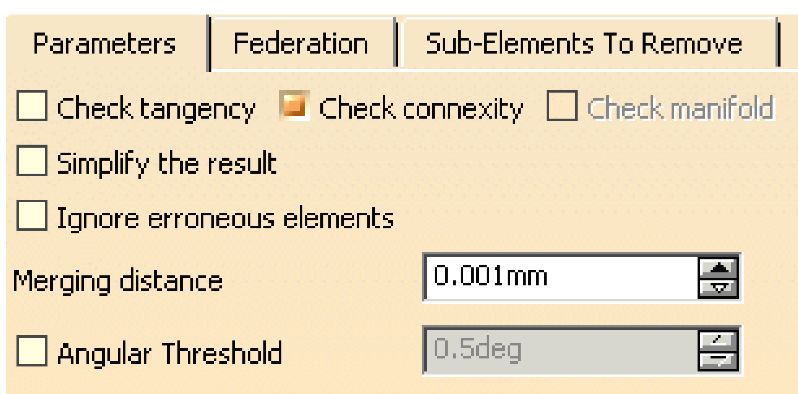
The joined element is previewed, and its orientation displayed. Click the arrow to invert it if needed.



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 Preview, 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 surfaces and edges that would not allow the join to be created.

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

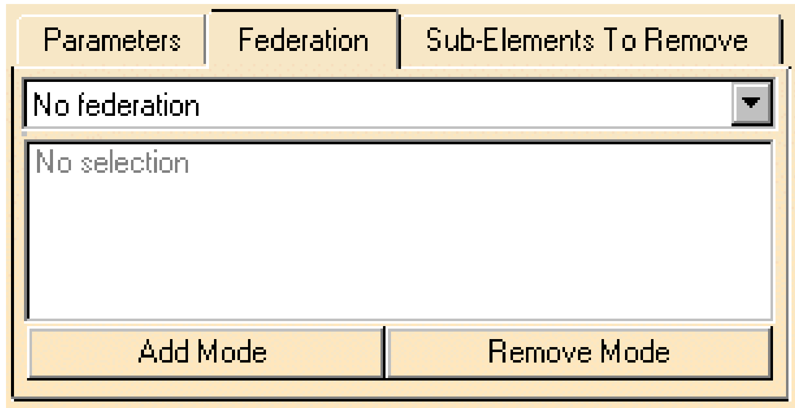
13. Check the **Angular Threshold** button to specify the angle value below which the elements are to be joined.
    If the angle value on the edge between two elements is greater than the **Angle Tolerance** value, the elements are not joined. This is particularly useful to avoid joining overlapping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

14. Click the **Federation** tab to generate groups of elements belonging to the join that will be detected together with the pointer when selecting one of them.

For further information, see Using the Federation Capability.

15. 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.

16. 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.

17. 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.

Using the Federation Capability

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

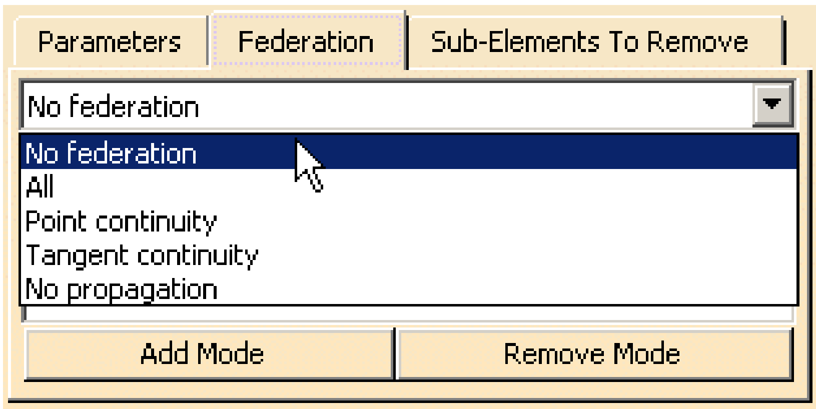
The purpose of the federation is to regroup several elements making up the joined surface or curve. This is especially useful when modifying linked geometry to avoid re-specifying all the input elements.

Open the Join2.CATPart document.

1. Create the join as usual, selecting all elements to be joined.

   (Make sure you do not select the Sketch.1).

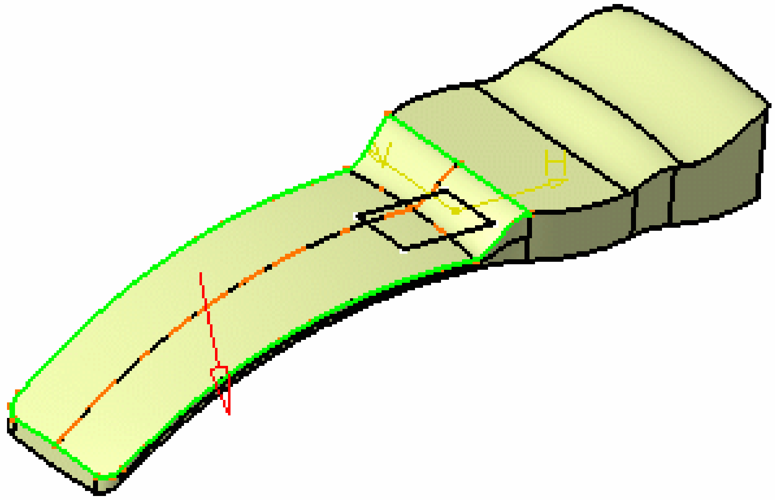
2. From the Join Definition dialog box click the **Federation** tab, then select one of the elements making up the elements federation.



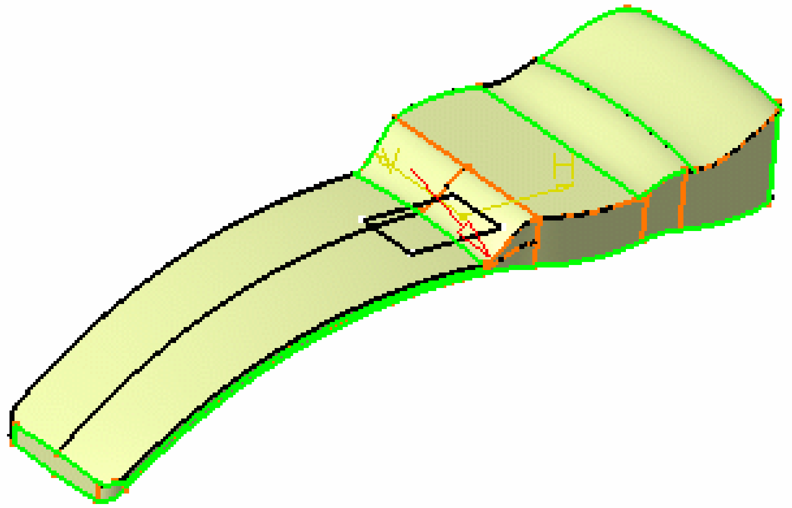
You can edit the list of elements taking part in the federation as described above for the list of elements to be joined.

3. Choose a propagation mode, the system automatically selects the elements making up the federation, taking this propagation mode into account.

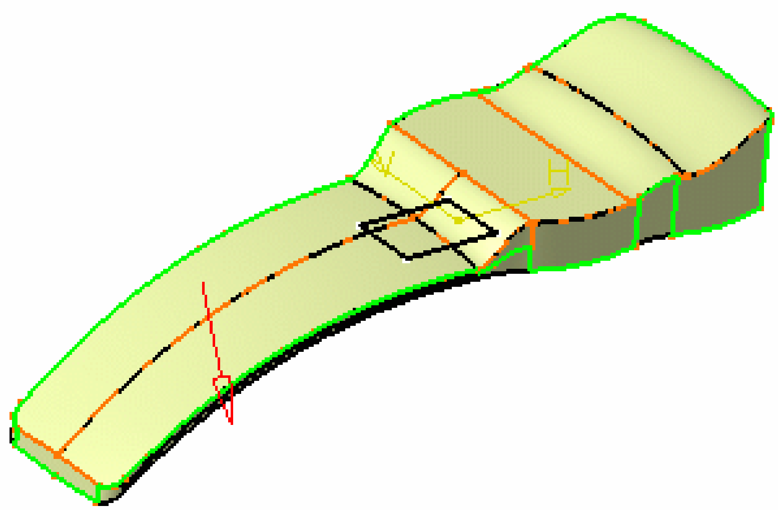
- **No federation**: only the elements explicitly selected are part of the federation



- **All**: all elements belonging to the resulting joined curve/surface are part of the federation

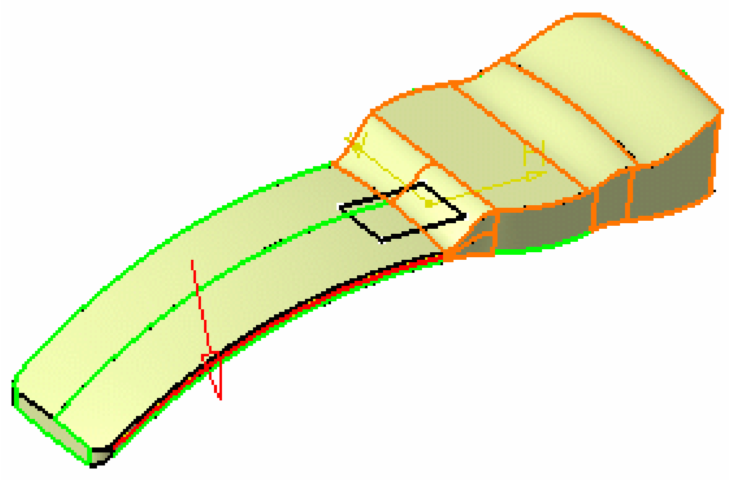


- **Point continuity**: all elements that present a point continuity with the selected elements and the continuous elements are selected; i.e. only those that are separated from any selected element is not included in the federation



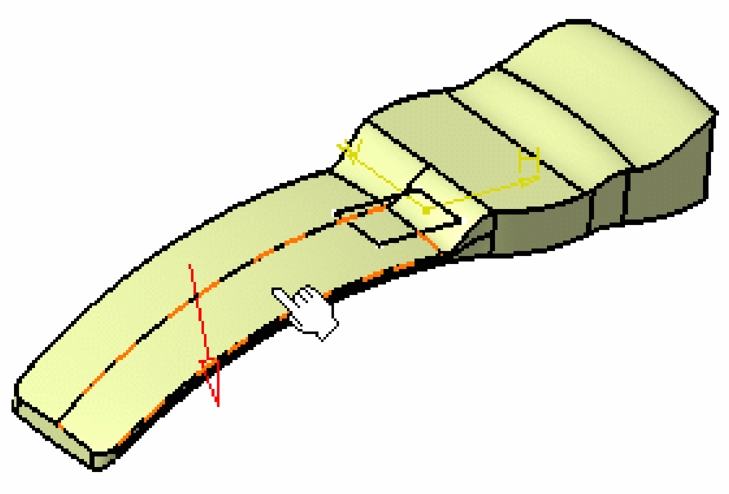
- **Tangent continuity**: all the elements that are tangent to the selected element, and the ones tangent to it, are part of the federation

  Here, only the top faces of the joined surface are detected, not the lateral faces.



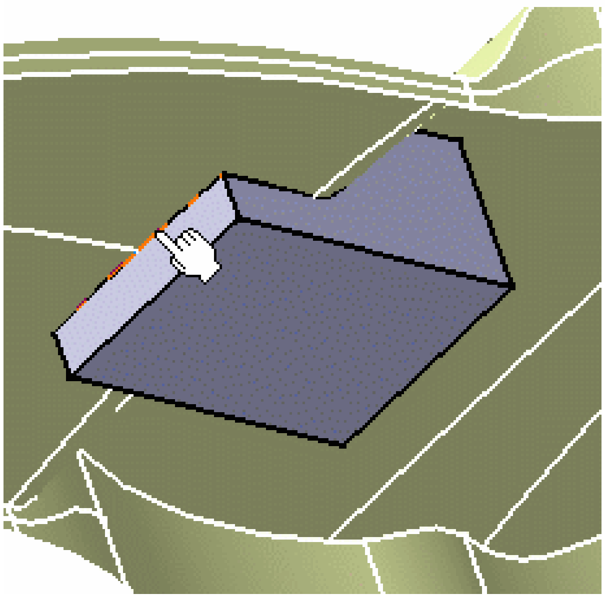
To federate a surface and its boundaries in tangency, you need to select the face as well as the edges: both face and edges will be federated.

- **No propagation**: only the elements explicitly selected are part of the propagation



4. Choose the **Tangency Propagation** federation mode as shown above.

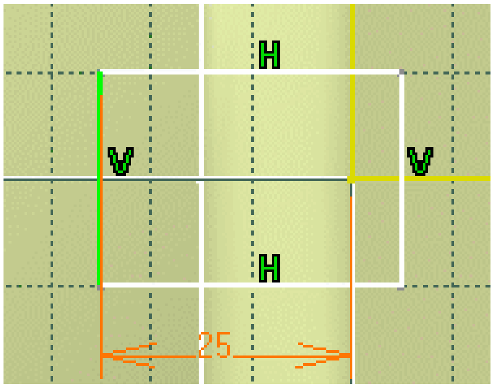
5. Move to the Part Design workbench, select the Sketch.1, and click the Pad icon to create an **up to surface** pad, using the joined surface as the limiting surface.



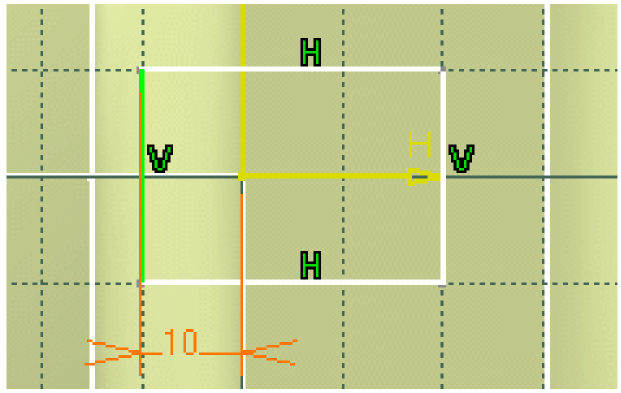
6. Select the front edge of the pad, and create a 2mm fillet using the Edge Fillet

 icon.

7. Double-click the Sketch.1 from the specification tree, then double-click the constraint on the sketch to change it to 10mm from the Constraint Definition dialog box.



Sketch prior to modification lying over two faces



Sketch after modification lying over one face only

8. Exit the sketcher .

The up to surface pas is automatically recomputed even though it does not lie over the same faces of the surface as before, because these two faces belong to the same federation. This would not be the case if the federation including all top faces would not have been created, as shown below.

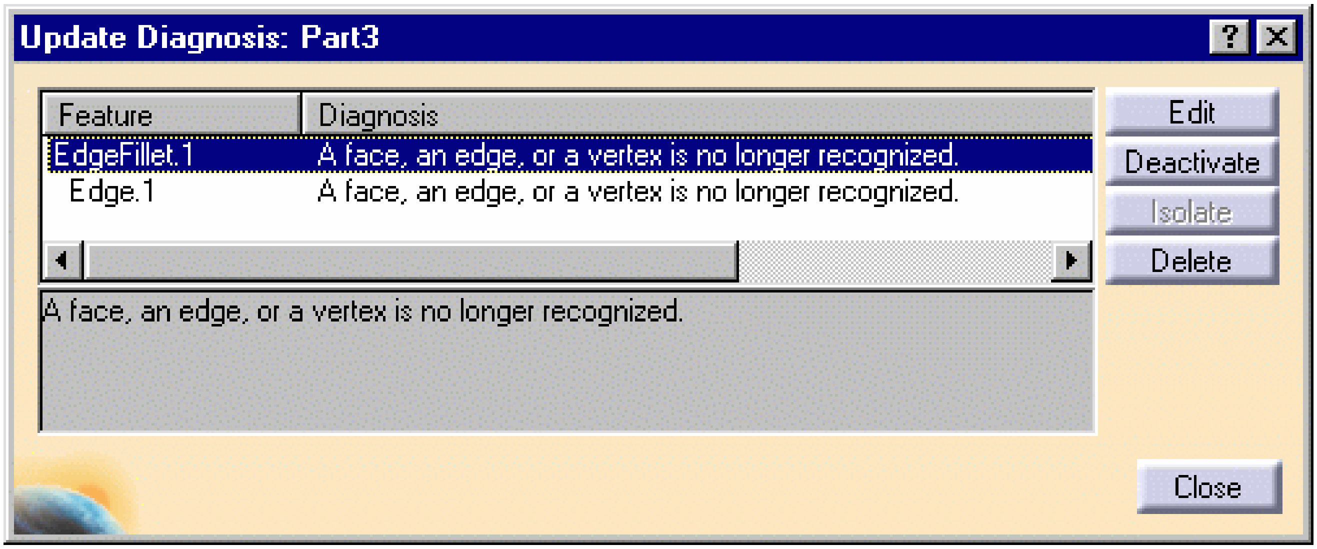
9. Double-click the joined surface (Join.1) to edit it, and choose the **No propagation** federation mode.

10. Click OK in the Join Definition dialog box.

A warning message is issued, informing you that an edge no longer is recognized on the pad.

11. Click OK.

The Update Diagnosis dialog box is displayed, allowing you to re-enter the specifications for the edge, and its fillet.



You then need to edit the edge and re-do the fillet to obtain the previous pad up to the joined surface.

12. Select the Edge.1 line, click the Edit button, and re-select the pad's edge in the geometry.

13. Click OK in the Edit dialog box.

    The fillet is recomputed based on the correct edge.

Healing Geometry

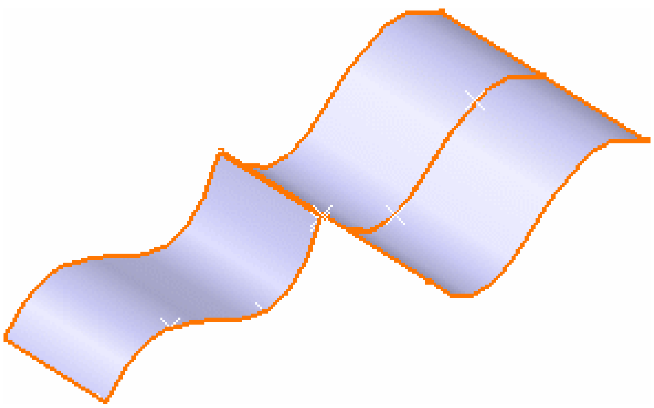
This task shows how to heal surfaces, that is how to fill any gap that may be appearing between two surfaces.
This command can be used after having checked the connections between elements for example, or to fill slight gaps between joined surfaces.

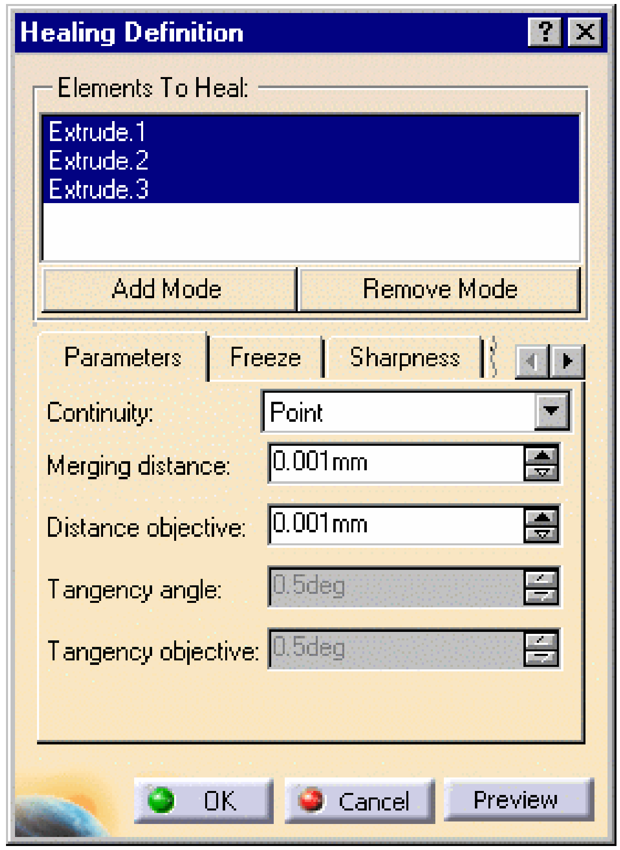
Open the Healing1.CATPart document from the Join Healing toolbar.

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 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.

Parameters tab

4. Define the distance below which elements are to be healed, that is deformed so that there is no more gap, using the **Merging distance.**

   Elements between which the gap is larger than the indicated value are not processed.

   In our example, we increase it to 1mm.

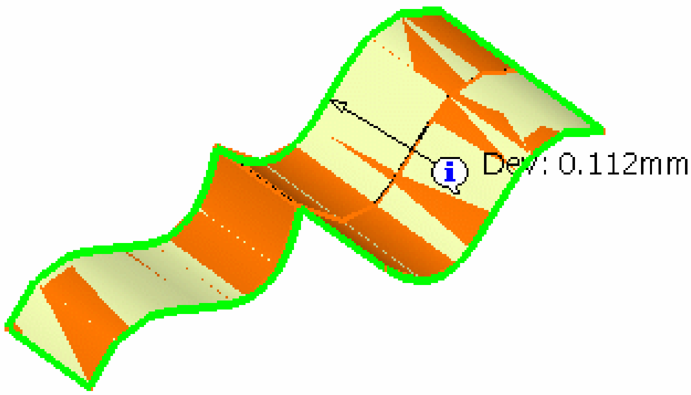
You can also set the **Distance objective**, i.e. the maximum gap allowed between two healed elements. By default it is set to 0.001 mm, and can be increased to 0.1 mm.

5. Change the continuity type to **Tangent**.

In that case, the **Tangency angle** field becomes active, allowing you to key in the angle below which the tangency deviation should be corrected.

The **Tangency objective** is, similarly to the **Distance objective**, the maximum allowed tangency deviation allowed between healed elements. The default value is 0.5 degree, but can range anywhere between 0.1 degree to 2 degrees.

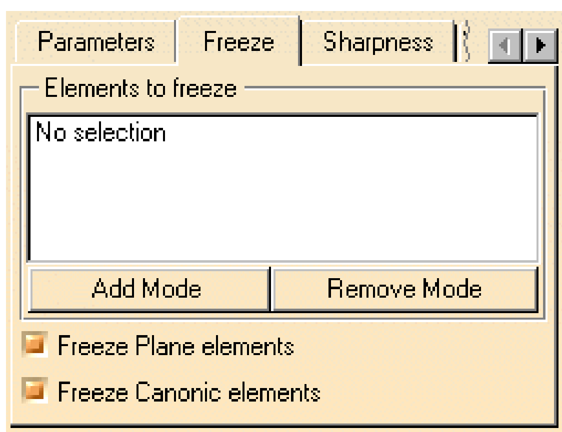
6. Click **Preview** to visualize the maximum deviation value between the input surfaces and the result in the 3D geometry.



The value is displayed on the edge or the face onto which the deviation is maximal, not exactly where the maximum deviation is located.

Freeze tab

6. Click the **Freeze** tab.



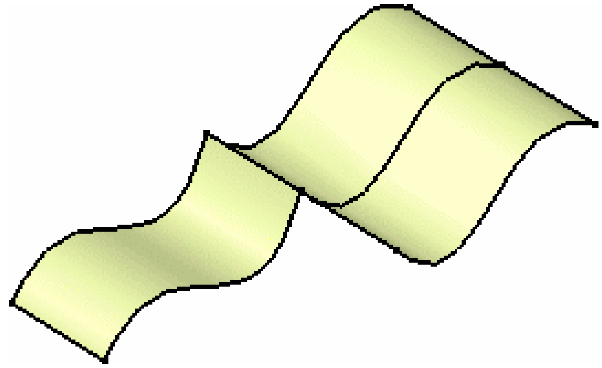
You can then define the list of frozen elements, that is the elements that should not be affected by the healing operation.

You can edit the list as described above for the list of elements to be healed.

Similarly to the **Elements to freeze** list, when the Freeze Plane elements or Freeze Canonic elements options are checked, no selected plane/canonic element is affected by the healing operation.

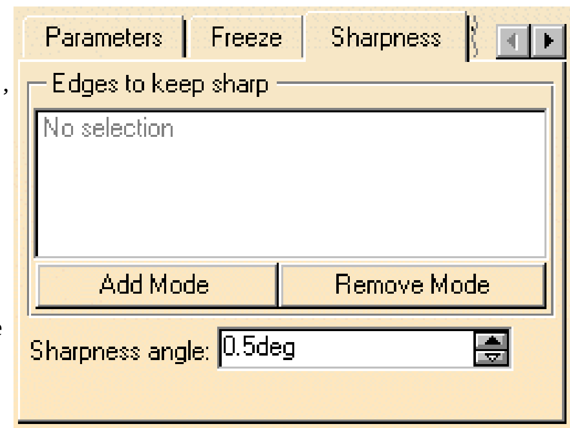
7. Click **OK** to create the healed surfaces.

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



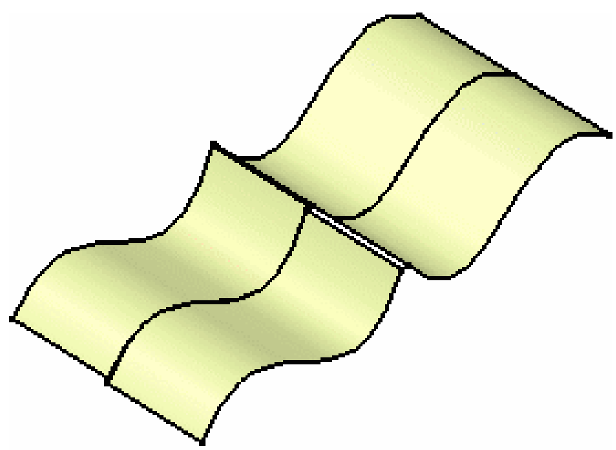
- Check the **Surfaces' boundaries** option from the **Tools -> Options** menu item, **General -> Display -> Visualization** tab to display the boundaries. This may be especially useful when selecting, and also to identify gaps.

Sharpness tab

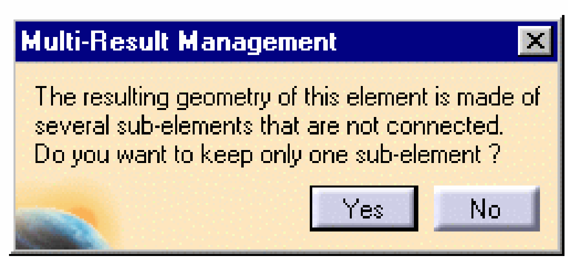


- Provided the **Tangent** mode is active, you can retain sharp edges, by clicking the **Sharpness** tab, and selecting one or more edges.
  You can edit the list of edges as described above for the list of elements to be healed.

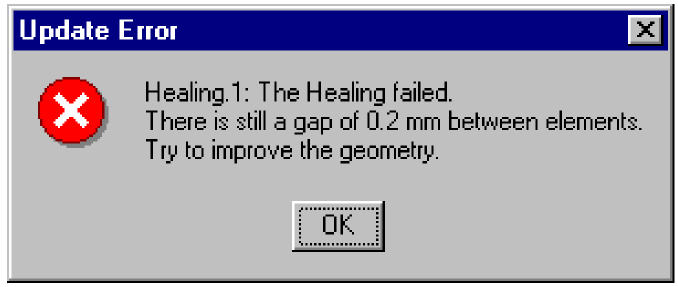
- The **Sharpness angle** allows to redefine the limit between a sharp angle and a flat angle. This can be useful when offsetting the resulting healed geometry for example. By default this angle value is set to 0.5 degree.



- 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 the Managing Multi-Result Operations chapter for further information.

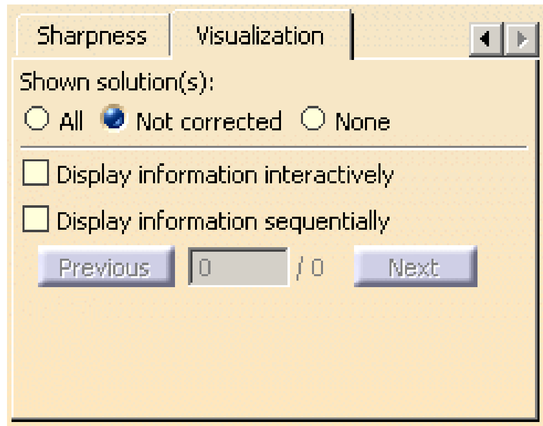


When the healing fail, an update error dialog is issued. Click OK to improve the geometry.



The erroneous elements are displayed on the geometry.

Visualization tab

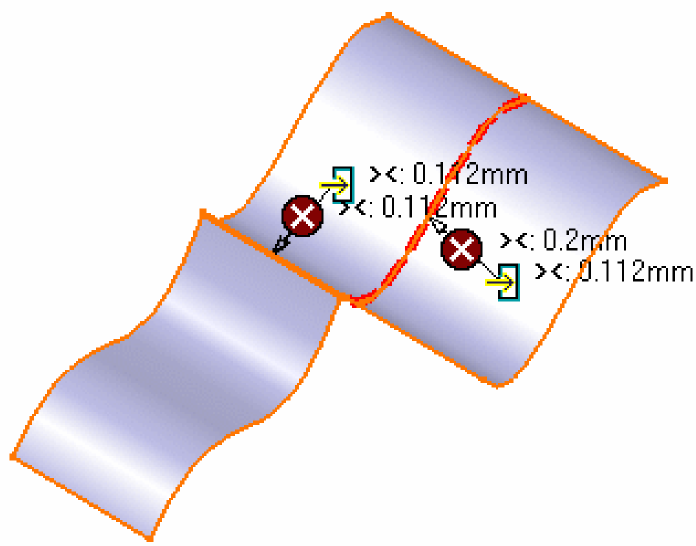


The Visualization tab enables you to better understand the discontinuities in the model and the results of the healing action.

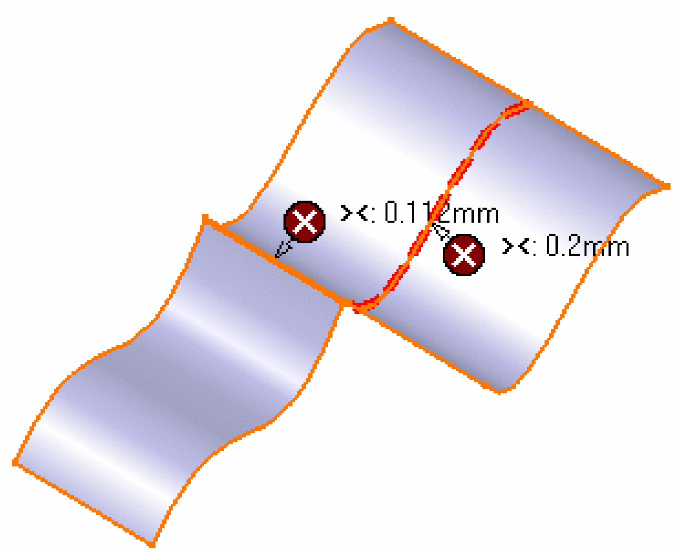
It lets you define the way the messages are displayed on the smoothed element.

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 (^)).



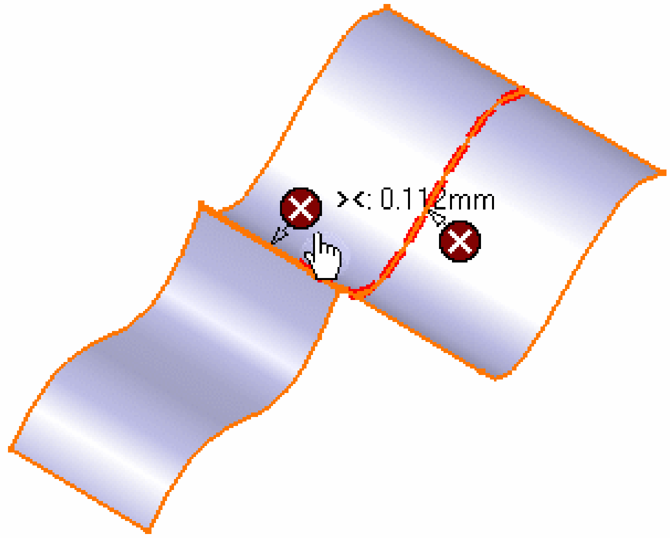
- only the messages indicating where the discontinuity is **Not corrected** and still remains.



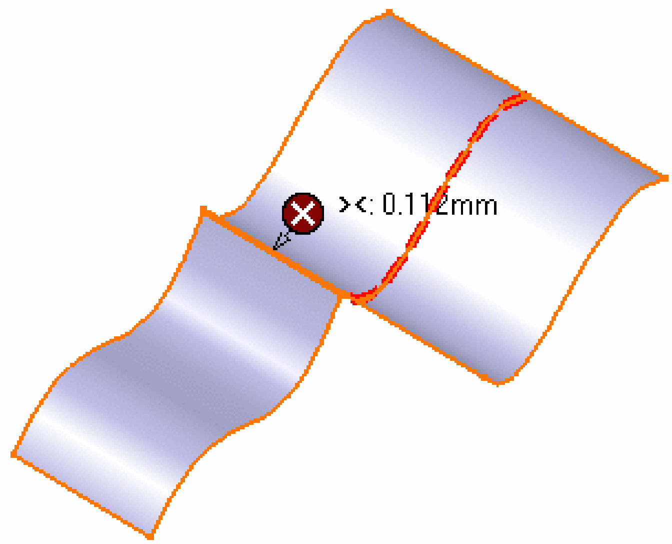
- **None** of the messages.

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



Smoothing Curves

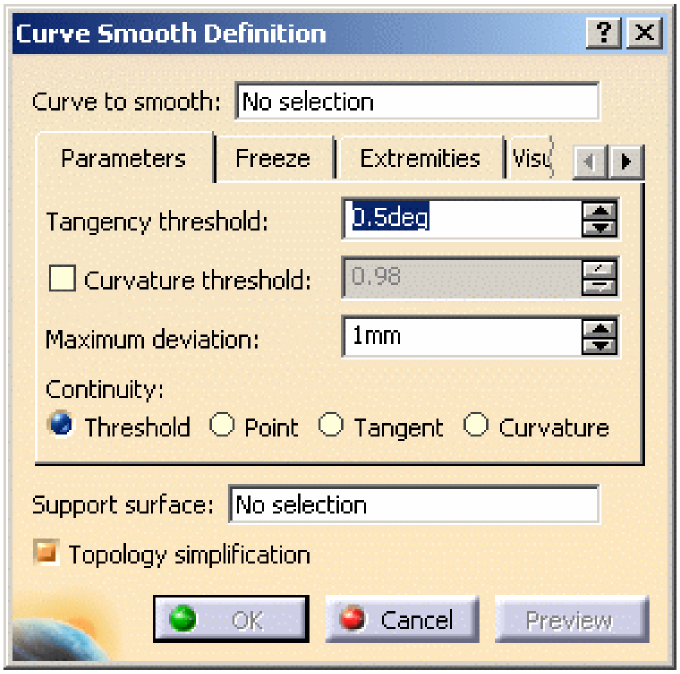
This option is only available with the Generative Shape Design 2 product.

This task shows how to smooth a curve, i.e. fill the gaps, and smooth tangency and curvature discontinuities, in order to generate better quality geometry when using this curve to create other elements, such as swept surfaces for example.

Open the Smooth1.CATPart document.

1. Click the **Curve Smooth** icon in the **Operations** toolbar.

The Curve Smooth Definition dialog box is displayed.



2. Select the curve to be smoothed.

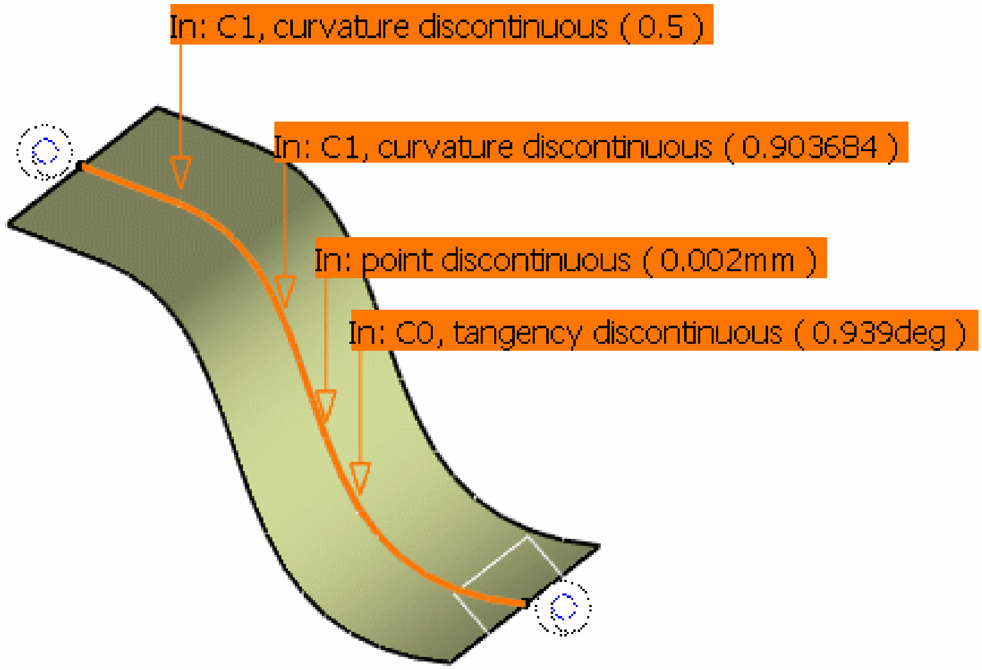
Texts are displayed on the curve indicating its discontinuities before smoothing, and type of discontinuity (point, curvature or tangency) and their values (**In** area). These values type are expressed in the following units:

- for a point discontinuity: the unit is the document's distance unit (mm by default)
- for a tangency discontinuity: the unit is the document's angular unit (degree by default)
- for a curvature discontinuity: the value is a ratio between 0 and 1 which is defined as follows:

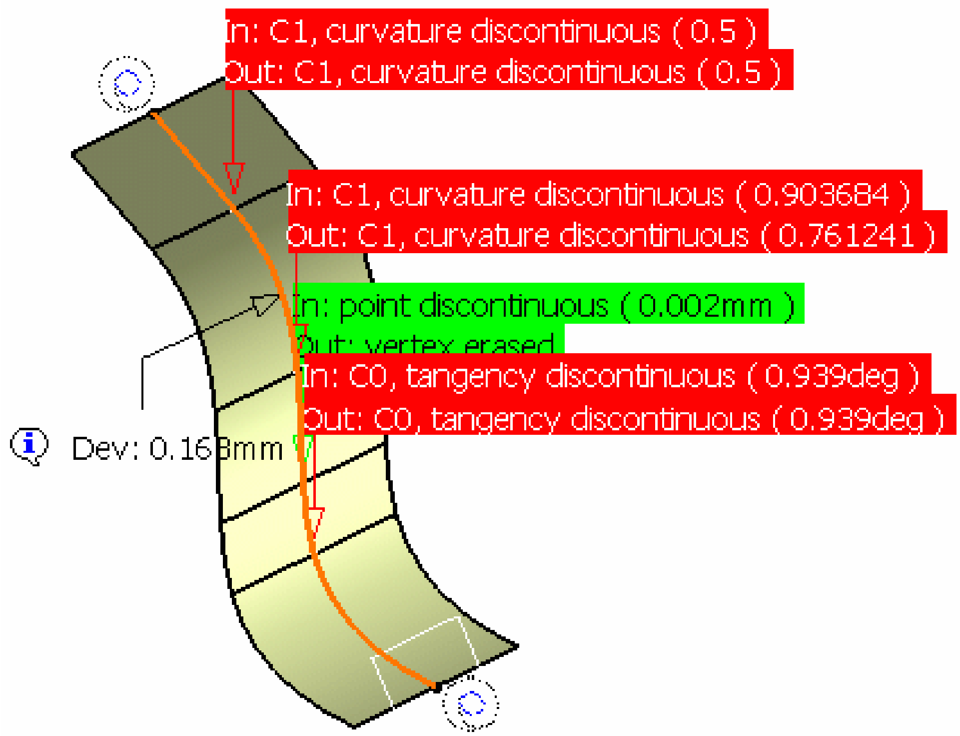
  if ||Rho1-Rho2|| / ||Rho2|| < (1-r)/r

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 smoothed.



3. Click **Preview** to display texts indicating the curve discontinuities still present after the smoothing operation, and whether they are within the threshold values (yellow box) or outside the set values (red box) (**Out** area).



The following elements C0, C1 and C2 that are displayed before the discontinuity information indicate that the vertex is respectively point continuous (C0), tangent continuous (C1) and curvature continuous (C2).

The value and location of the Maximum deviation between the curve to be smoothed and the smoothed curve are displayed in the 3D geometry.

In the example, from top to bottom, once the curve is smoothed:

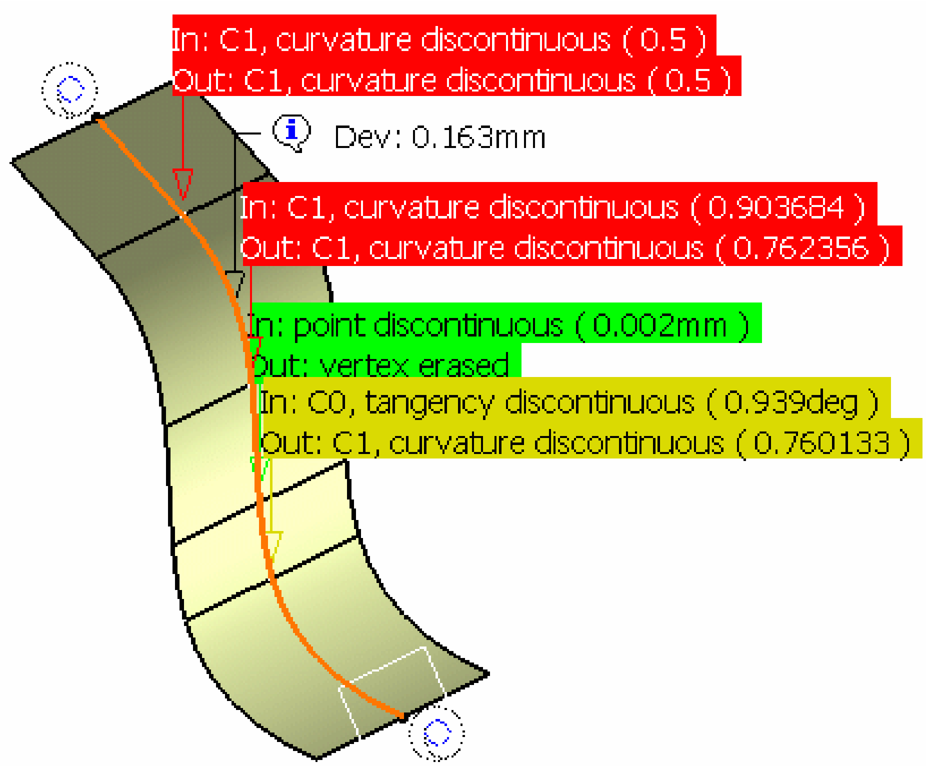
- the tangency discontinuity still is present
- there is no more discontinuity, the point discontinuity is corrected
- the curvature discontinuity still is present, even though it is slightly modified (different **In** and **Out** values)
- the curvature discontinuity still is present and not improved at all

Basically:

- a red box indicates that the system could not find any solution to fix the discontinuity while complying with the specified parameters
- a yellow box indicates that some discontinuity has been improved, where there was a point discontinuity there now is a tangency discontinuity for example
- a green box indicates that the discontinuity no longer exists; it has been smoothed.

Defining tangency and curvature thresholds, the maximum deviation and the continuity

4. From the **Parameters** tab modify the **Tangency threshold**, that is the tangency discontinuity value below which the curve is smoothed. If the curve presents a tangency discontinuity greater than this threshold, it is not smoothed.



If you increase the threshold value to

1.0 in our example, you notice that the Tangency discontinuity which value was below 1 changes to a curvature discontinuity.

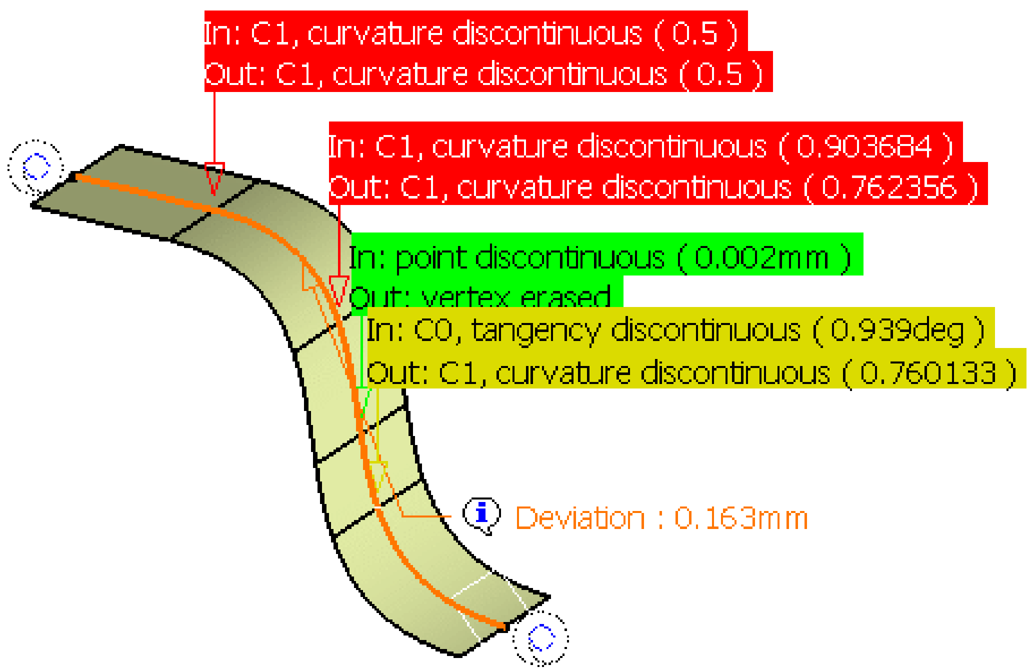
5. Similarly, you can check the **Curvature threshold** button to set curvature discontinuity value below which the curve is smoothed.

6. Define a **Maximum deviation** value to set the allowed deviation between the initial curve and the smoothed curve.
   Therefore, the resulting smoothed curve fits into a pipe which radius is the maximum deviation value and the center curve is defined by the selected curve.

7. Define the **Continuity**, that is the correction mode for the smoothing:

- **Threshold**: default mode. The tangency and curvature thresholds options are taken into account.
- **Point** (there is no point discontinuity in our example)
- **Tangency**

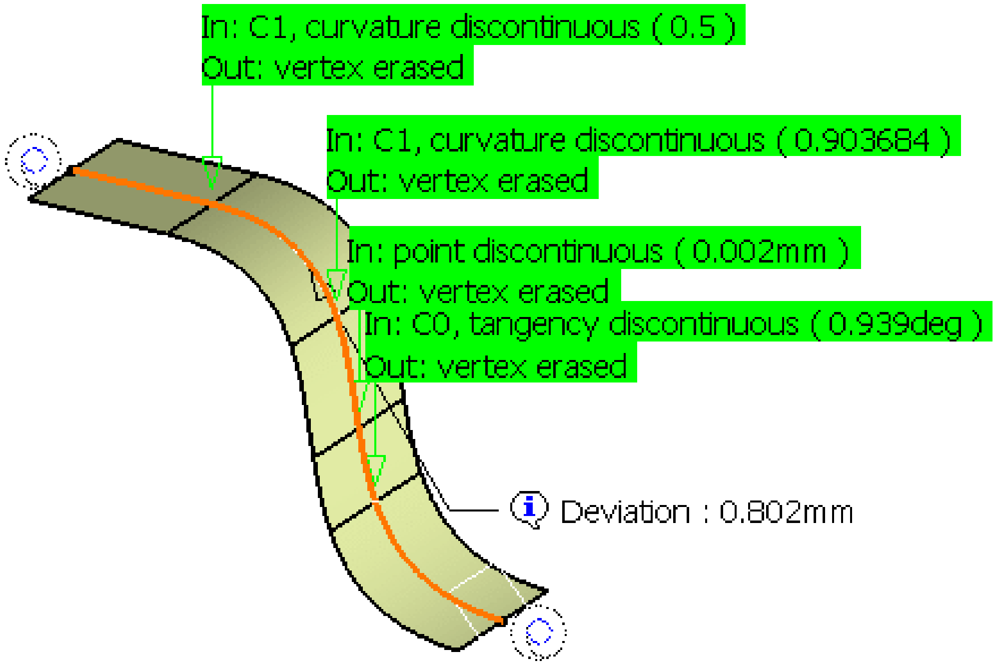
  You notice that the Tangency discontinuity changes to a curvature discontinuity.



In this case, the **Tangency threshold** field is grayed out and the defined value is ignored.

- **Curvature**

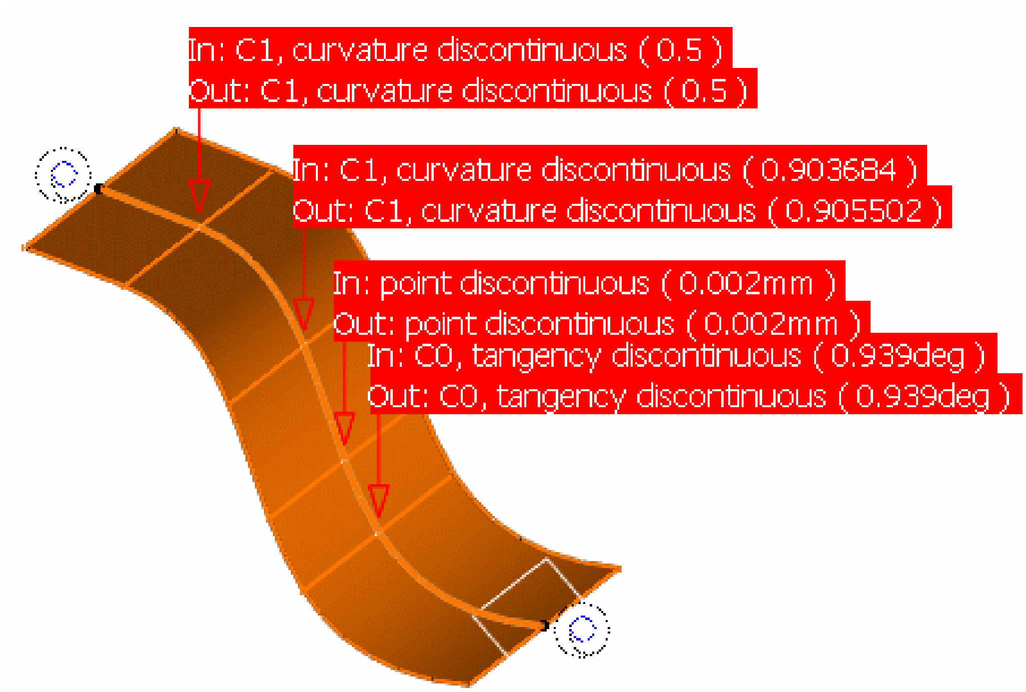
  You notice that there is no discontinuity any more.



In this case, the **Curvature threshold** field is grayed out and the defined value is ignored.

Optionally, you can select a surface on which the curve lies.

In this case the smoothing is performed so that the curve remains on the **Support surface**. From this ensues that the maximum degree of smoothing is limited by the support surface's level of discontinuity.

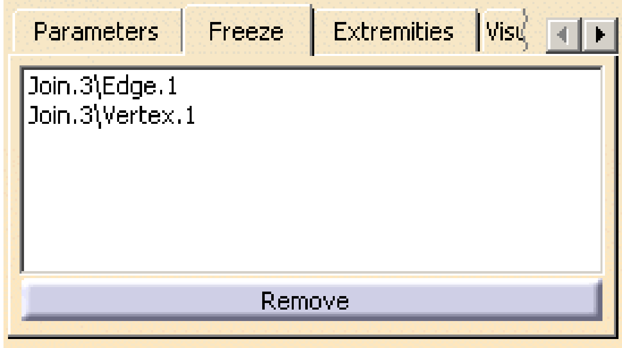


Selecting Elements not to be smoothed

8. Click the **Freeze** tab.

This tab enables you to select sub-elements of the curve that should not be smoothed. These sub-elements can either be vertices or edges. In case of a vertex, the local neighborhood remains unchanged, thus keeping the discontinuity.

The **Remove** button enables to remove a single or a set of sub-elements.

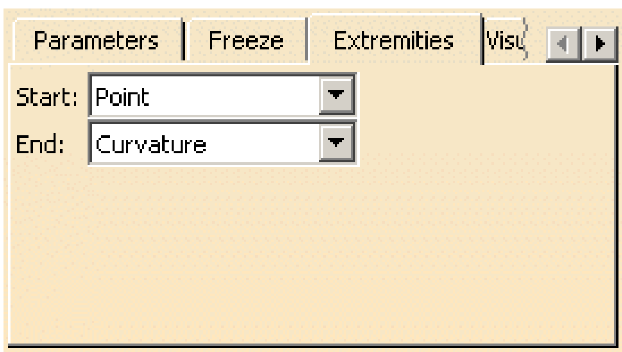


External elements cannot be selected as sub-elements.

Setting continuity conditions

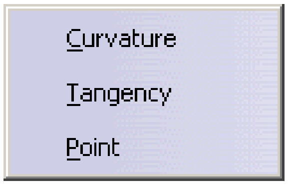
You now set continuity conditions on the resulting smoothed curve for each extremity with regards to the input curve. As a comparison basis, the continuity condition was previously always curvature: the output curve had the same extremity points, tangencies and curvatures as the input curve.

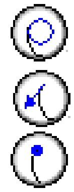
9. Click the **Extremities** tab and define the continuity conditions at each curve's extremity:



- **Curvature** (by default): extremity point, tangency and curvature are the same
- **Tangency**: extremity point and tangency are the same (curvature can be different)
- **Point**: extremity points are the same (tangency and curvature can be different)

You can also right-click the icon at the curve's extremity and choose one of the following options:



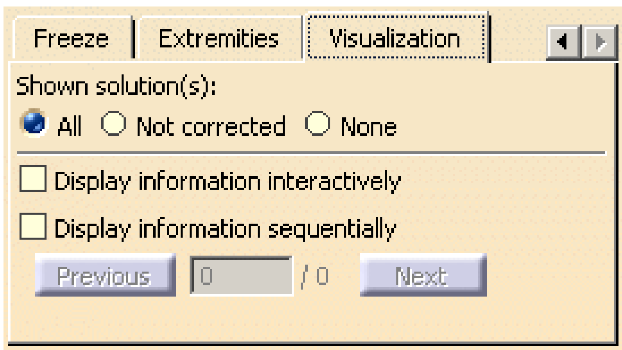


Point and Tangency conditions can only be successfully applied if the Maximum Deviation is larger than 0.005mm. Note that these extremity conditions do not affect closed curves.

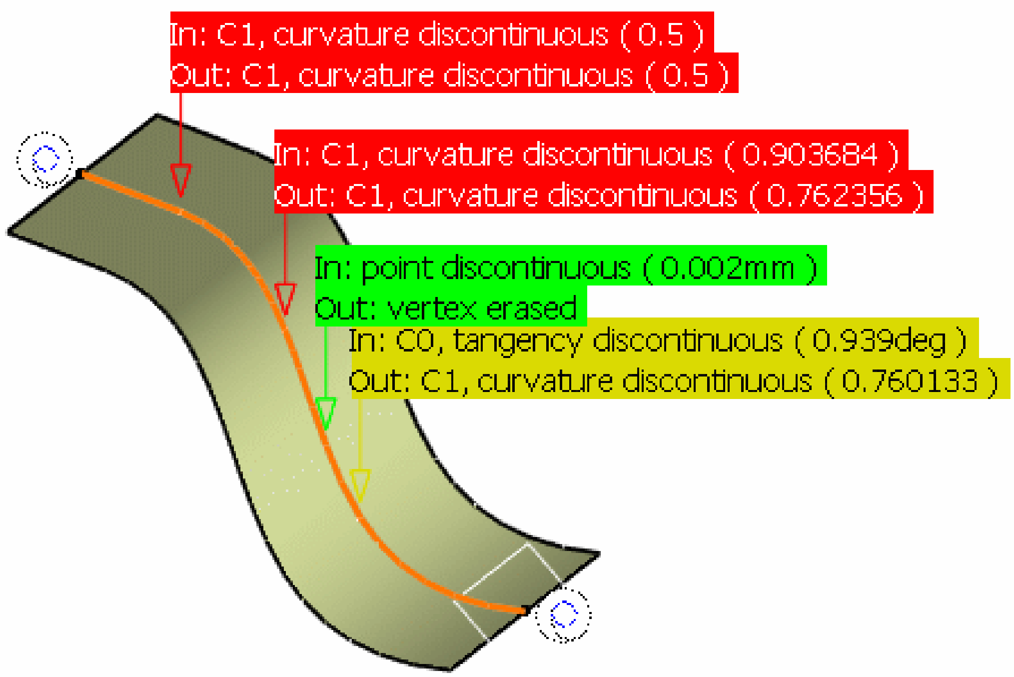
You can also sequentially move from one conditions to the next one by clicking on the icons.

Visualizing messages

10. Click the **Visualization** tab.

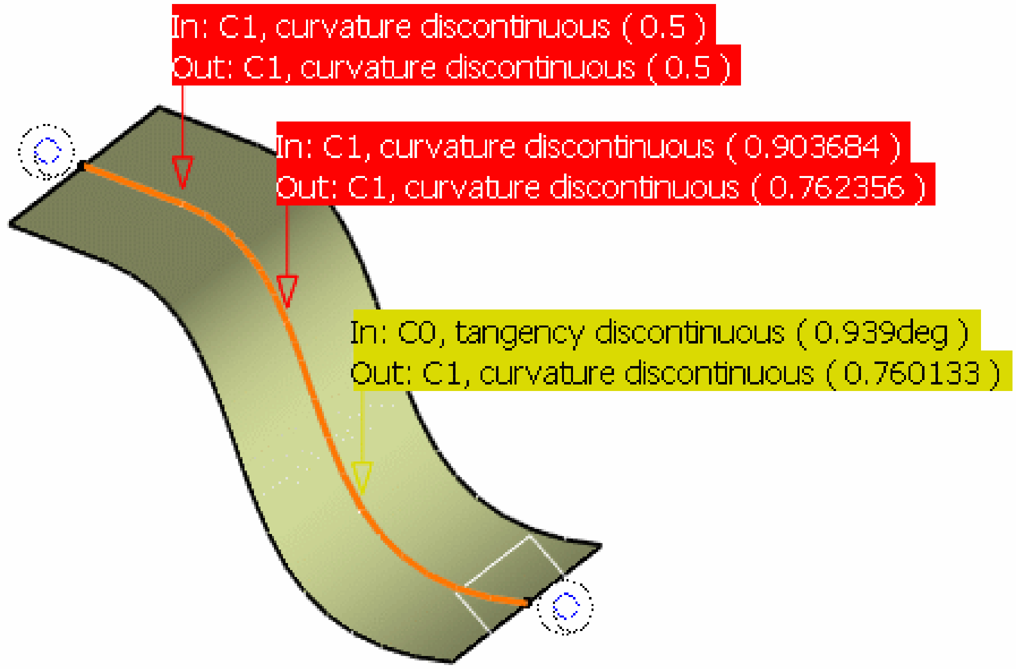


This tab lets you define the way the messages are displayed on the smoothed element. You can choose to see:



- **All** the messages: those indicating where the discontinuity remains (red box) as well as those indicating where the discontinuity type has changed, or allows smoothing.

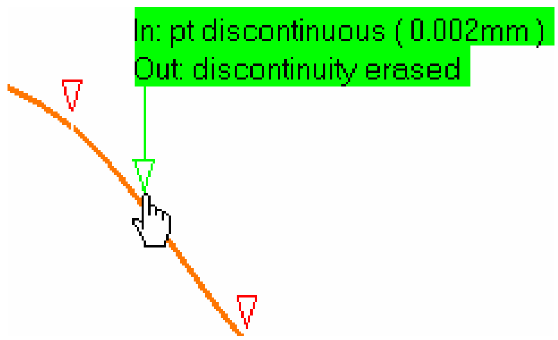
- only those messages indicating where the discontinuity is **Not corrected** and remains.



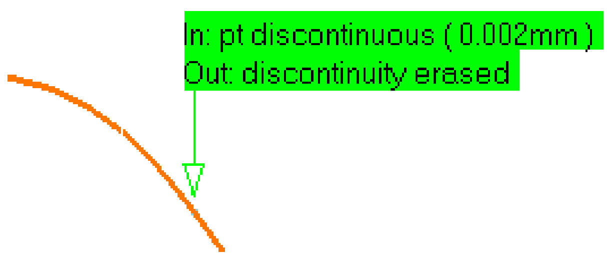
- **None** of the messages.

You can also choose to:

- **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 the other using the backward/forward buttons



The **Topology simplification** check button automatically deletes vertices on the curves when the curve is curvature continuous at these vertices, thus reducing its number of segments.
When this is the case, the displayed text indicates: **Out: discontinuity erased** to inform you that a simplification operation took place.
This text is also displayed when two vertices are very close to each other and the system erases one to avoid the creation of very small edges (i.e. shorter than 10 times the model tolerance) between two close vertices.

11. Click **OK**.

    The smoothed curve (identified as Curve smooth.xxx) is added to the specification tree.

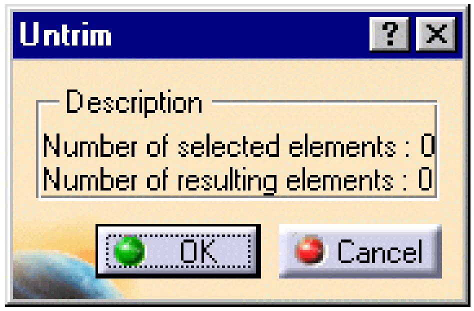
When smoothing a curve on support that lies totally or partially on the boundary edge of a surface or on an internal edge, a message may be issued indicating that the application found no smoothing solution on the support. In this case, you must enter a Maximum deviation value smaller than or equal to the tolerance at which two elements are considered as being only one (0.001mm by default) to keep the result on the support.

Restoring a Surface

In this task you will learn how to restore the limits of a surface or a curve when it has been split using the **Break Surface or Curve** icon (see Splitting Geometry for the Generative Shape Design workbench).

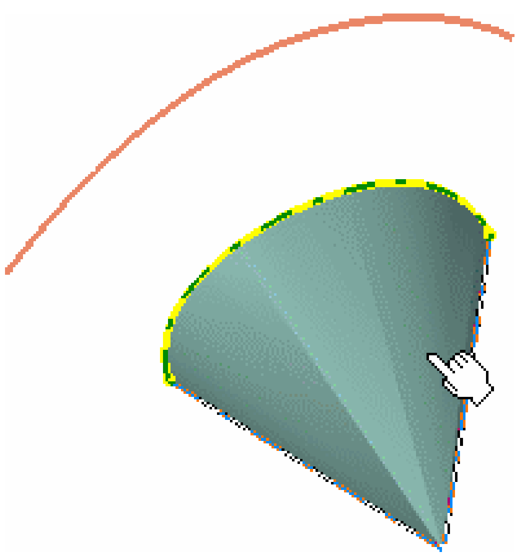
Open the Untrim1.CATPart document.

1. Click the **Untrim** icon in the **Operations** toolbar.



The Untrim dialog box is displayed.

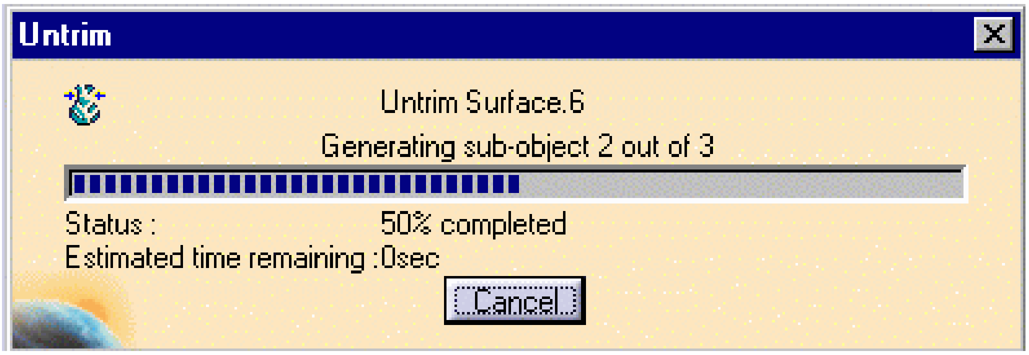
2. Select the surface which limits should be restored.



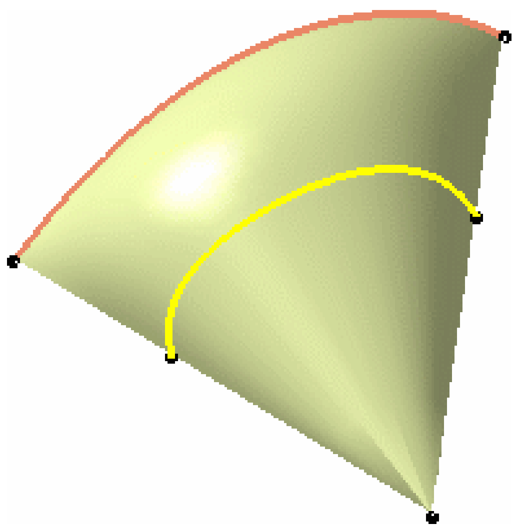
The dialog box is updated accordingly.

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%).



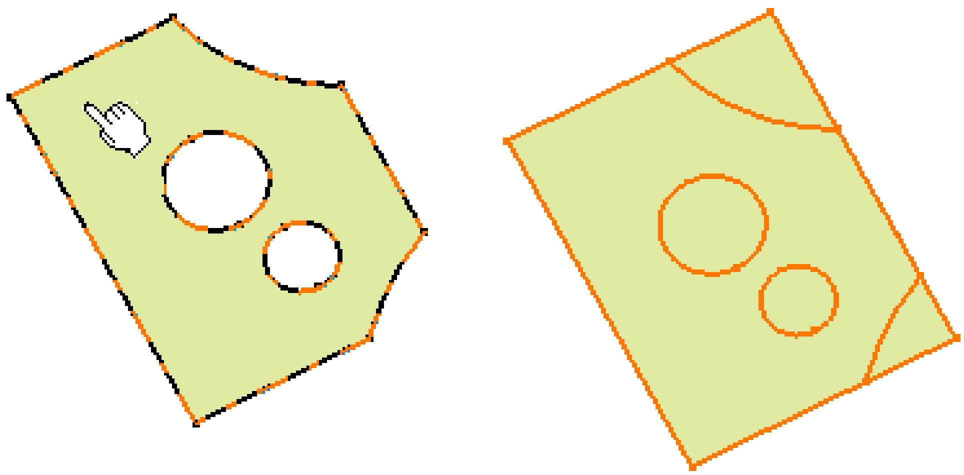
The initial surface is automatically restored.



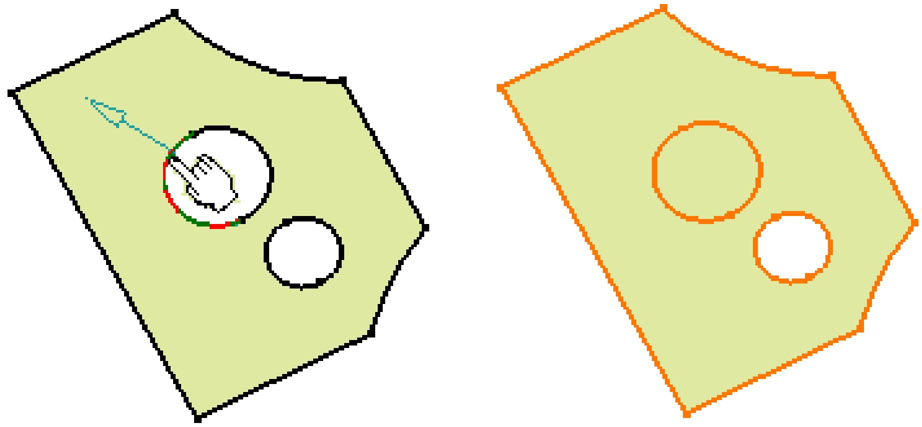
The restored surface or curve is identified as Surface Untrim.xxx or Curve Untrim.xxx.

You can perform a local untrim on faces. Three modes of selection are available:

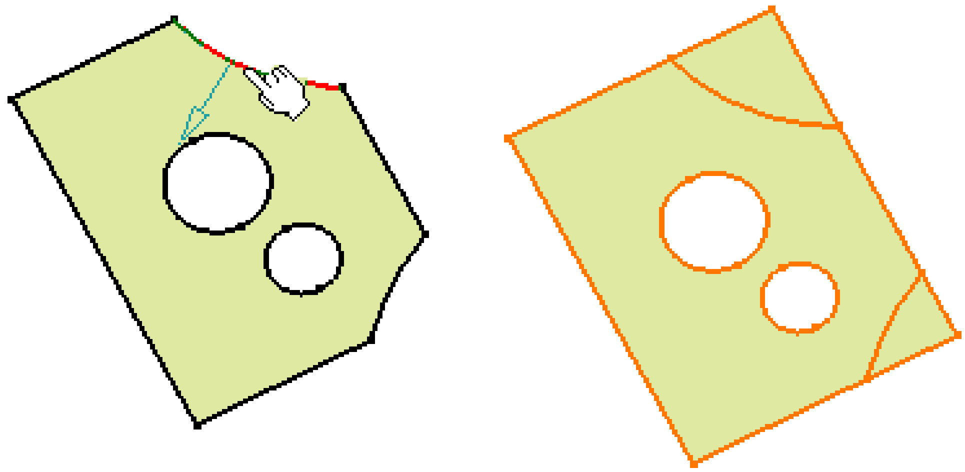
- selection of the face: the initial surface is restored



- selection of an inner loop: only this loop is restored



- selection of the outer loop: only this loop is restored



- 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.
- If the surface to be restored is closed (in the case of a cylinder) or infinite (in the case of an extrude), the limits of the untrim feature will be the bounding boxes of the initial surface. Therefore, the initial surface and the untrim surface may be identical.
- You can individually select a vertex or a boundary from the restored surface or curve.
- Multi-selection is available and allows to create several untrim features in one step. All untrim features will appear in the specification tree.
- The datum creation capability is available from the Dashboard.

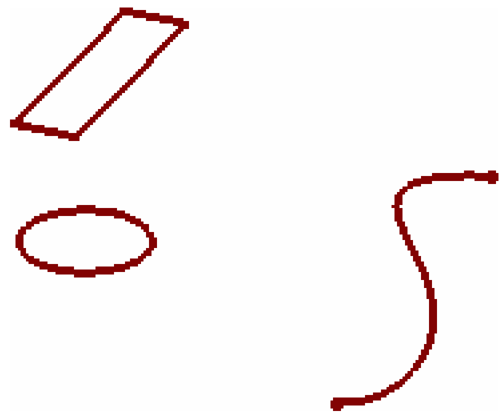
Disassembling Elements

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.

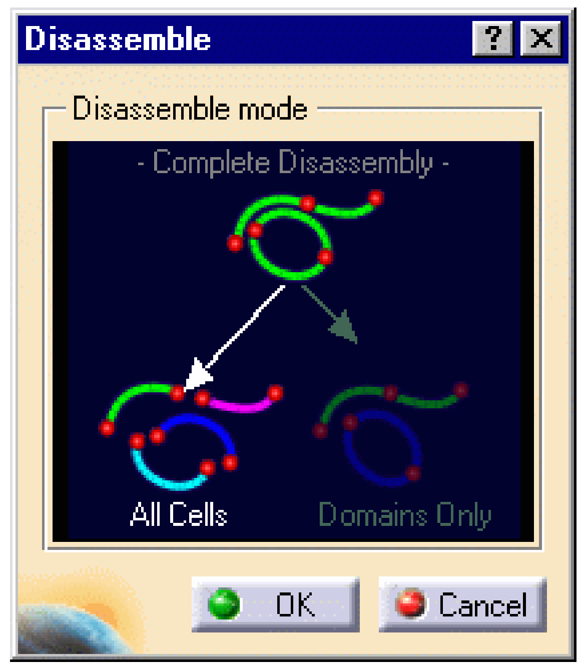
1. Select the element to be disassembled.

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

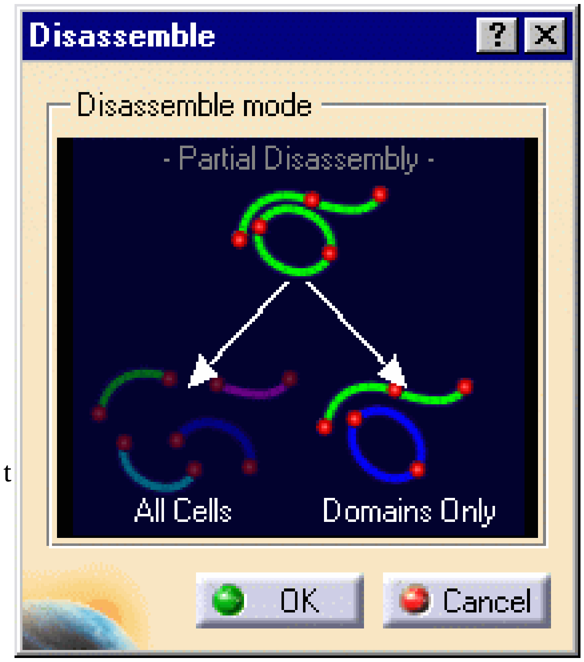


Here we selected the join made of three elements, each made of several cells.

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



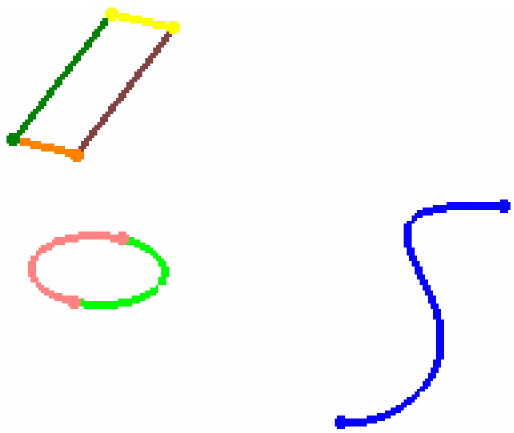
The Disassemble dialog box is displayed.



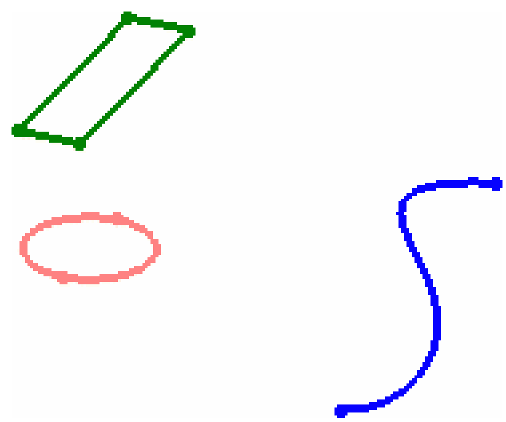
3. Choose the disassembling mode:

- **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.



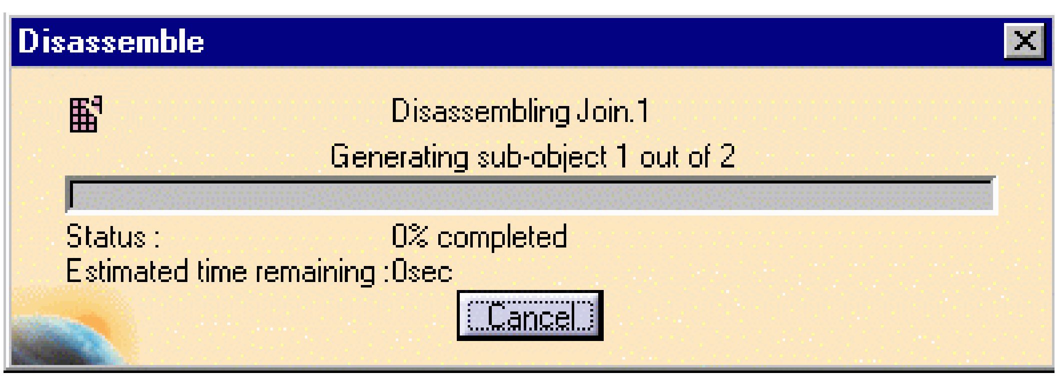
Results when disassembling all cells (seven curves are created)



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 surface is disassembled, that is to say independent surfaces are created, that can be manipulated independently.

Multi-selection is available.

Splitting Geometry

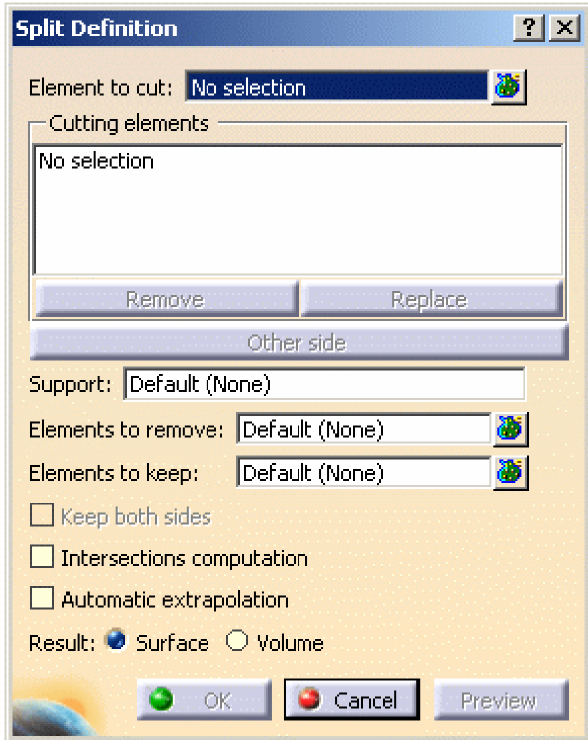
This task shows how to split a surface or wireframe element by means of a cutting element.
You can split a wireframe element by a point, another wireframe element or a surface; or a surface by a wireframe element or another surface.

- Keeping or Removing Elements
- Intersections and extrapolations
- Splitting Wires
- Splitting a surface by a curve or a surface by a surface
- Splitting Volumes

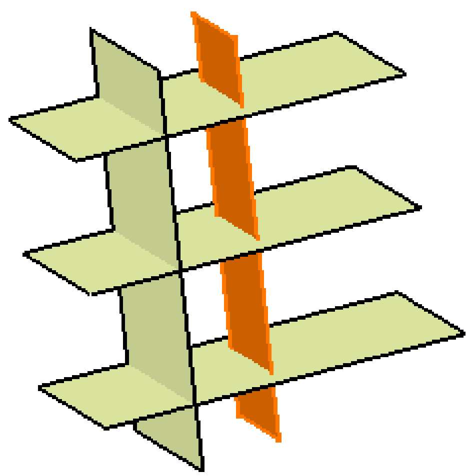
Open the Split1.CATPart document.

1. Click the **Split** icon .

The Split Definition dialog box appears.

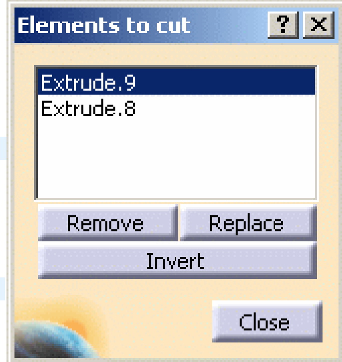


2. Select the element to be split.



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

You can select several elements to cut. In that case, click the **Element to cut** field again or click the bag icon . The Elements to cut field opens. Select as many elements as needed. Click **Close** to return to the Split Definition dialog box. The number of selected elements is displayed in the **Element to cut** field.

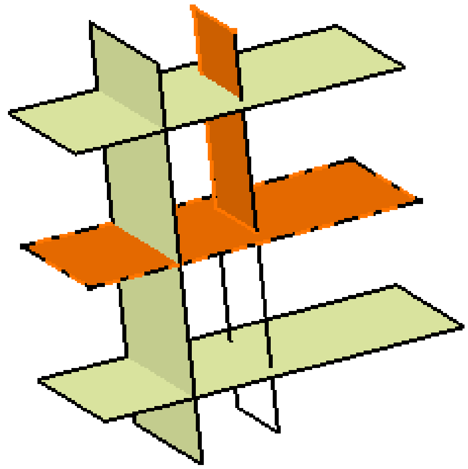


Use the **Remove** and **Replace** buttons to modify the elements list.
Use the **Invert** button to reverse the portion to be kept, element by element.

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.
This option applies on all selected elements to cut.



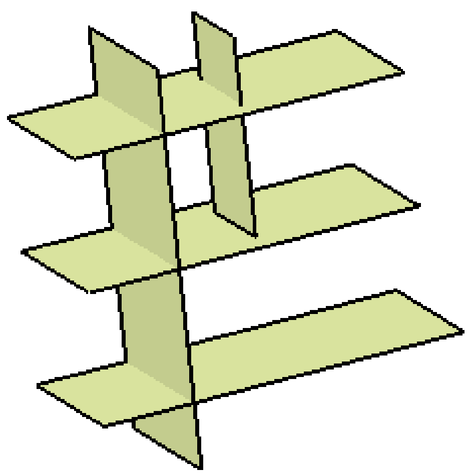
- You can select several cutting elements. In that case, note that the selection order is important as the area to be split is defined according to the side to be kept in relation to the current splitting element.

- 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.

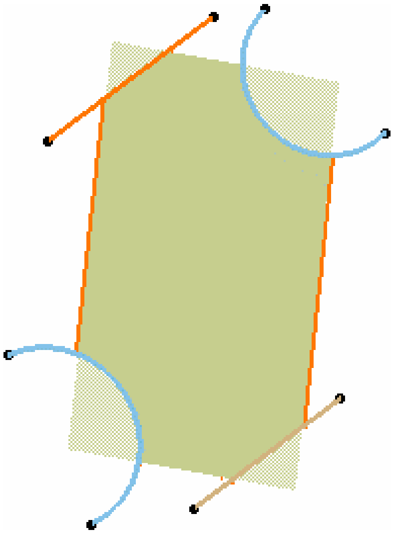
4. Click OK to split the element.

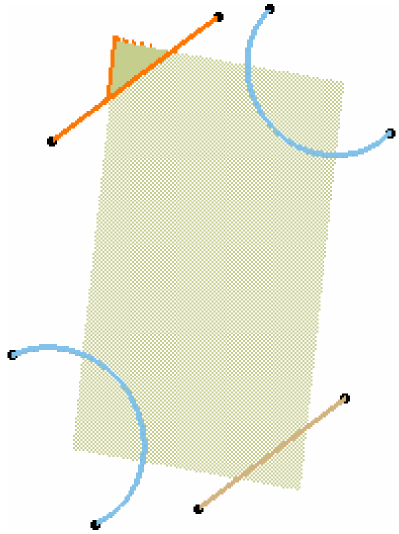
The created element (identified as Split.xxx) is added to the specification tree.

In the case several elements to cut were used, the created elements are aggregated under a Multi-Output.xxx feature.



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.



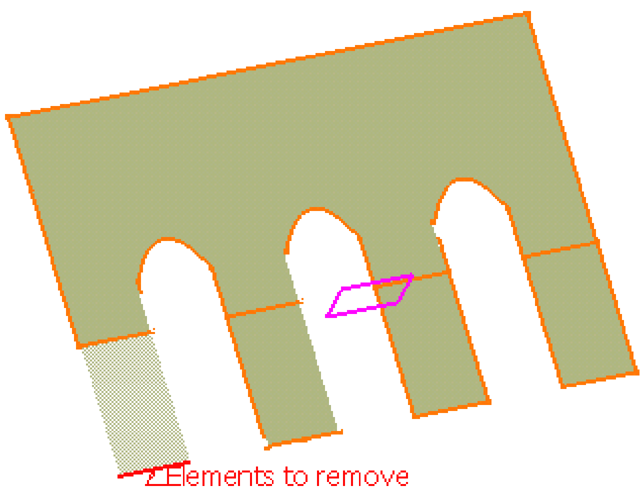


Would you need to remove, or replace, one of these cutting elements, select it from the list and click the **Remove** or **Replace** button.

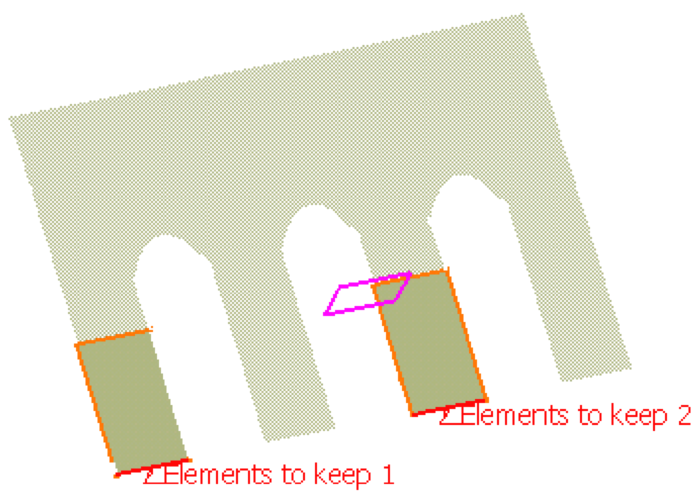
Keeping or Removing Elements

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

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

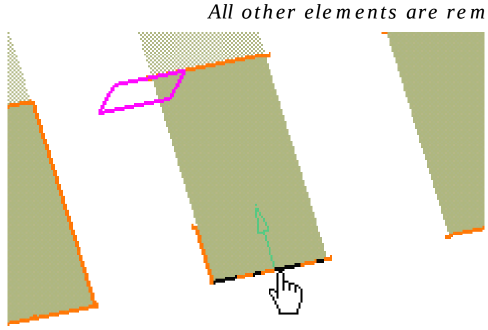


Only the selected element is removed.
All other elements are kept.



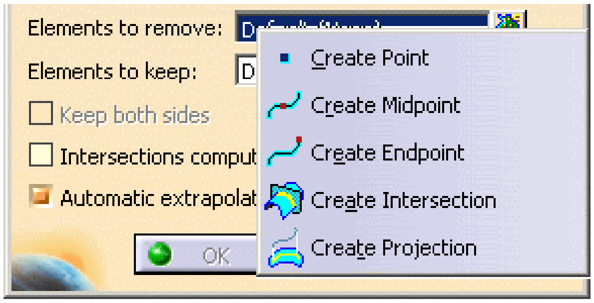
The selected elements are kept.
All other elements are removed.

- 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.

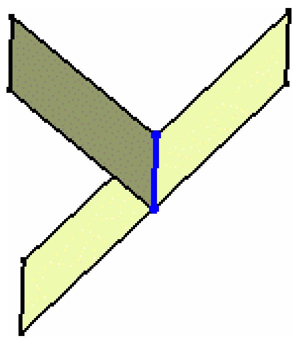
- Check the **Keep both sides** option to retain the other side of the split element after the operation. In that case it appears as aggregated under the first element.
  Therefore both split elements can only be edited together and the aggregated element alone cannot be deleted.

  If you use the Datum mode, the second split element is not aggregated under the first one, but two datum surfaces are created.

In case there are several elements to cut, the **Keep/Remove** options only apply on the first selected element.

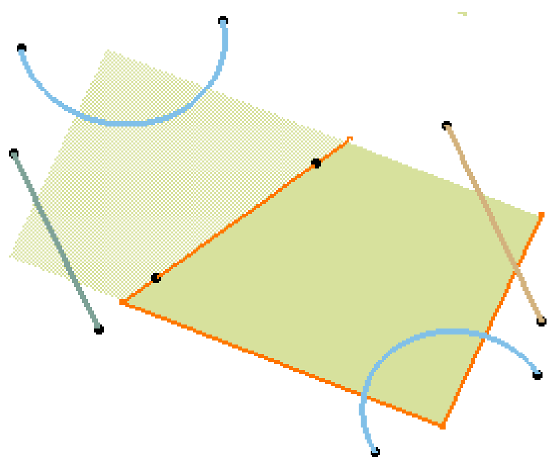
Intersections and extrapolations

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



In case there are several elements to cut, the **Intersections computation** option only applies on the first selected element.

- Uncheck the **Automatic extrapolation** button if do not you want the automatic extrapolation of the cutting curve.
  When a splitting curve is extrapolated, the extrapolation will performed on the original curve, providing the underlying geometry (that is the curve) is long enough to be used for the extrapolation.
  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.

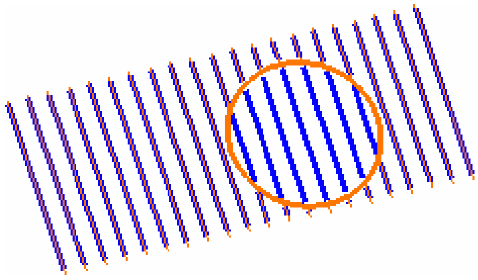


This option is available in the case of a split surface/curve or surface/surface.

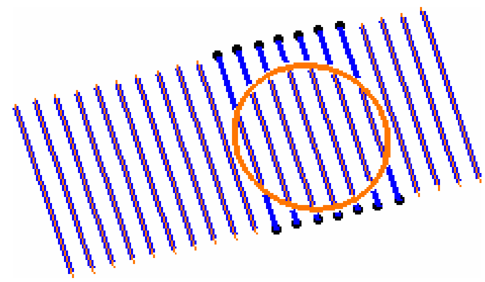
Splitting Wires

- 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.

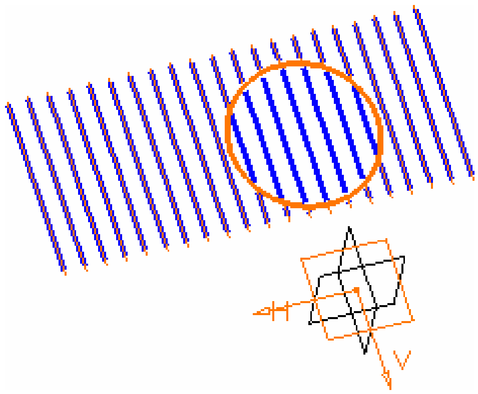
The non disconnected elements of the element to cut are kept in the result of the split.



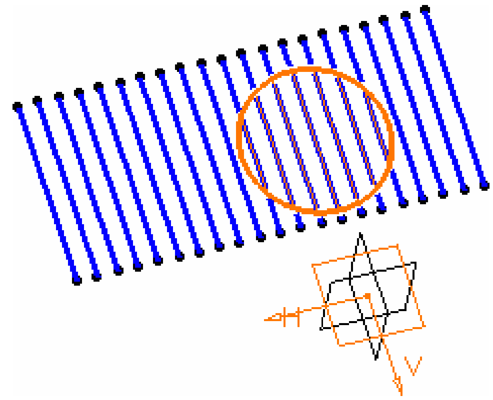
Splitting with no support selected: first solution



Splitting with no support selected: second solution



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



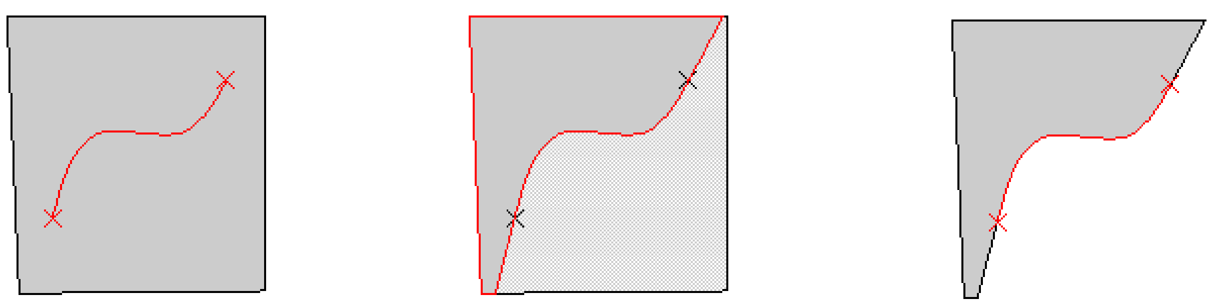
Splitting with a selected support (xy plane): second solution

Splitting a surface by a curve or a surface by a 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.

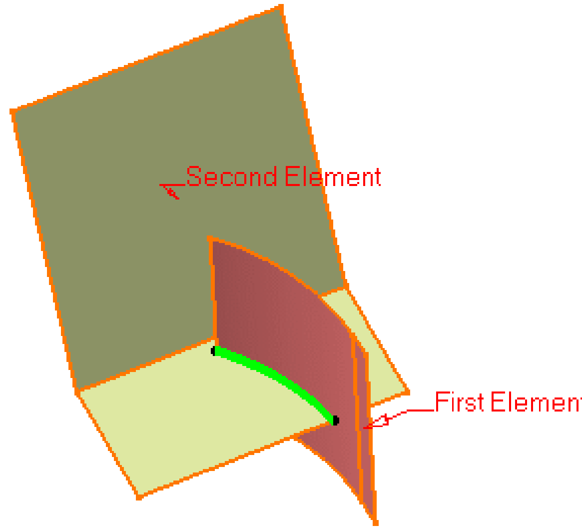


If the cutting element does not reach the free edges of the element to cut, an extrapolation in tangency is performed using the part of the cutting element that lays down the surface.

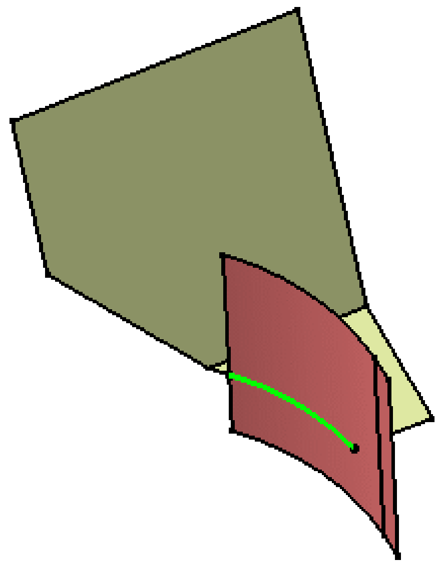
Split surface/surface

Open the Split2.CATPart document.

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



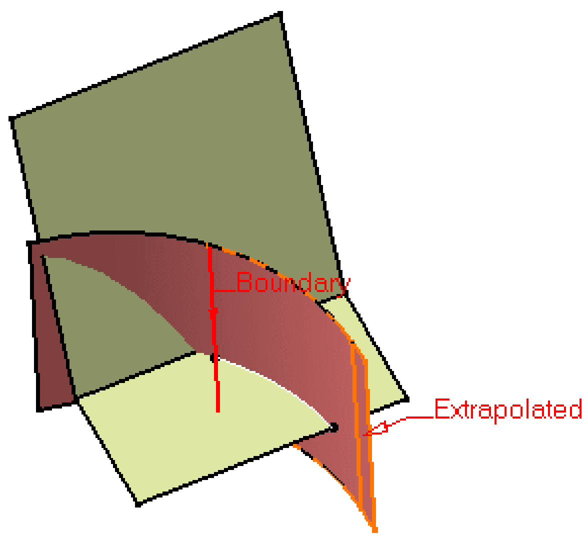
2. Then, the result of the intersection is automatically extrapolated in tangency up to the closest free edges of the element to cut.
   The result of the extrapolation is used as the cutting element and the split is created.



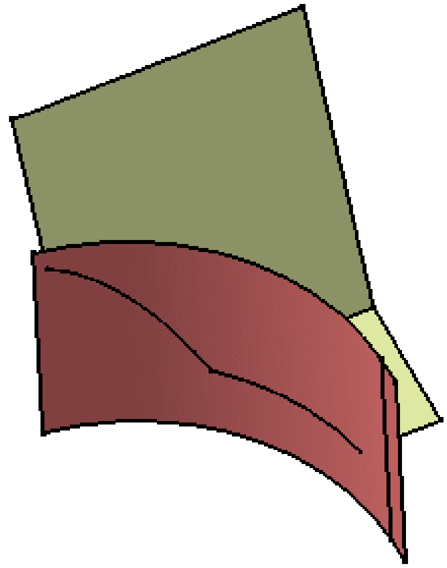
Please note that it is **not** the cutting element which is extrapolated but the result of the intersection.

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.

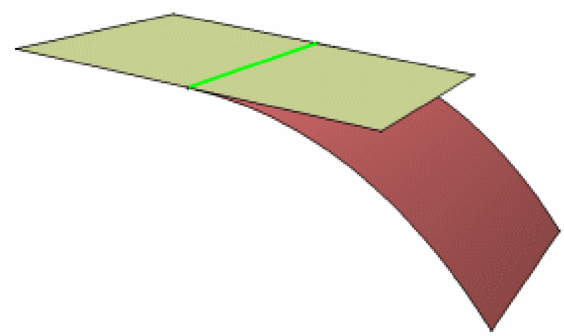


Avoid using input elements which are tangent to each other since this may result in geometric instabilities in the tangency zone.

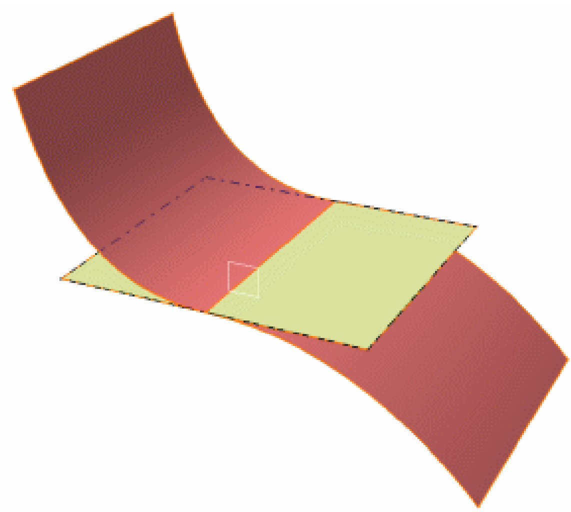
In case surfaces are tangent or intersect face edges, please process as follow in order to avoid indeterminate positioning.

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

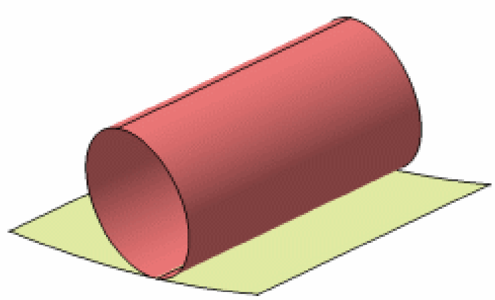


Steps 2 and 3 may be optional if the tangency constraint between the two surfaces has been clearly defined by the user during the surface creation.



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.

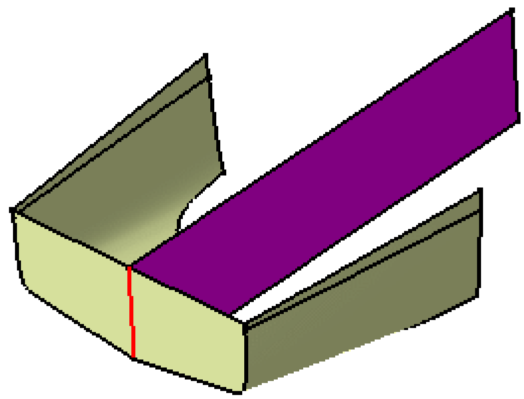
When these cases cannot be avoided, it is recommended, first to create the intersection between the two surfaces, then to split the element to cut with the resulting intersection. Doing so, the position can be properly defined but the instability of the result relating to the intersection remains.



Splitting Volumes

Providing the element to be cut is a volume and the cutting element is a volume or a surface, you can choose whether you want the result of the split to be a surface or a volume. To do so, switch to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface.
Note that the switch between surface and volume is greyed out when editing the feature.
If the result of the split is a volume, the split is a modification feature.
If the result of the split is a surface, the split is a creation feature.
To have further information about volumes, please refer to the Creating Volumes chapter.

- 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.



- When splitting a closed surface or a curve by connex elements, an error message is issued. You need to create a join feature of non connex elements and cut the closed surface or curve with this join feature.

- The selection of the feature prevails over the selection of the sub-element.
  To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
  For further information, refer to the Selecting using a Filter chapter in the *CATIA Infrastructure User's Guide*.

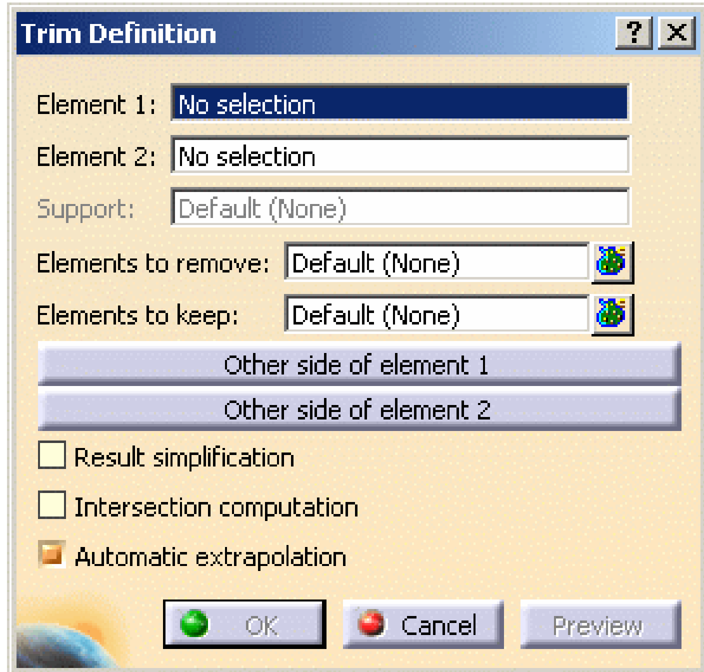
Trimming Geometry

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

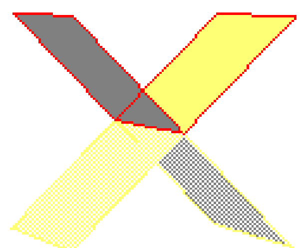
Open the Trim1.CATPart document.

1. Click the **Trim** icon .

The Trim Definition dialog box appears.



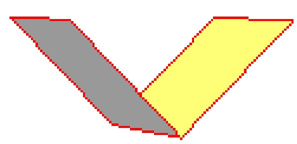
2. Select the two surfaces or two wireframe elements to be trimmed.



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.

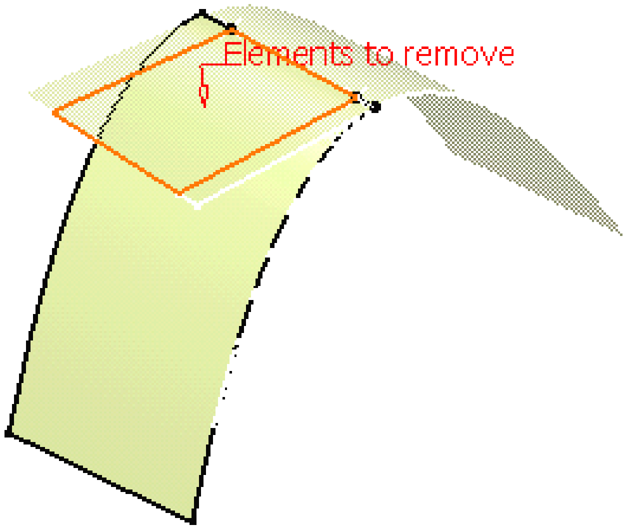


The trimmed element (identified as Trim.xxx) is added to the specification tree.

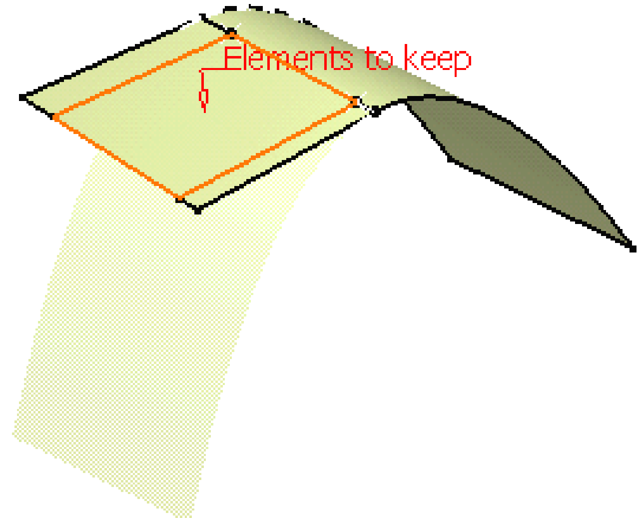
- You should make your selections by clicking on the portions that you want to keep after the trim.
- Please refer to the Splitting Geometry chapter in the case surfaces intersect face edges.

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.



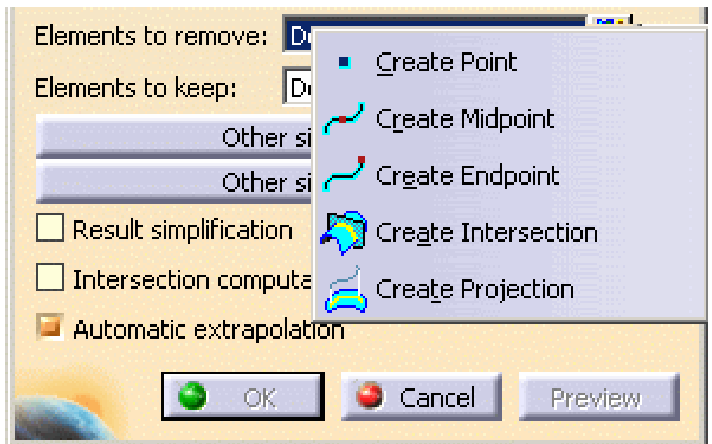
Only the selected portion is removed.
All other elements are kept.



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.

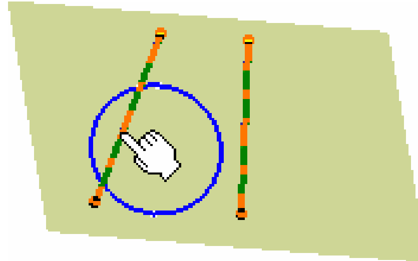


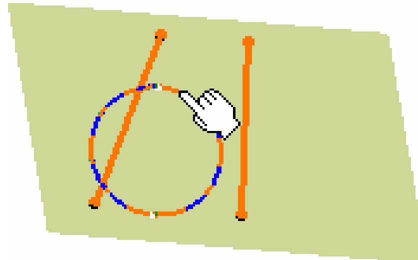
- You do not need to select elements to keep if you already selected elements to remove and vice-versa.

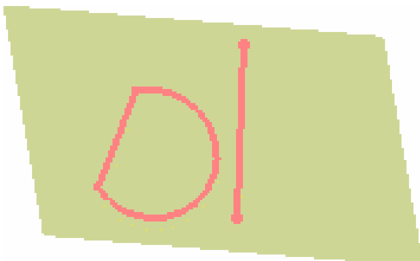
- When trimming wires (curve, line, sketch and so forth) by another wire, you can select a support to define the area that will be kept after trimming the element. It is defined by the vectorial product of the normal to the support and the tangent to the trimming element.

This is especially recommended when trimming a closed wire.

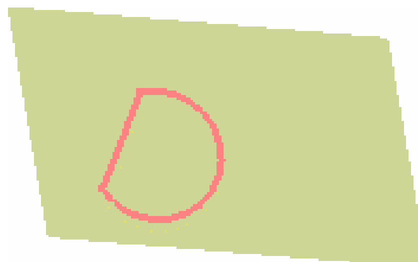
In our example, the Sketch composed of two lines (Sketch.11) is trimmed by the circle (Sketch.10).





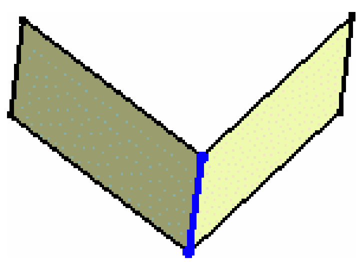


Resulting trimmed element without support selection

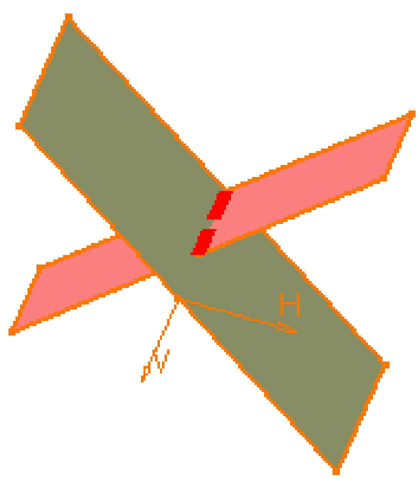


Resulting trimmed element with support selection

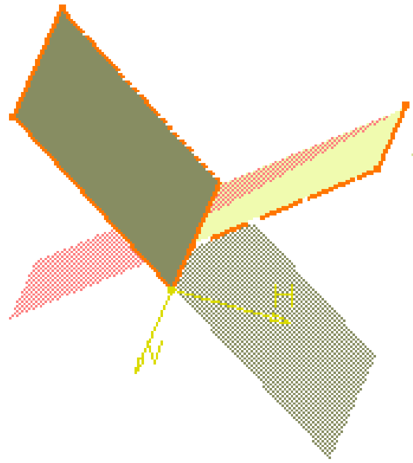
- Check the **Result simplification** button to allow the system to automatically reduce the number of faces in the resulting trim whenever possible.
- Check the **Intersection computation** button to create a completely independent element when performing the trimming operation. In that case it appears 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.



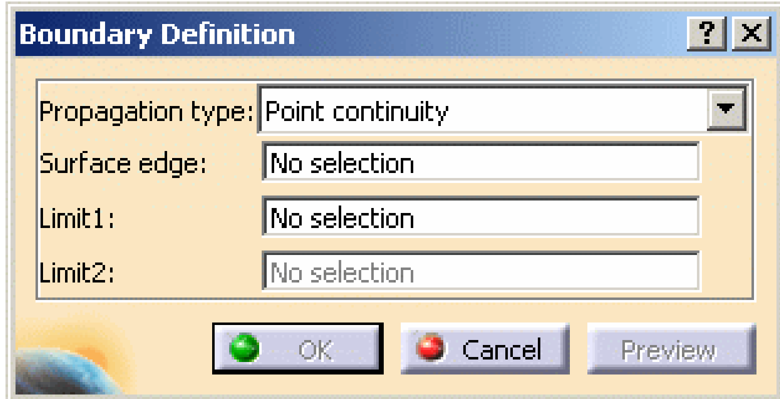
Creating Boundary Curves

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.

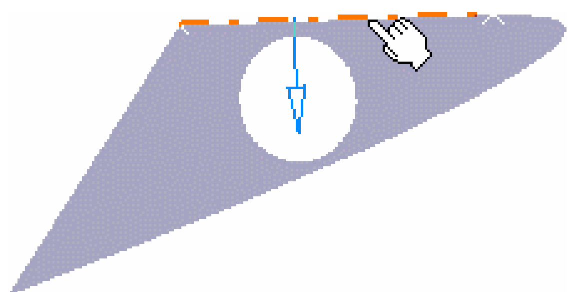


2. Use the combo to choose the **Propagation type**:

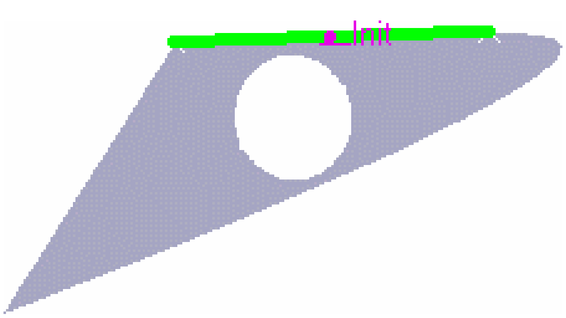
- **Complete boundary:** the selected edge is propagated around the entire surface boundary.
- **Point continuity:** the selected edge is propagated around the surface boundary until a point discontinuity is met.
- **Tangent continuity:** the selected edge is propagated around the surface boundary until 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.

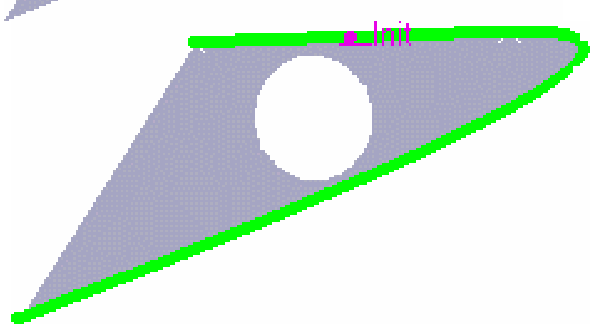
3. Select a Surface edge.



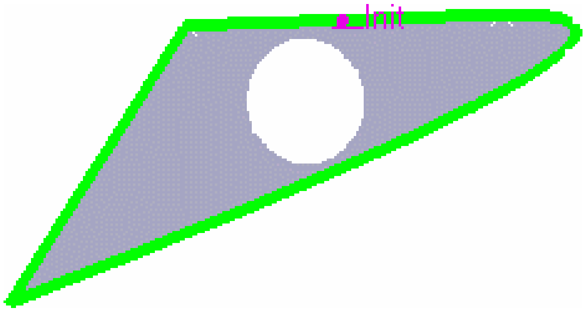
The boundary curve is displayed according to the selected propagation type.



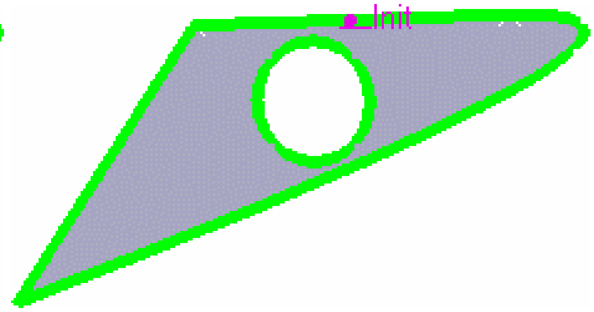
No propagation



Tangent continuity



Point continuity



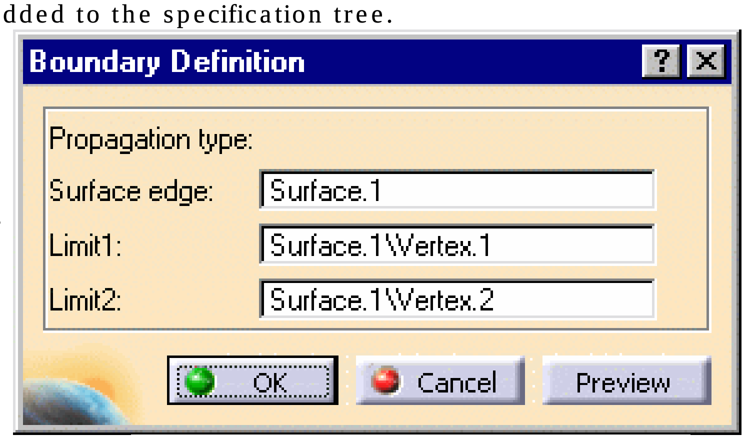
Complete boundary

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

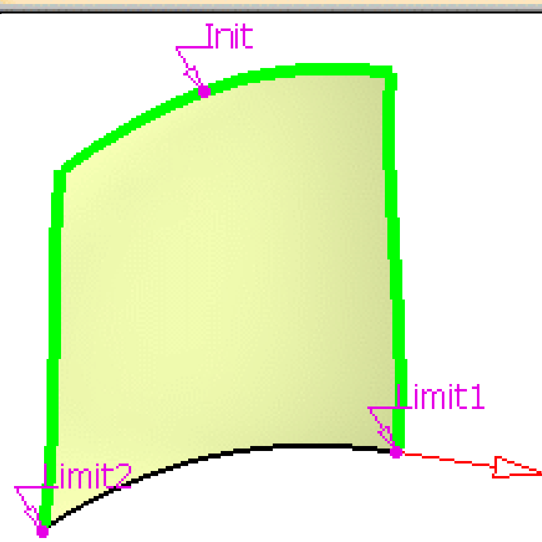
If you relimit a closed curve by means of only one element, a point on curve curve for example, the closure vertex will be moved to the relimitation point, allowing this point to be used by other features.

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

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

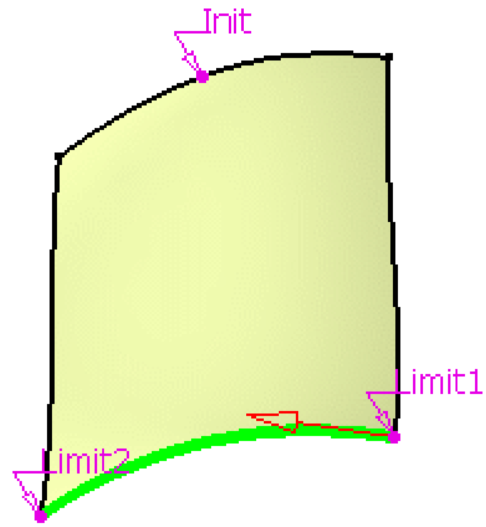


- If you select the surface directly, the Propagation type no longer is available, as the complete boundary is automatically generated.



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, volumes 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.

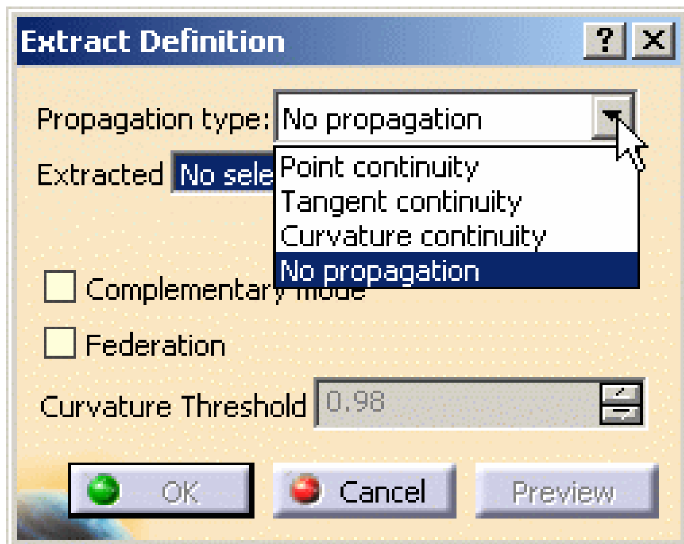
Open the Extract1.CATPart document.

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

The selected element is highlighted.

2. Click the **Extract** icon .

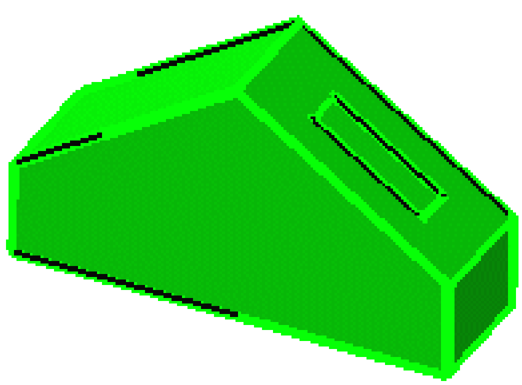
The Extract Definition dialog box is displayed.

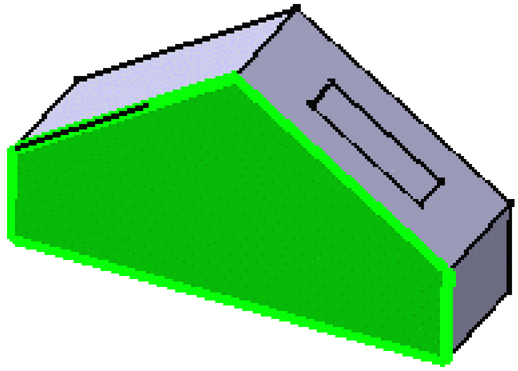


In Part Design workbench, the **Extract** capability is available as a contextual command named **Create Extract** that you can access from Sketch-based features dialog boxes.

3. Choose the **Propagation** type:

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



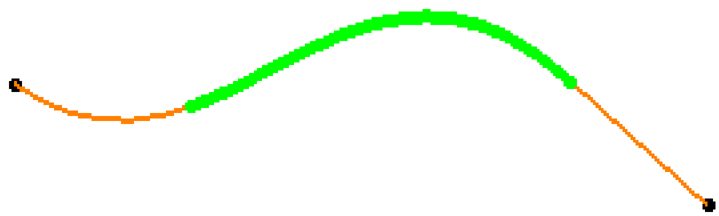


- **Tangent continuity**: the extracted element will be created according to tangency conditions.

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

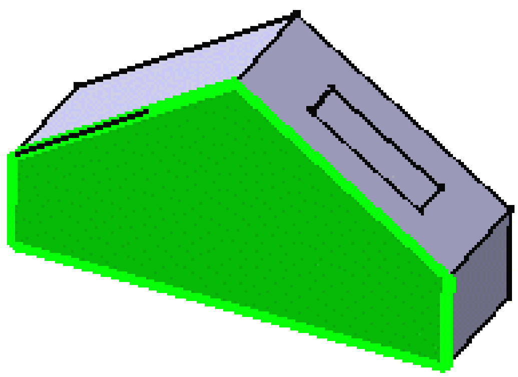
  Define a **Curvature Threshold** value. For a curvature discontinuity: the value is a ratio between 0 and 1 which is defined as follows:

  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.

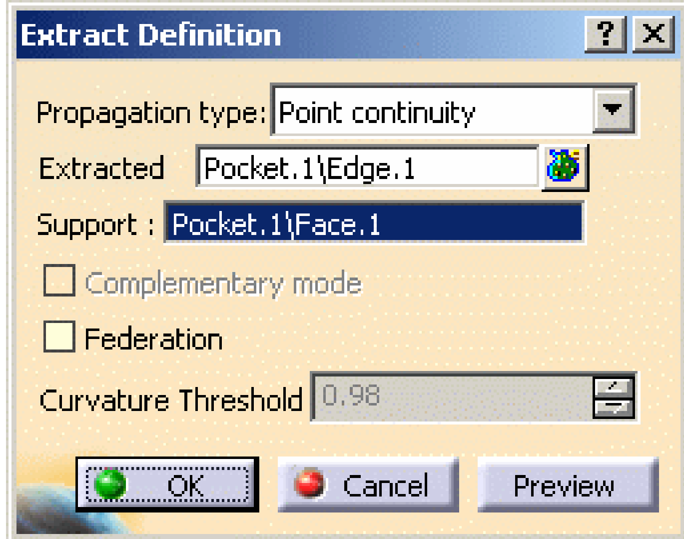
The extracted element must be a wire.



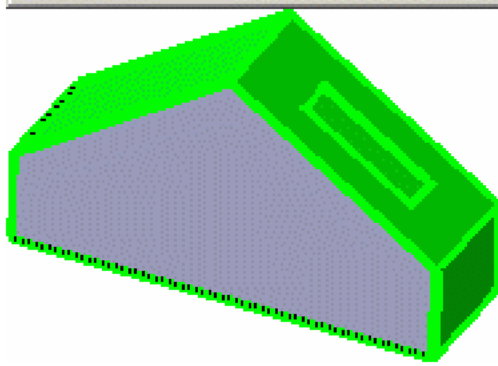
- **No propagation**: only the selected element will be created.

4. Click OK to extract the element.

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

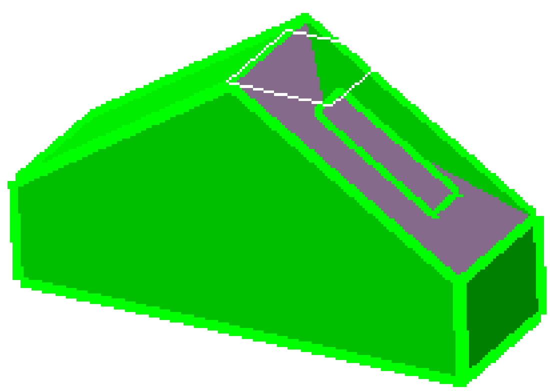


- If you extract an edge that you want to propagate, and there is an ambiguity about the propagation side, a warning is issued and you are prompted to select a support face. In this case, the dialog box dynamically updates and the **Support** field is added.



- 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.
- 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.

You can select a volume as the element to be extracted.
To do so, you can either:



1. select the volume in the specification tree, or
2. use the User Selection Filter toolbar and select the Volume Filter mode.
   For further information, refer to the Selecting Using A Filter chapter in the *CATIA Infrastructure User's Guide*.

   In both cases, the result of the extraction is the same whatever the chosen propagation type.

- In a .CATProduct document containing several parts, you can use the extract capability in the current part from the selection of an element in another part, provided the propagation type is set to No Propagation.
  In this case, a curve (respectively a surface or point) is created in the current part if the selected element is a curve (respectively a surface or point); the Extract parent therefore being the created curve (respectively the surface or point).
  Note:
  - if another propagation type is selected, the extraction is impossible and an error message is issued.
  - when editing the extract, you can change the propagation type as the parent belongs to the current part.
- 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.
- 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.
- 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.

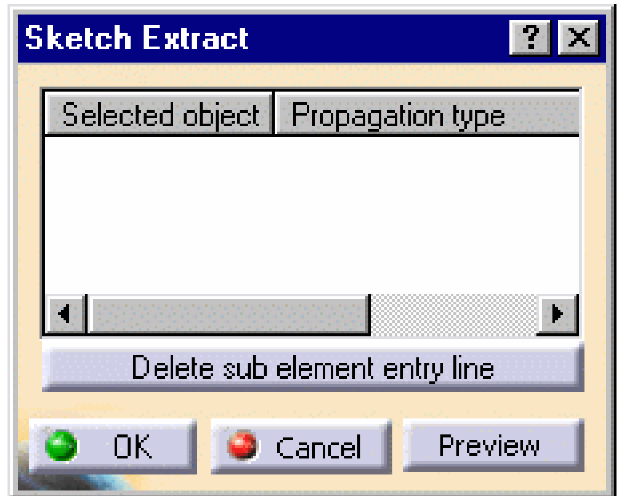
Extracting Multiple Edges

This task shows how to extract a subpart from a sketch to generate geometry from only one of the sketch's elements.

Open the Extract2.CATPart document.
It contains a sketch composed of several elements.

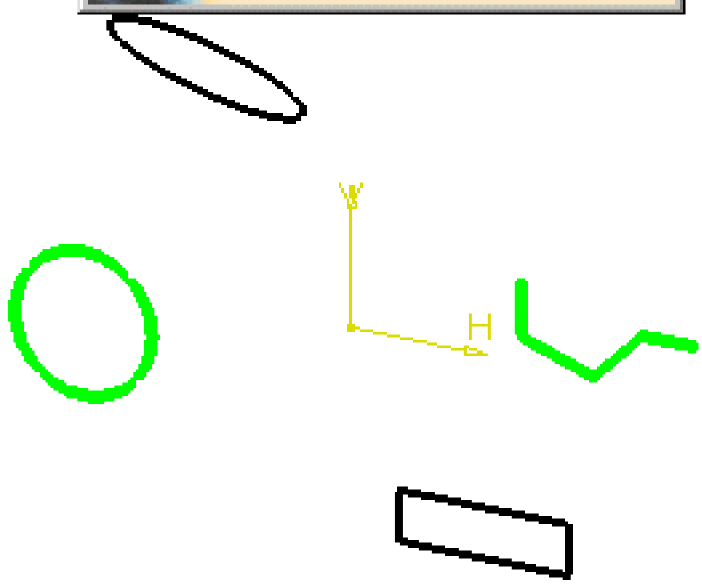
1. Click the **Multiple Edge Extract** icon  from the Extracts toolbar.

The Extract Definition dialog box is displayed.



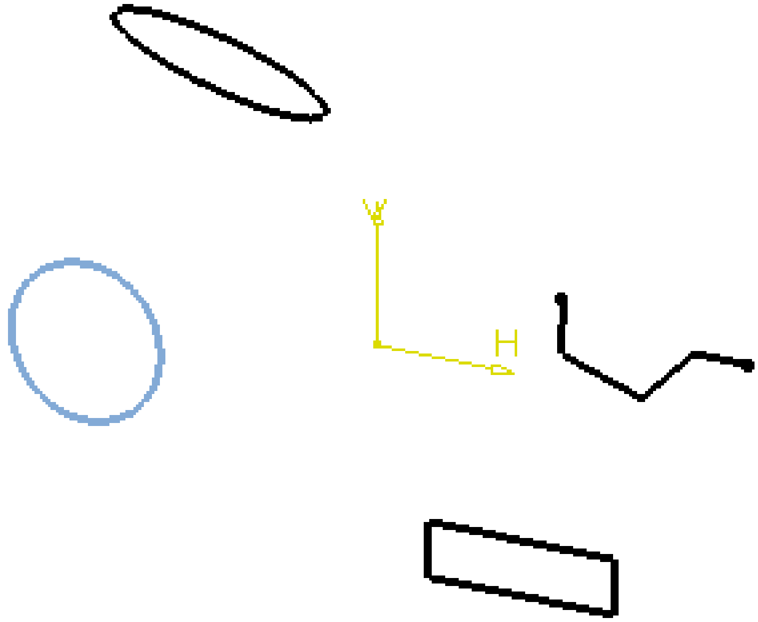
2. Select the element(s) you want to extract from the sketch.

The selected element(s) is highlighted.



To remove one of the selected element, select it from the list then click the **Delete sub-element entry line** button.

3. Click OK to extract the element.



The extracted element (identified as Extract.xxx) is added to the specification tree and can be used as any regular geometry.

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.

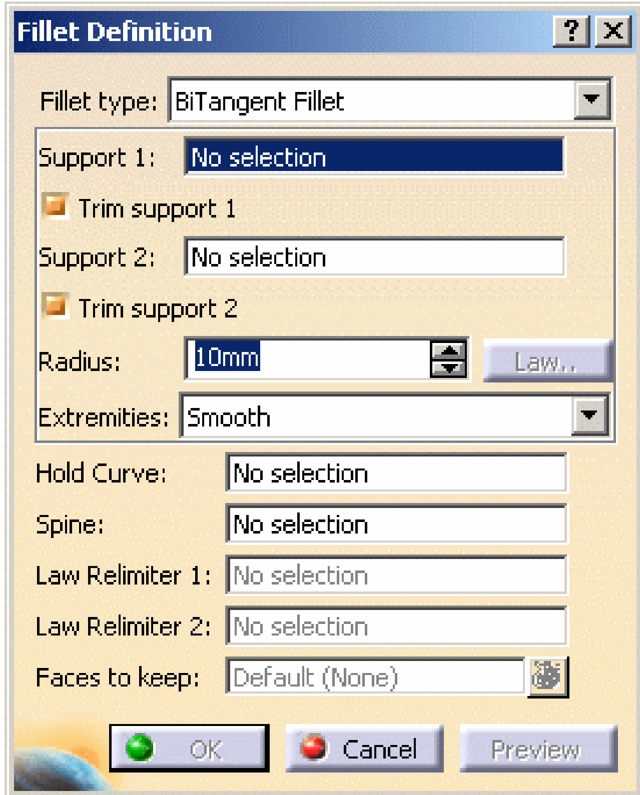
Creating Bitangent Shape Fillets

This task shows how to create a shape fillet between two surfaces.
The fillet surface is obtained by rolling a sphere between the selected surfaces.

Open the ShapeFillet1.CATPart document.

1. Click the **Shape Fillet** icon .

   The Fillet Definition dialog box appears.



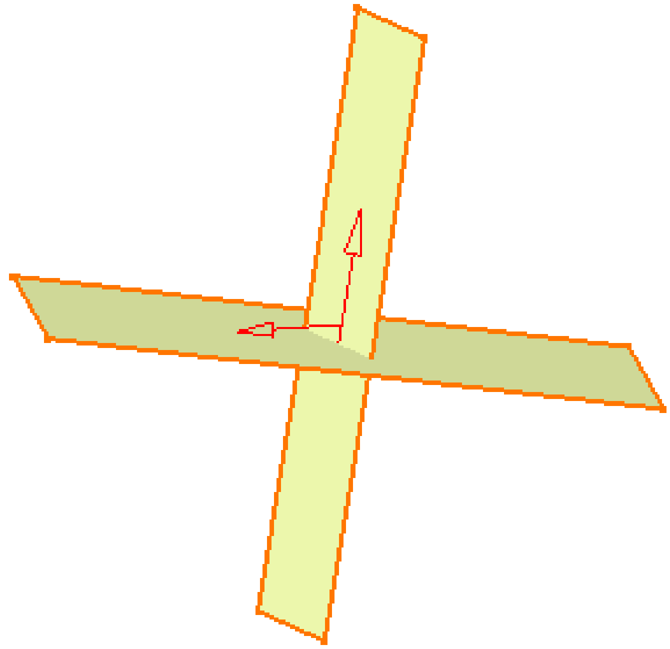
2. Choose the **BiTantent Fillet** ype.
3. Select a surface as the first support element.
4. Select another surface as the second support element.

Constant Radius Fillet

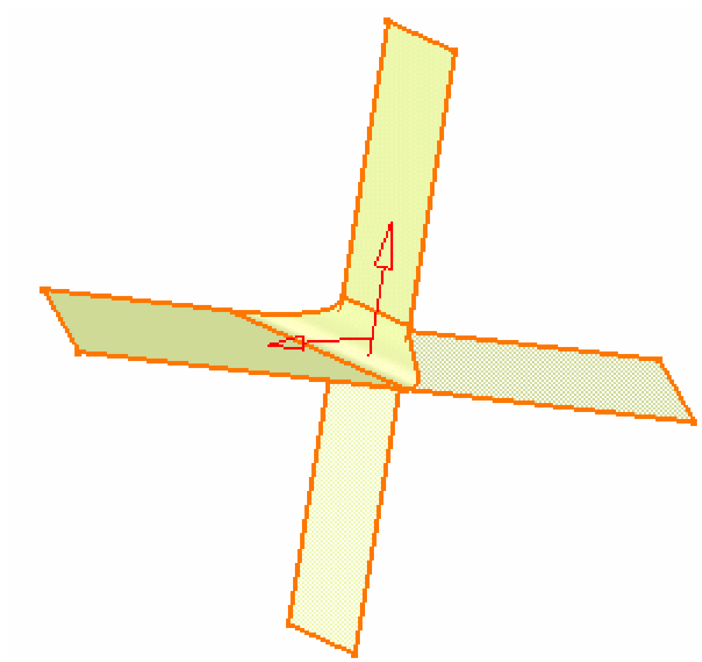
1. Enter the value of the fillet **Radius**.

Up to four fillet locations may be possible.
To help you decide on the location an arrow is displayed on each selected surface. You can click on the arrows to specify the

desired fillet location.



2. Click **Preview** to see the filleted surface.



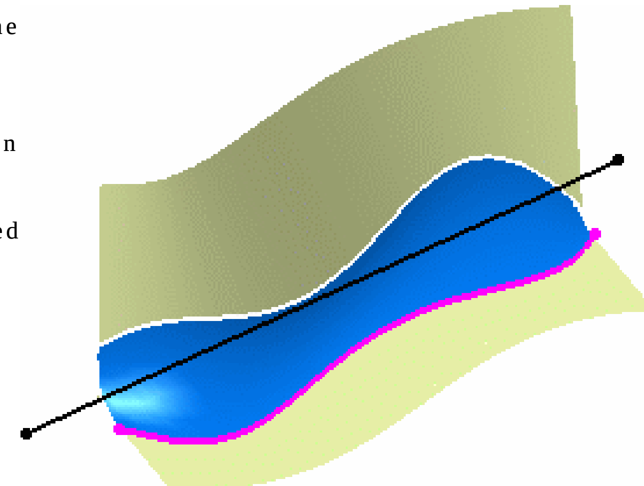
Variable Radius Fillet

With Spine and Hold Curve

Open the ShapeFillet3.CATPart document.

You can generate a variable bi-tangent radius fillet by selecting a **Hold curve**. In this case, you need to select a previously defined limiting curve that will control the fillet radius, and a previously defined spine that defines the planes in which the filleted surface section will pass. Both these curves must be larger than the surfaces involved, and the Hold curve must lie on one of these initial surfaces. The resulting filleted surface is tangent to the initially selected surfaces and limited by the hold curve.

- The **Radius** field is deactivated as the hold curve defines the variable radius.
- The Spine curve can be close or open.



The **Hold Curve** and **Spine** options are only available with the Generative Shape Design 2 product.

See also Creating a variable bi-tangent circle radius fillets using a spine.

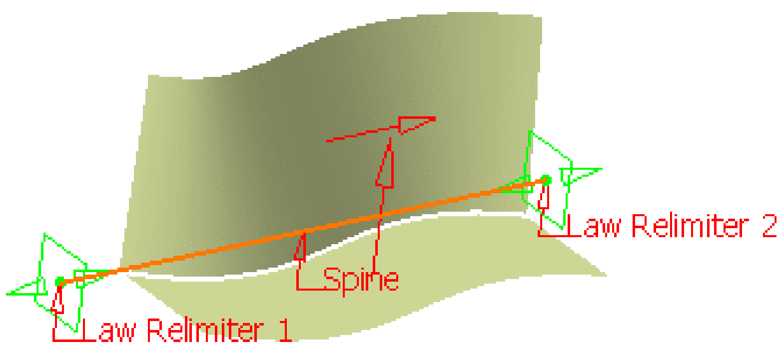
With Laws and Relimiters

The **Law** button becomes available

when the **Spine** field is filled in.

Open the ShapeFillet3.CATPart document.

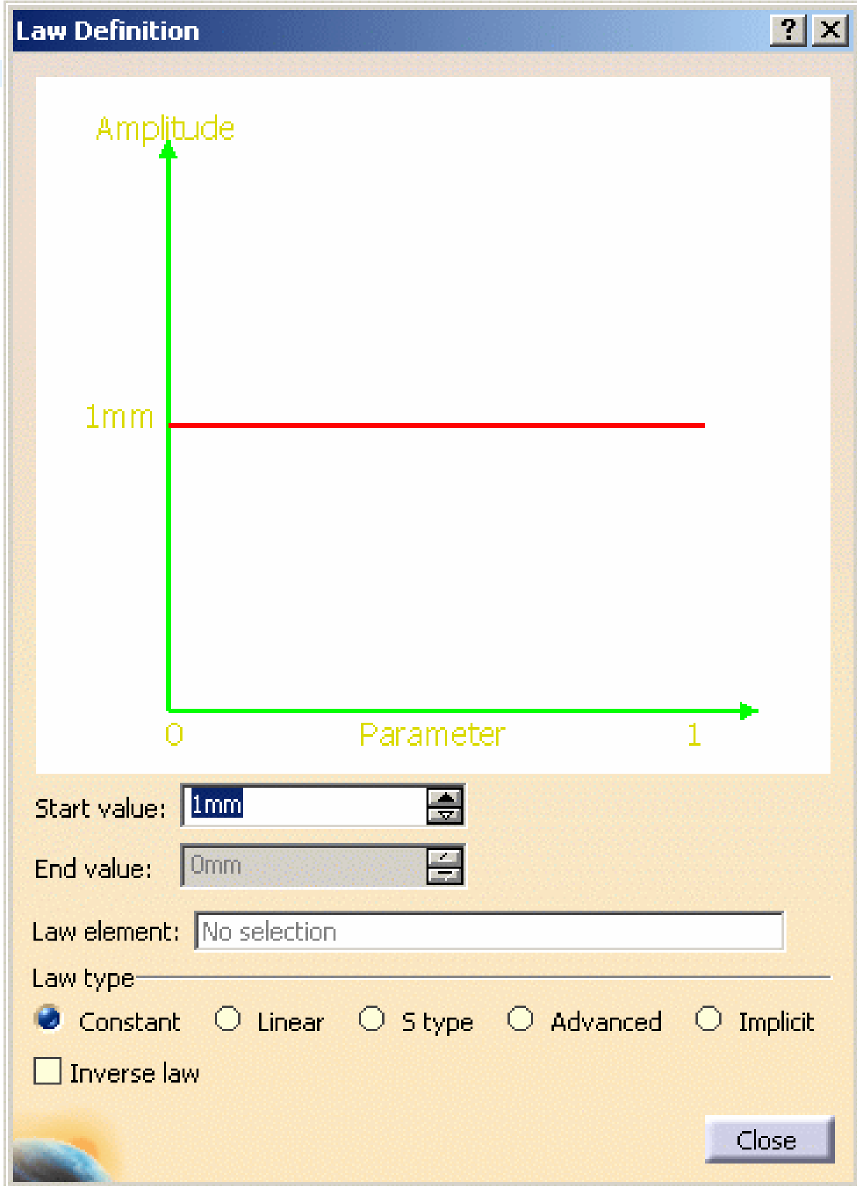
1. Select two surfaces.
2. Select Line.1 as the spine.
   Law relimiters are displayed on each extremity of the spine to delimit the radius law range. Manipulators enable you to move them along the spine.



You can use a close spine, in that case, only the **Law relimiter 1** field is enabled.

With Basic or Advanced Law

3. Click the **Law...** button to display the Law Definition dialog box. The 2D viewer enables you to previsualize the law evolution before applying it.



4. Enter **Start** and **End values**.
5. Choose the law type to be applied to the pitch.
   Four law types are available:

a. **Constant**: a regular law, only one value is needed.
b. **Linear**: a linear progression law between the **Start** and **End** indicated values
c. **S type**: an S-shaped law between the two indicated values. The pitch distance will vary between these two pitch values, over the specified number of revolutions.
d. **Advanced**: allowing to select a Law element as defined in Creating Laws.

The Law Viewer allows you to:

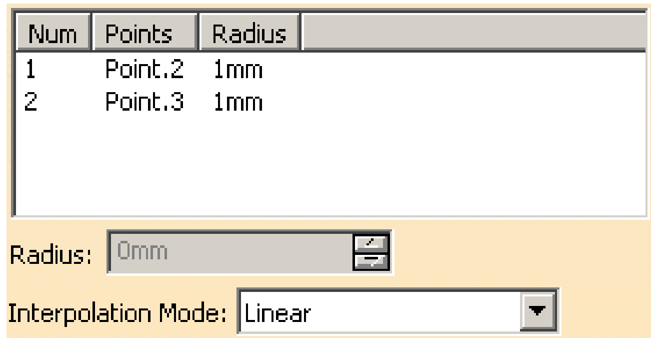
- visualize the law evolution and the maximum and minimum values,
- navigate into the viewer by panning and zooming (using to the mouse),
- trace the law coordinates by using the manipulator,
- change the viewer size by changing the panel size
- reframe on by using the viewer contextual menu
- change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

You can check the **Inverse law** button to reverse the law as defined using the above options.

With Implicit Law and Relimiters

e. **Implicit**: allowing the selection of points on the spine the association of values for these points.

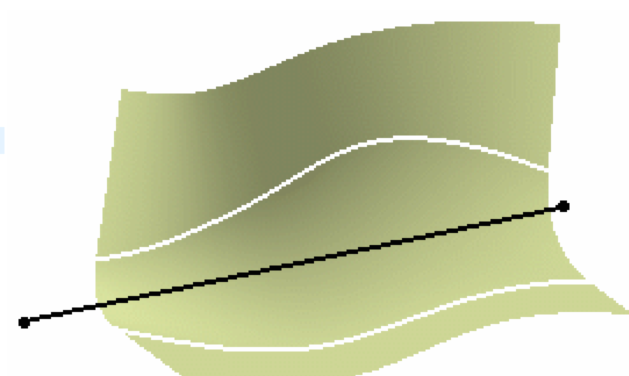
- Select points on the spine. By default, the spine extremities (relimiters) are selected.
- Define a radius for each point. For instance, set 30mm for Point.2 and 60 for Point.3. The radius law applies between these two points. The radius values are interpolated into a curvature continuous radius law defined over the whole spine curve.



- Create a third point between Point.2 and Point.3 with a radius of 90mm.
- Define the **Interpolation Mode**: Linear (straight interpolation) or Cubic (smooth interpolation). Here we chose the Linear mode.

Num	Points	Radius
1	Point.2	30mm
2	Point.4	90mm
3	Point.3	60mm

6. Click Close to return to the Fillet Definition dialog box.
7. Click Preview to see the filleted surface.



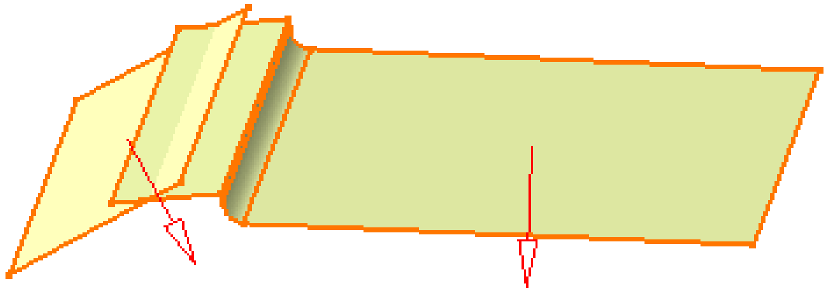
- The spine curve must must be curvature continuous.
- At least two points must be selected on the spine.
- If the spine is closed, only one point is selected.
- You can edit a point, by right-clicking it and choosing the Edit Point contextual item.
- You can add a point by right-clicking and choosing the Create Point contextual item.
- Points are automatically reordered in the list, according to their ratio on the spine.

BiTangent Fillet with Multi-ribbons

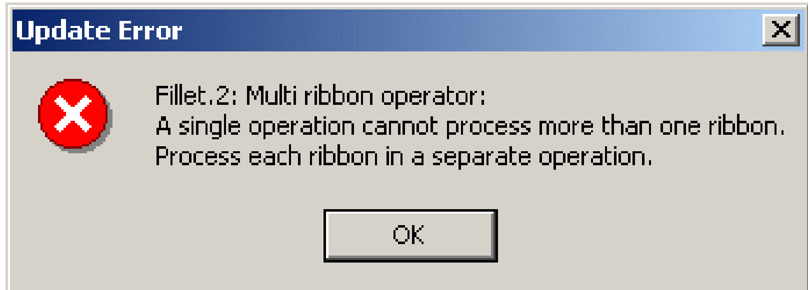
The **Faces to keep** option lets you manage multi-ribbons.

Open the ShapeFillet2.CATPart document.

1. Select two surfaces.
2. Set the Radius value as 20.
3. Click **Preview**.

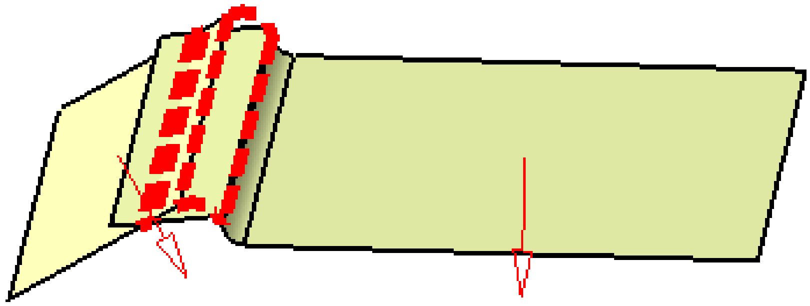


As you selected two ribbons, an error message pops up asking you to process each ribbon in a separate operation.

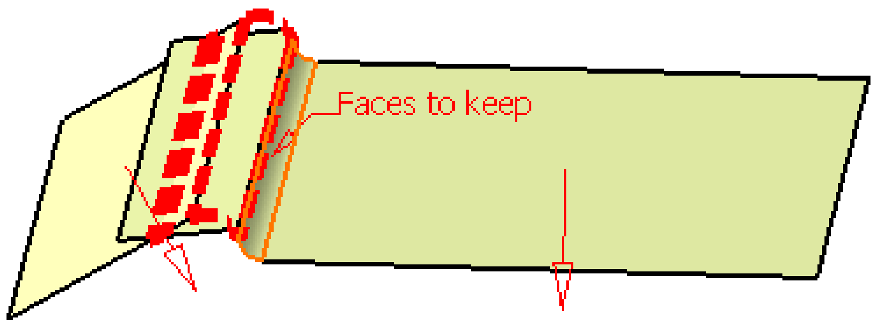


4. Click **OK** in the dialog box.

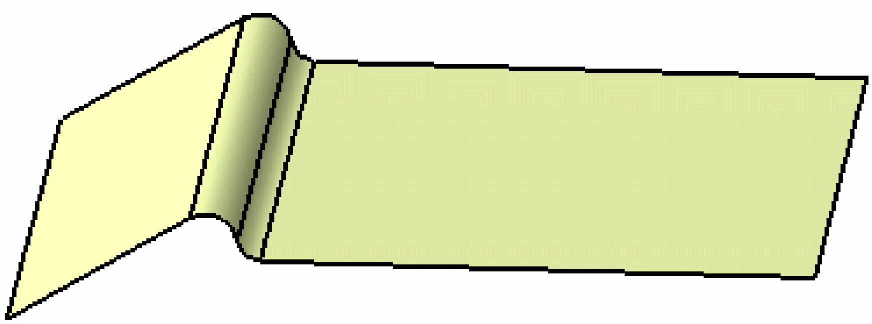
The solutions are shown in red.



5. In the **Faces to keep** field, select the face(s) you want to keep to create the shape fillet.

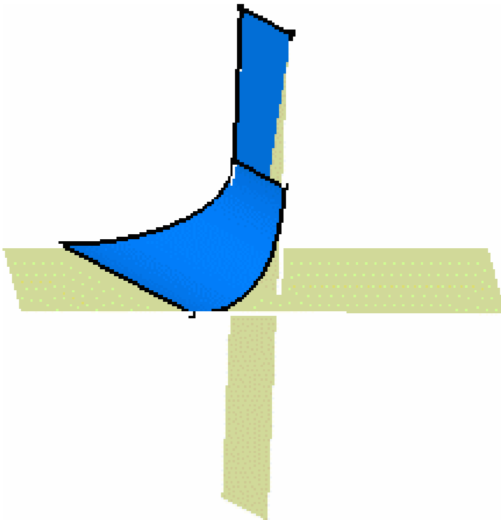


6. Click Preview to see the filleted surface.

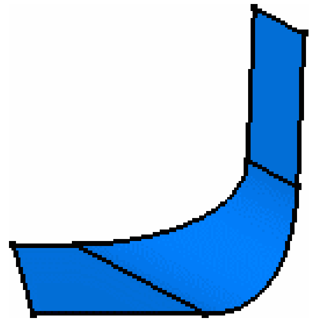


The Multi-Output capability is available.

You can uncheck any of the **Trim support** check buttons. In this case, the support element involved will not be trimmed and assembled to the filleted surface.
By default, both trimming check buttons are checked, thus relimiting both support elements.
In the examples below, we changed the filleted surface's color to better visualize it.



With only one support element trimmed

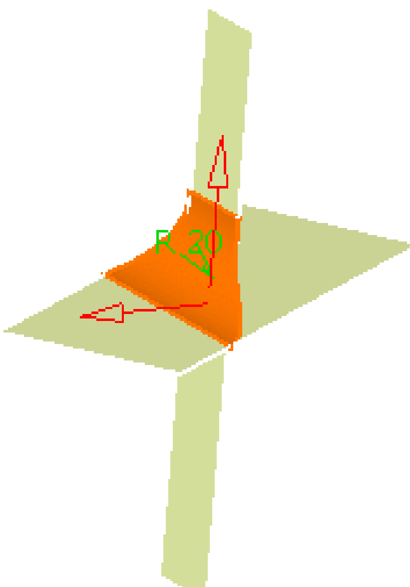


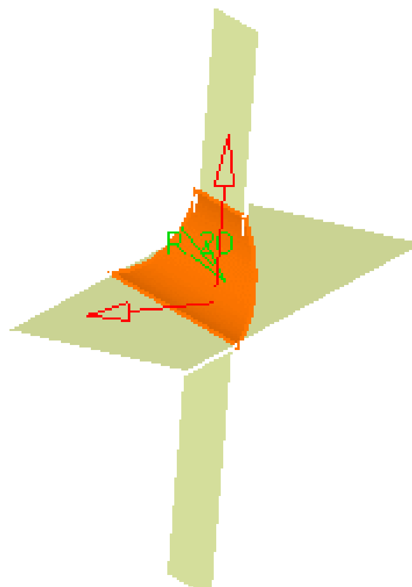
With both support elements trimmed

The **Trim** option is only available with the Generative Shape Design 2 product.

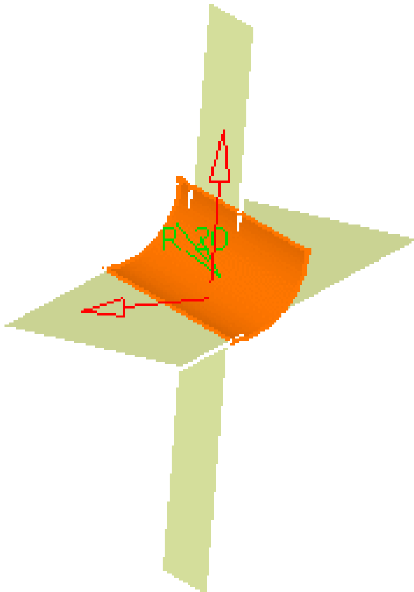
5. Use the combo to choose the desired type of extremity for the fillet:

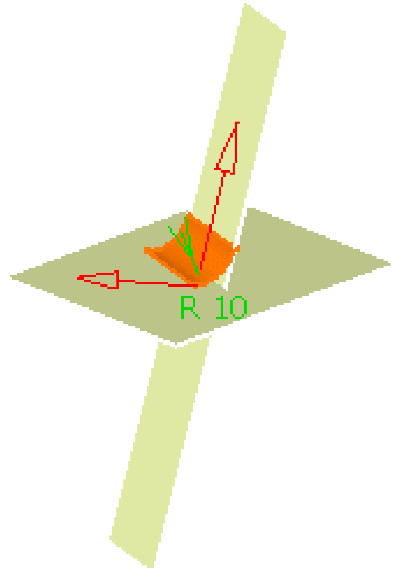
- **Smooth:** a tangency constraint is imposed at the connection between the fillet surface and the support surfaces, thus smoothing the connection
- **Straight:** no tangency constraint is imposed at the connecting point between the fillet and the initial supports, generating sometimes a sharp angle.





- **Maximum:** the fillet surface is limited by the longest selected support's edge
- **Minimum**: the fillet surface is limited by the shortest selected support's edge.





6. Click **OK** to create the shape fillet.

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

- In case the selected supports are partially tangent, it is advisable to create an edge fillet.
- Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.
- Stacking commands is available.

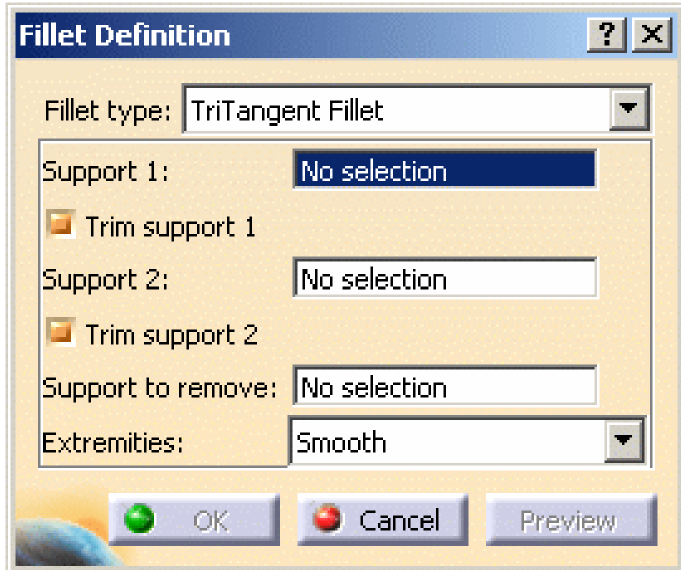
Creating Tritangent Shape Fillets

This task shows how to create a shape fillet between three surfaces.
The fillet surface is obtained by rolling a sphere between the selected surfaces. It involves the removal of one of the three faces selected.

Open the ShapeFillet4.CATPart document.

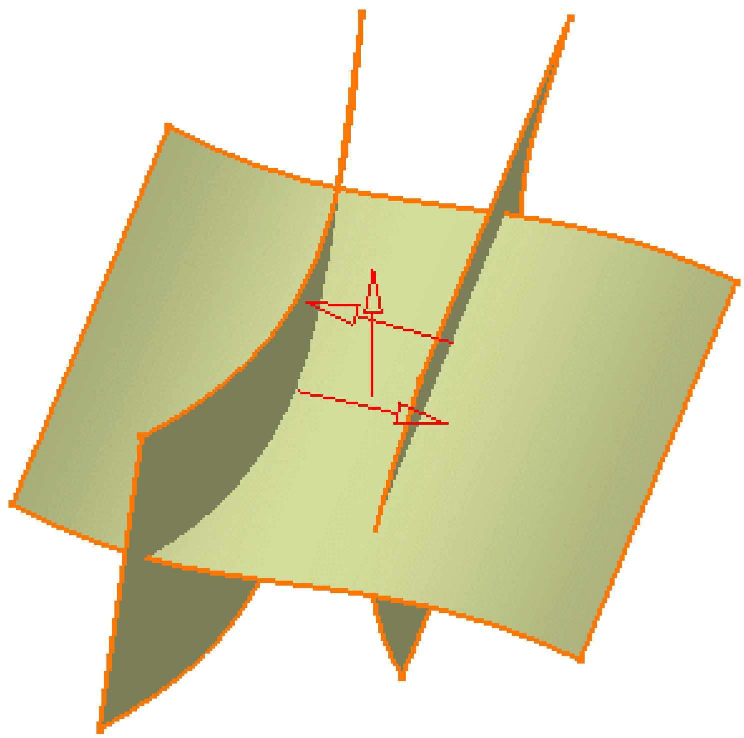
1. Click the **Shape Fillet** icon .

   The Fillet Definition dialog box appears.



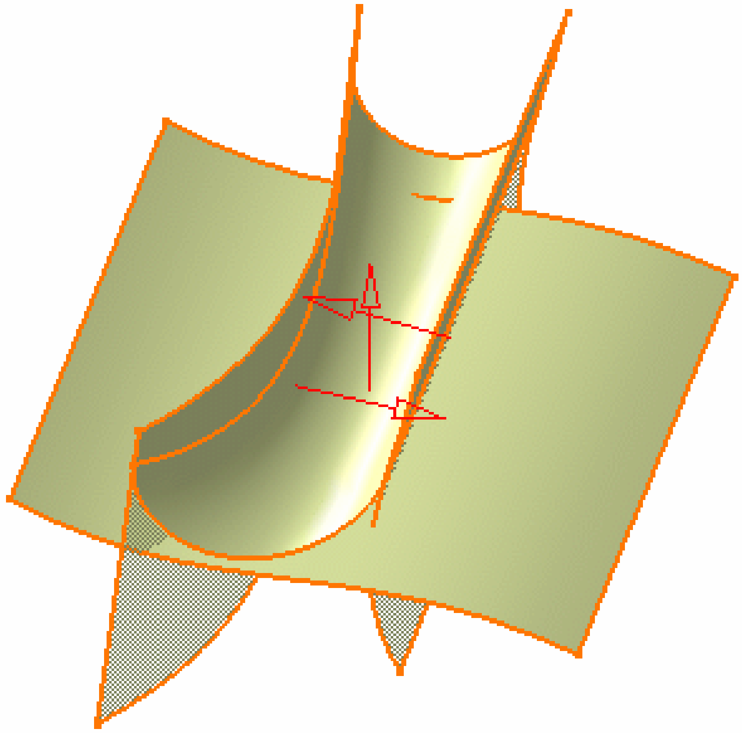
2. Choose the **Tritangent Fillet** Type.
3. Select a surface as the first support element.
4. Select another surface as the second support element.
5. Select the surface to be removed.

   The fillet will be tangent to this face.

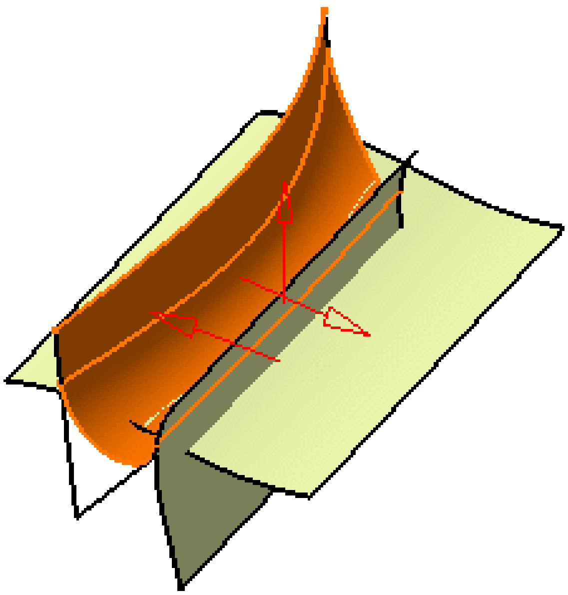


Up to six fillet locations may be possible.
To help you decide on the location an arrow is displayed on each selected surface. You can click on the arrows to specify the desired fillet location.

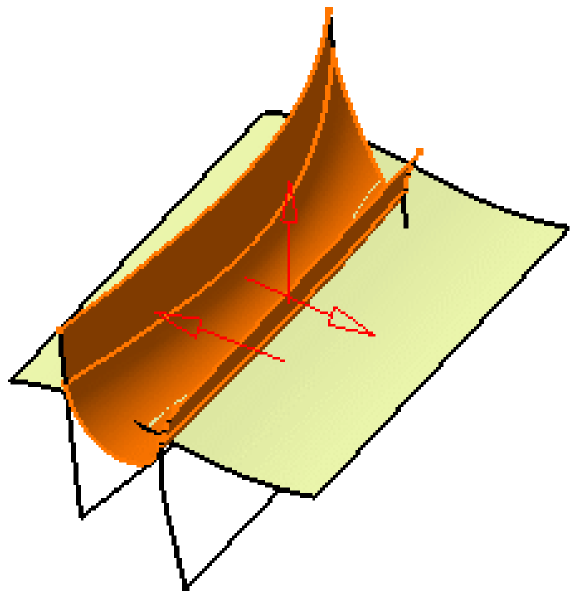
6. Click Preview to see the filleted surface.



You can uncheck any of the **Trim support** check buttons. In this case, the support element involved will not be trimmed and assembled to the filleted surface.
By default, both trimming check buttons are checked, thus relimiting both support elements.



With only one support element trimmed

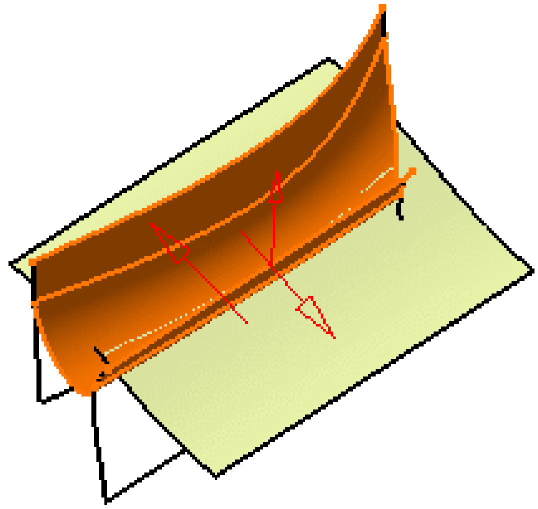


With both support elements trimmed

The **Trim** option is only available with the Generative Shape Design 2 product.

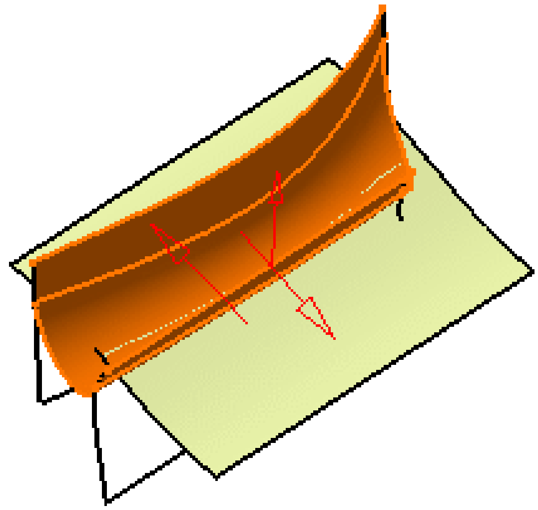
7. Use the combo to choose the desired type of extremity for the fillet:

- **Smooth:** a tangency constraint is imposed at the connection between the fillet surface and the support surfaces, thus smoothing the connection



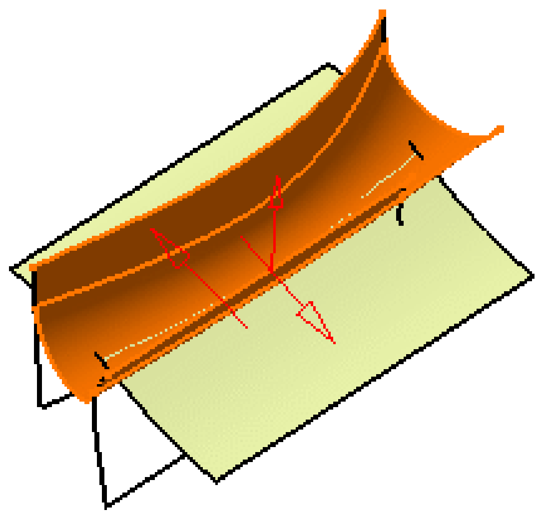
Smooth fillet

- **Straight:** no tangency constraint is imposed at the connecting point between the fillet and the initial supports, generating sometimes a sharp angle.



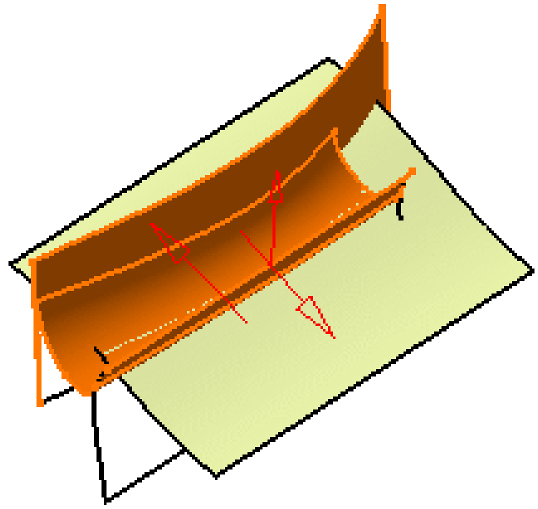
Straight fillet

- **Maximum:** the fillet surface is limited by the longest selected support's edge



Maximum fillet

- **Minimum**: the fillet surface is limited by the shortest selected support's edge.



Minimum fillet

8. Click **OK** to create the shape fillet.

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

- In case the selected supports are partially tangent, it is advisable to create an edge fillet.
- Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.
- Stacking commands is available.

Creating Edge Fillets

Edge fillets are useful to provide a transitional surface along a sharp internal edge of a surface.

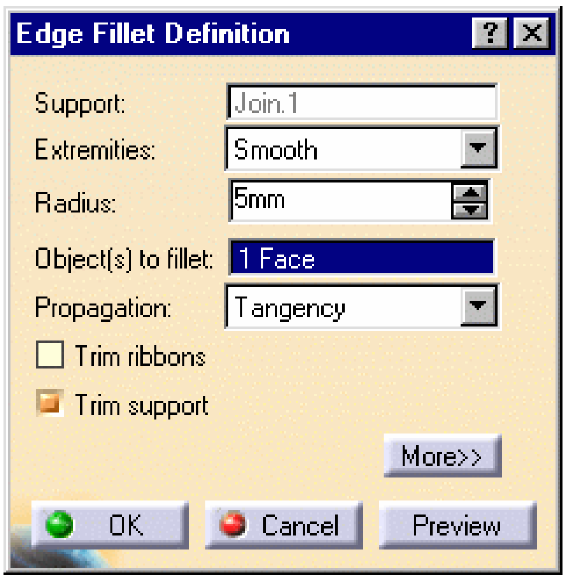
This task shows how to create a constant radius fillet along the internal edge of a joined surface.
The fillet surface is obtained by rolling a sphere over the selected edge.

- Keeping Edges
- Limiting Fillets
- Ignoring Edges
- Trimming Overlapping Fillets
- Filleting Volumes

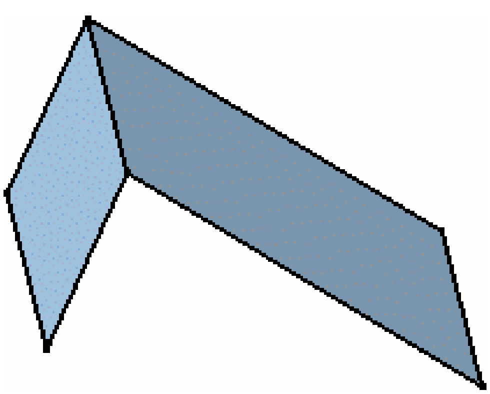
Open the EdgeFillet1.CATPart document.

1. Click the **Edge Fillet** icon.

   The Edge Fillet Definition dialog box appears.



2. Select the edge to be filleted.



You can also select a face, provided there is no ambiguity as to the edge(s) to be filleted.

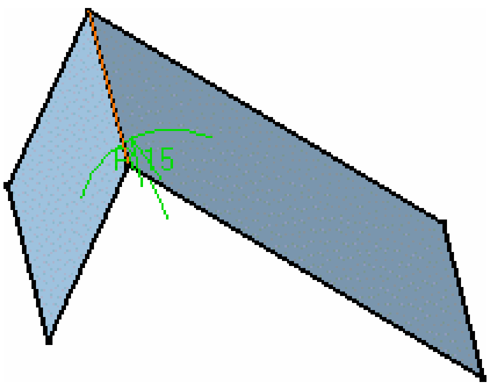
3. Use the combo to select the desired type of extremity for the fillet:

- **Straight:** no tangency constraint is imposed at the connecting point between the fillet and the initial support, generating sometimes a sharp angle
- **Smooth:** a tangency constraint is imposed at the connection between the fillet surface and the support surfaces, thus smoothing the connection
- **Maximum:** the fillet surface is limited by the longest selected edge
- **Minimum**: the fillet surface is limited by the shortest selected edge

(Refer to Shape Fillets)

4. Enter the value of the fillet **Radius**.

A preview of the fillet appears.



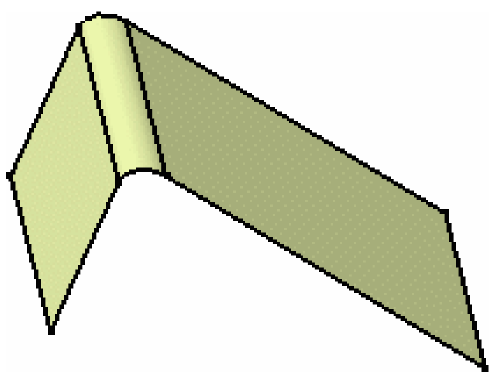
5. You can choose the **Propagation** type:

- **Tangency**: the fillet is propagated up to the first edge that is not continuous in tangency.
- **Minimal**: the fillet is propagated up to the first geometric limitation.

Use the **More>>** button to access further options: Edge(s) to keep, Limiting element, and Blend corner.

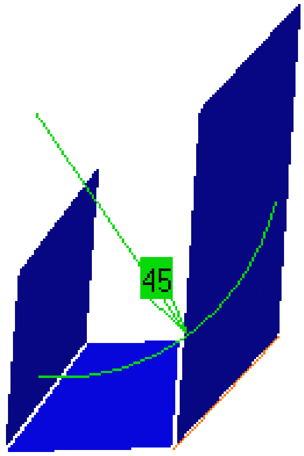
6. Click **OK** to create the fillet surface.

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



Check the **Trim support elements** option to relimit the support elements and assemble them to the fillet.

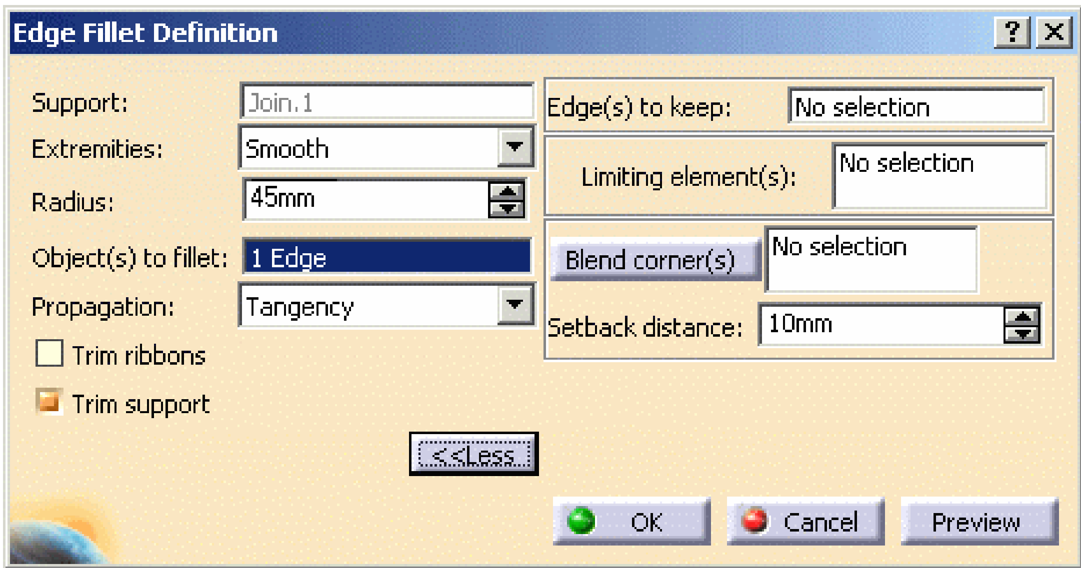
Keeping Edges



You may also need to explicitly indicate edges that should not be filleted, if a radius is too large for example. In this case you cannot select boundary edges to be kept, but only internal edges, i.e. edges limiting two faces.

Open the EdgeFillet2.CATPart document.

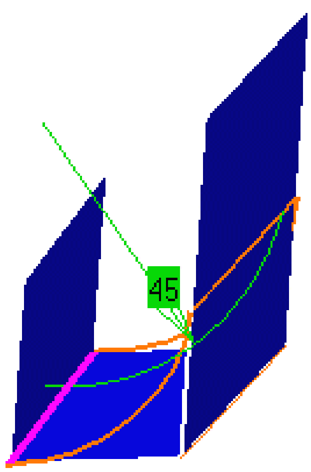
To do this, proceed as above, but once you have selected the edge to be filleted, click the **More** to expand the dialog box, then click the Edge(s) to keep field and select the edge you wish to keep.



If you have difficulties selecting the edge, use the up/down arrows to display the preselection navigator.

CATIA displays this edge in pink, meaning that it will not be affected by the fillet operation.

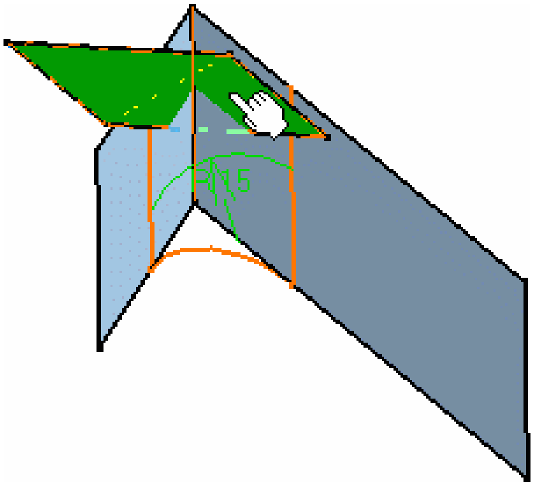
Then, click **OK** to create the fillet surface.



Limiting Fillets

While creating the fillet, you can limit it by selecting a surface that intersects it completely:

1. Once the edge to be filleted has been selected, and the radius keyed in, click Preview then the More>> button.

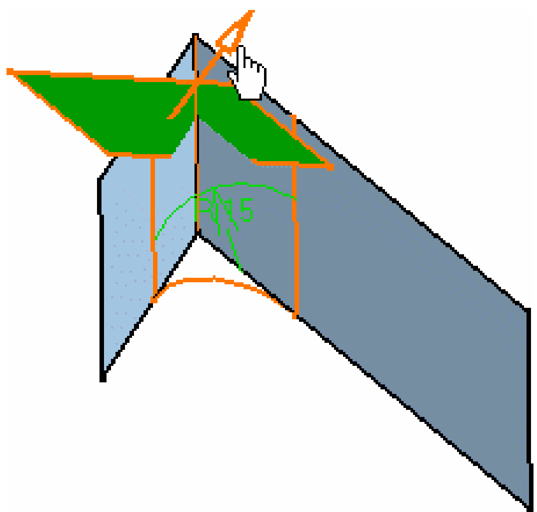


2. Click in the **Limiting element(s)** field, then select the trimming element(s). These elements can either be surfaces, planes, or points on edges.

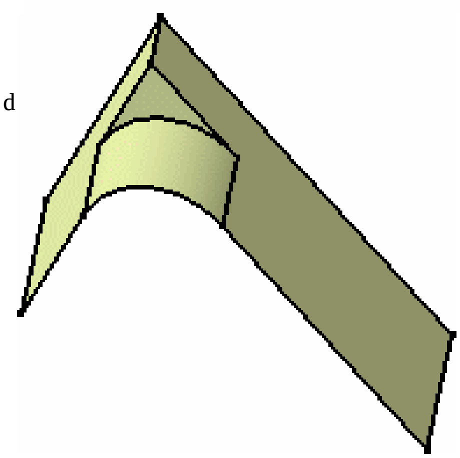
An arrow indicates which portion of the fillet is to be retained.

- You can use or more limiting elements.
- You can define a limiting element just by clicking a point on one of the selected edges to be filleted.

3. Click on this arrow to inverse it, if needed, to retain the opposite side of the fillet.



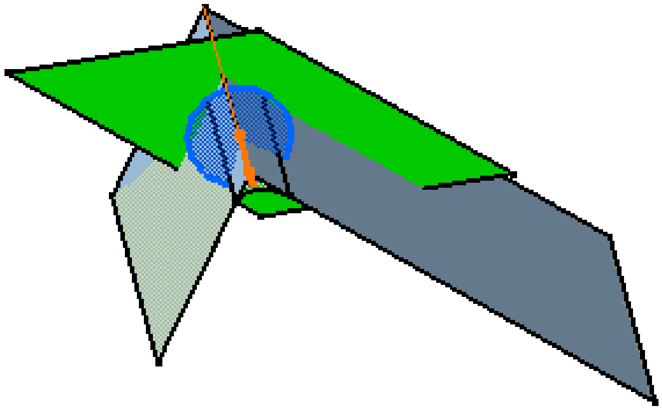
4. Click OK to create the limited fillet.



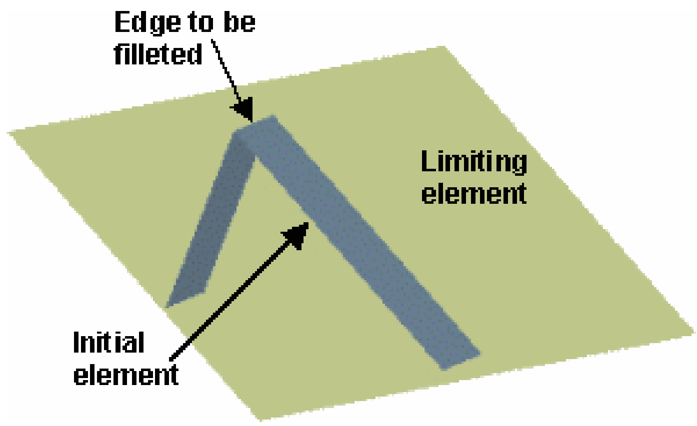
In the illustration, the limiting surface has been hidden.

You can create limiting elements just by clicking on the edge to be filleted.

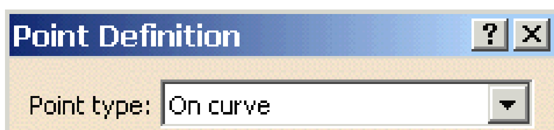
The application displays this element as a blue disk.



Make sure that the limiting element is not larger than the initial element, as illustrated here. In this case, decrease the size of the limiting element as prompted by the warning message.

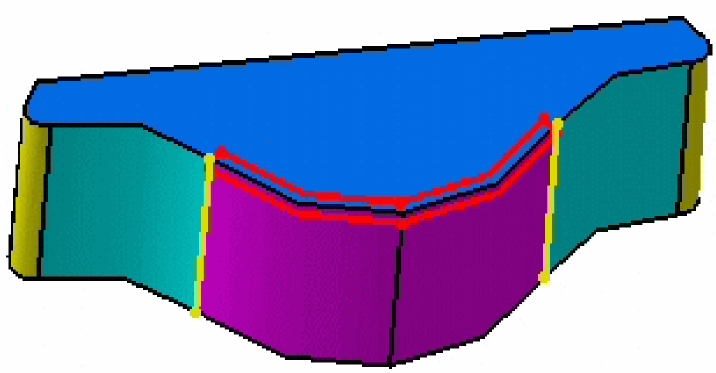


You can select planes or points as limiting elements.
Points must be located on the edge to be filleted and they must have been created using the **On curve** option available in the Point Definition dialog box.

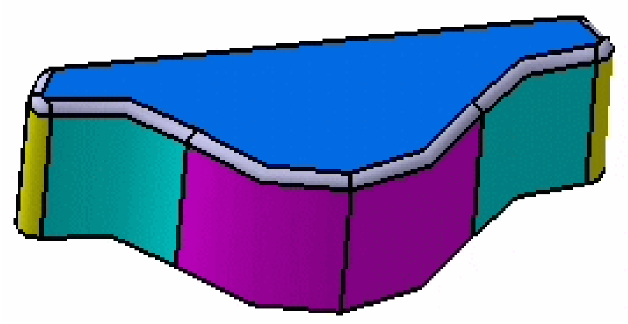


Ignoring Edges

When the update process detects that sharp edges (edges are considered as sharp when the angle between the two faces is greater than 0.5 deg) interrupt fillet operations, it is possible to continue filleting just by selecting an edge adjacent to the edge to be filleted. In the example below, the application displays the edge causing trouble in yellow:



An error message is issued, prompting you to select an edge adjacent to the filleted edge. Just by selecting the edge to the right of the previewed fillet, the application can then compute the whole fillet properly:

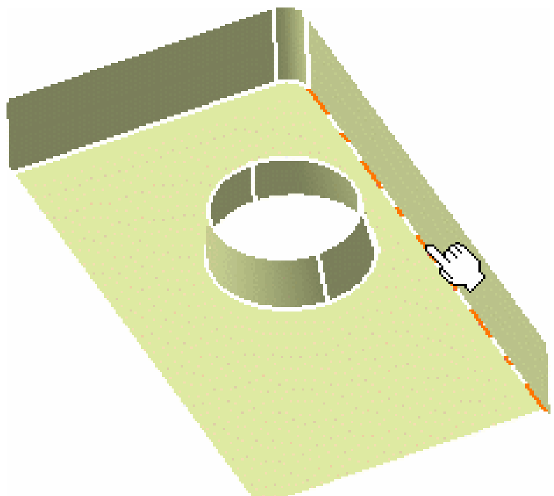


Trimming Overlapping Fillets

In some cases, fillets may be overlapping. The **Trim ribbons** option lets you solve this by trimming the fillets where they overlapping.

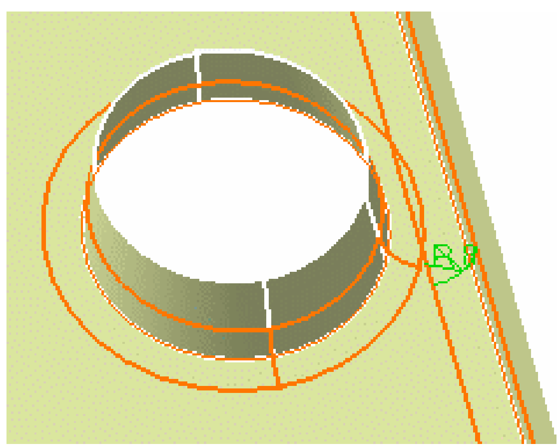
Open the EdgeFillet3.CATPart document.

1. Click the **Edge Fillet** icon and, using the Ctrl key, select the edges at the base of the cylinder and the one along the vertical surface.

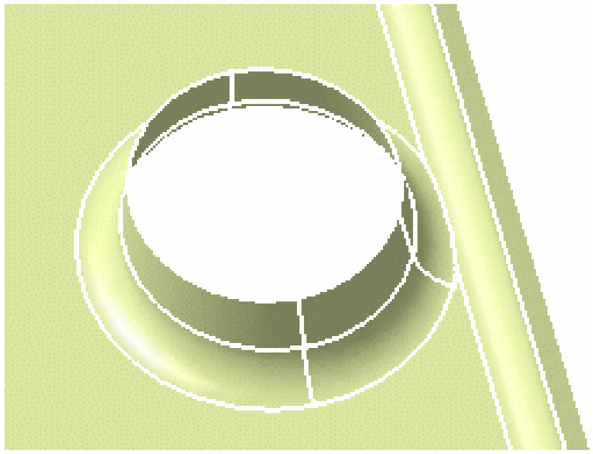


2. Click Preview.

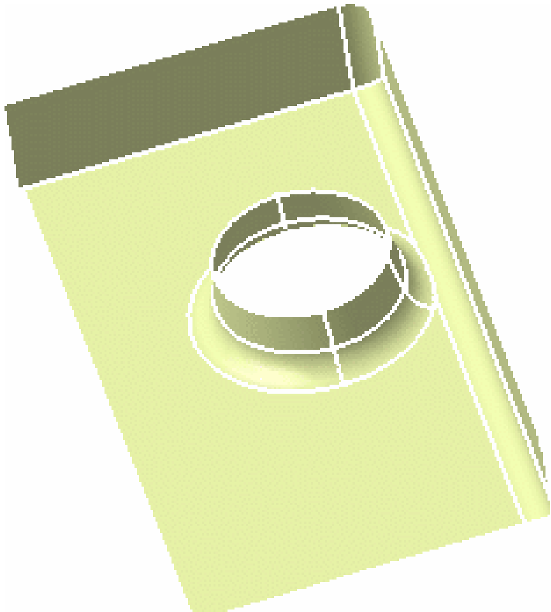
The two fillets clearly overlap.



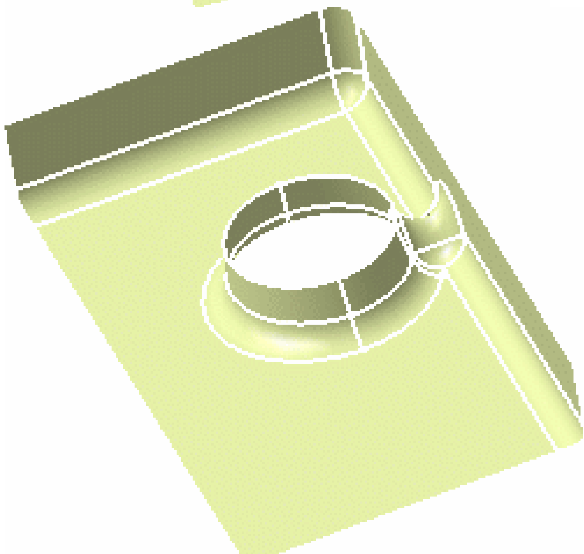
3. In the Edge Fillet Definition dialog box, check the **Trim ribbons** option and click OK.



Note that the **Trim ribbons** option is available with the **Tangency** propagation mode:

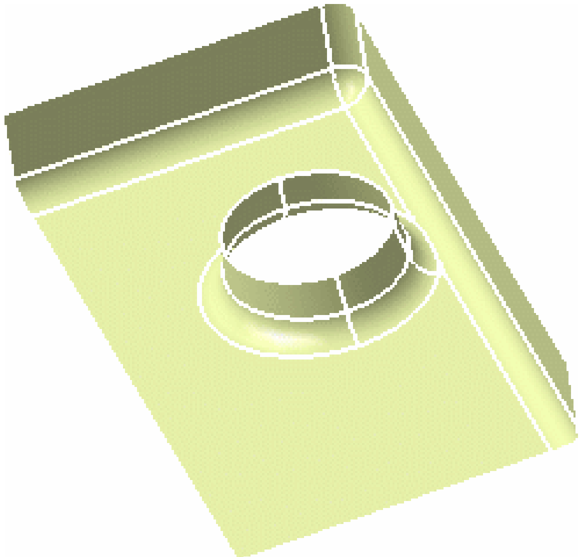


- In **Minimal** mode, the **Trim ribbons** option is grayed, as it is implicitly active. The results would be trimmed fillets, and no propagation.



- In **Tangency** mode, with the **Trim ribbons** option unchecked, the fillets intersect, with no trimming, and the propagation is performed.

- In **Tangency** mode, with the **Trim ribbons** option checked, the fillets are trimmed and the propagation is performed.

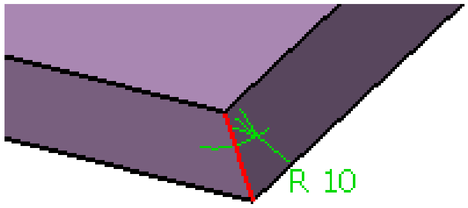


Filleting Volumes

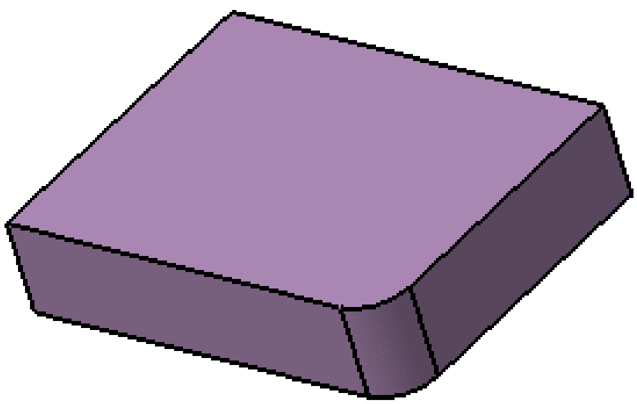
Open the FilletingVolumes1.CATPart document.

1. Click the **Edge Fillet** icon .

   The Edge Fillet Definition dialog box appears.



2. Select the edge to be filleted.
3. Set the **Radius** to 10mm.
4. Click OK to create the fillet volume.



Extremities and **Trim support** options are grayed out, they cannot be used with volumes.

Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting Using A Filter chapter in the *CATIA Infrastructure User's Guide*.

Creating Variable Radius Fillets

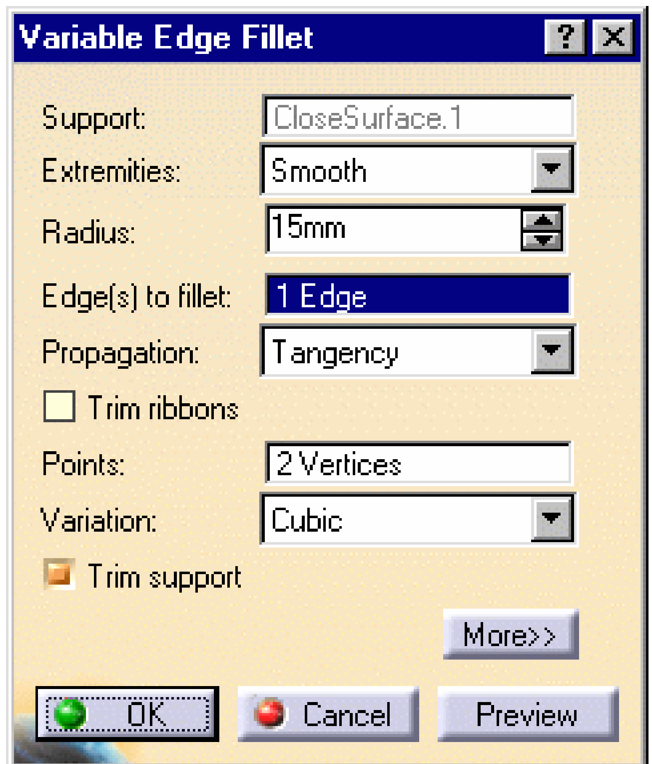
This task shows how to create a variable radius fillet. In this type of fillet, the radius varies at selected points along a selected edge.
The fillet surface is obtained by rolling a sphere, which radius would vary, over the selected edge.

- Limiting Fillets
- Trimming Overlapping Fillets
- Filleting Volumes

Open the FilletVariableRadius1.CATPart document.

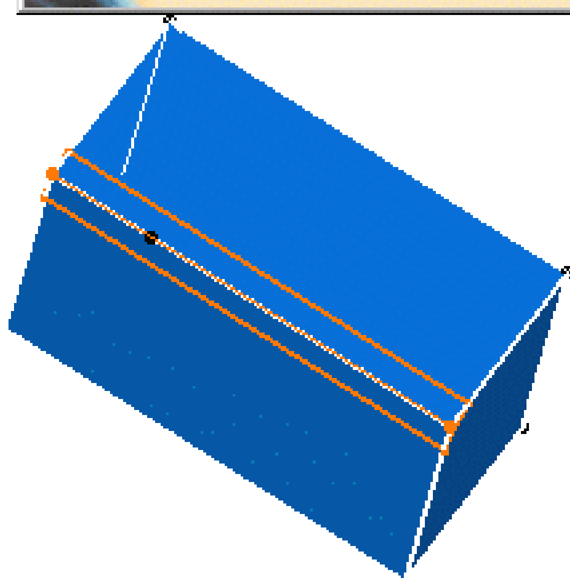
1. Click the **Variable Radius Fillet** icon .

   The Variable Edge Fillet dialog box appears.



2. Select the edge to be filleted and click Preview.

   The system detects the two vertices and displays the default radius value.



3. Use the combo to select the desired type of extremity for the fillet:

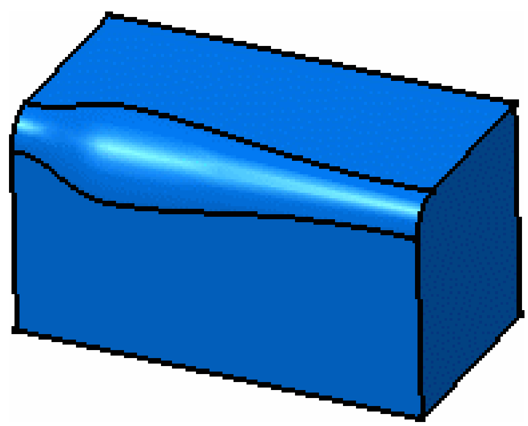
- **Straight:** no tangency constraint is imposed at the connecting point between the fillet and the initial support, generating sometimes a sharp angle.
- **Smooth:** a tangency constraint is imposed at the connection between the fillet surface and the support surfaces, thus smoothing the connection
- **Maximum:** the fillet surface is limited by the longest selected edge
- **Minimum**: the fillet surface is limited by the shortest selected edge

(Refer to Shape Fillets)

4. You can also choose the propagation type:

- **Tangency**: the fillet is propagated up to the first edge that is not continuous in tangency.
- **Minimal**: the fillet is propagated up to the first geometric limitation.

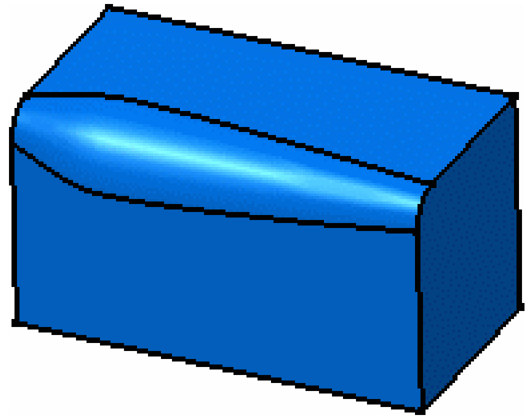
5. To add an additional point on the edge to make the variable radius fillet, click the **Points** field and select a point on the edge.
6. Enter a new **Radius** value for this point.
7. Set the **Propagation** mode to **Cubic** to obtain a smooth transition from one radius to another.



8. Click **OK** to confirm the operation.

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

This is the fillet you would obtain using the **Linear** propagation mode. In this case there is a straight transition from one radius to another.

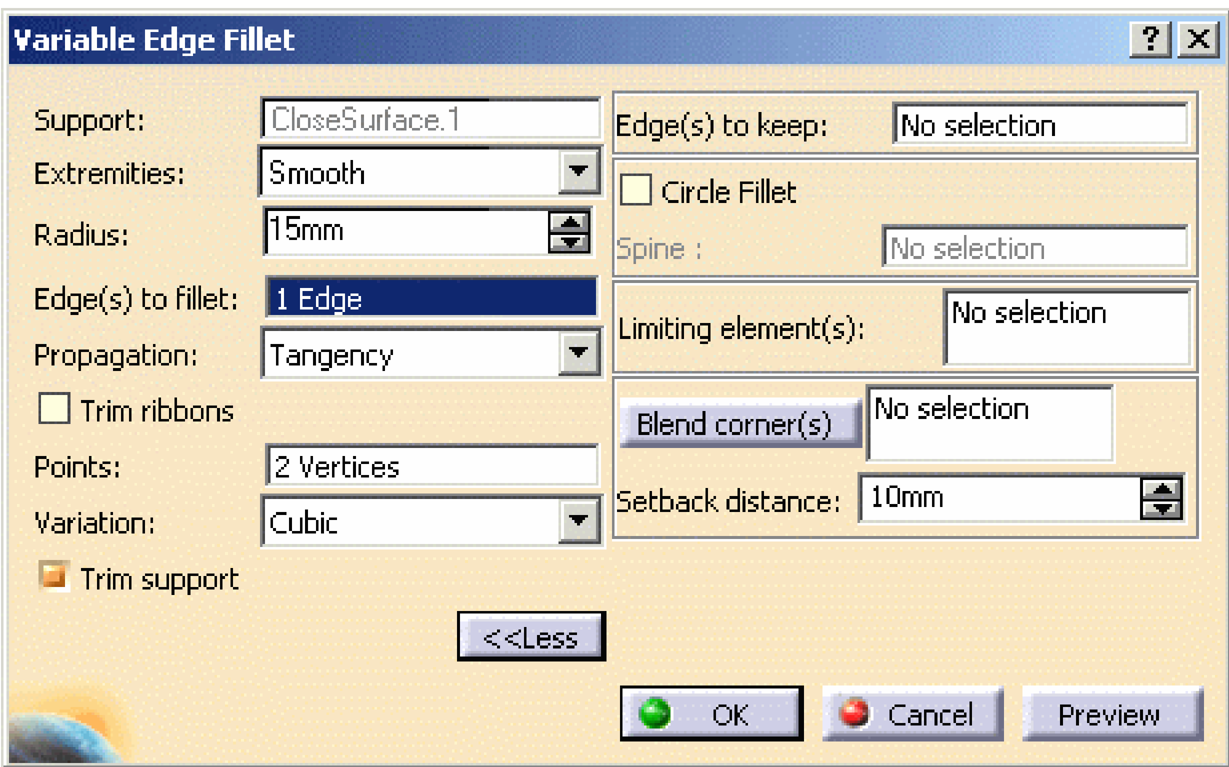


Limiting Fillets

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

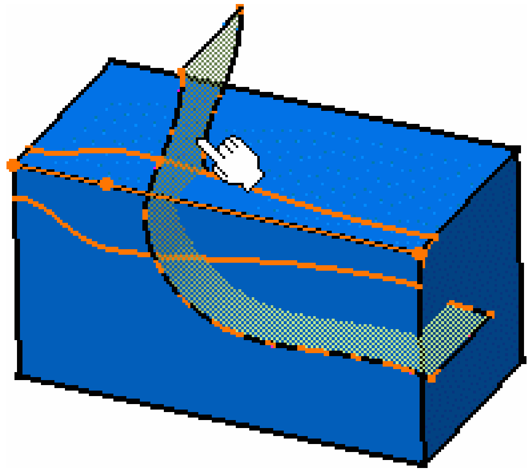
While creating the fillet, you can limit it by selecting an element (plane or surface) that intersects it completely:

1. Once the edge to be filleted has been selected, and the radius keyed in, click **Preview** then the **More>>** button.



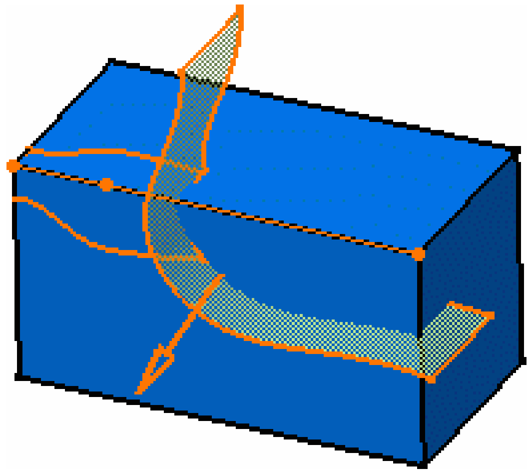
2. Click in the **Limiting element(s)** field, then select the trimming element(s). Here we chose Extrude.7.

These elements can either be surfaces, planes, or points on edges.



An arrow indicates which portion of the fillet is to be retained.

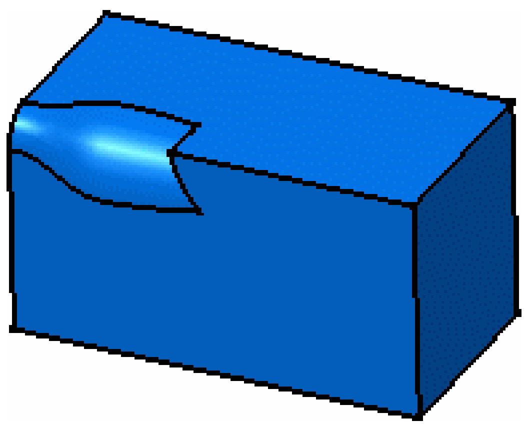
If needed, you can click on this arrow to inverse it, to retain the opposite side of the fillet.



- It is possible to use one or more limiting elements.
- You can define a limiting element just by clicking a point on one of the selected edges to be filleted.

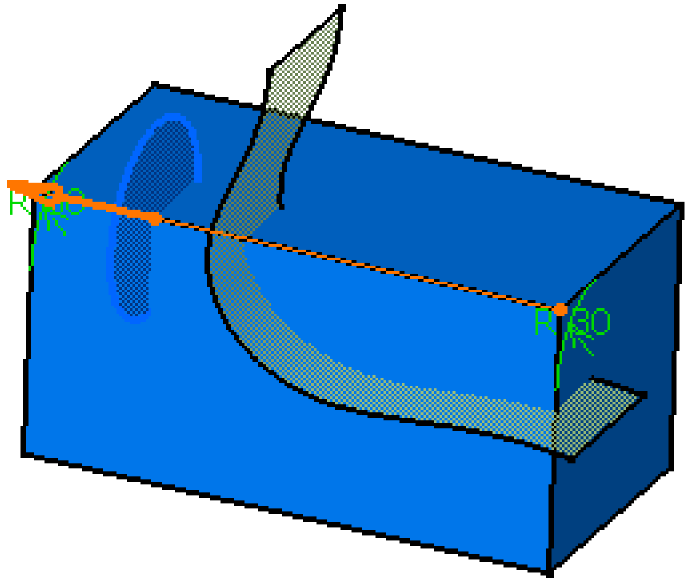
3. Click OK to create the limited fillet.

In the illustration, the limiting surface has been hidden.



- You can create limiting elements just by clicking on the edge to be filleted.

The application displays this element as a blue disk.



- You can select points as limiting elements. These points must be located on the edge to be filleted and they must have been created using the **On Curve** option available in the Point Definition dialog box.

- You can also define variable radius fillets on closed edges.

However, the application defines a default vertex on closed edges when applying the Edge Fillet command. To define the fillet, you need to remove this vertex first of all, then use 3D points only.

Note that the **Linear** variation mode is not valid for closed edges or closed sets of edges that are continuous in tangency. In these cases, the **Cubic** mode is automatically applied.

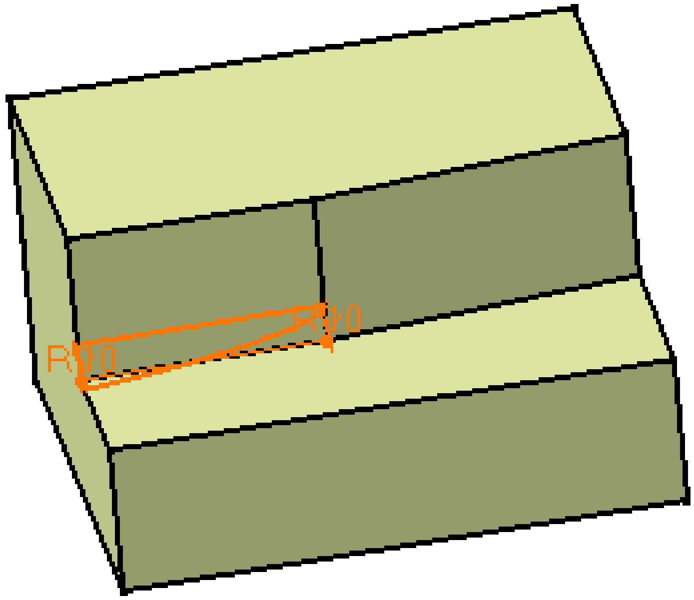
- Check the **Trim support elements** option to relimit the support elements and assemble them to the fillet.

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

Trimming Overlapping Fillets

- In some cases, fillets may be overlapping. The **Trim ribbons** option lets you solve this by trimming the fillets where they overlapping. For further information on this option, refer to Trimming Overlapping Fillets.

- Make sure the support involved does not present any sharp edge, because the fillet would be relimited, and may yield unexpected results, or could not be computed. For example, in the illustration, the fillet cannot be propagated along the whole edge because the fillet is stopped onto the vertical edge.
  In this case, you should try to remove this discontinuity by either filleting the sharp edge or modifying the support surfaces.



- Use the **More>>** button to display further options:
  - the edge that should not be filleted (see Edge Fillet)
  - the circle fillet using a spine (see Variable Radius Fillet Using a Spine)
  - the Blend corner(s) option (see Reshaping Corners).

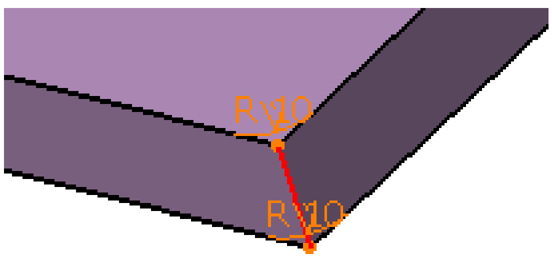
The **Circle Fillet** and the **Blend Corner** options are only available with the Generative Shape Design 2 product.

Filleting Volumes

Open the FilletingVolumes1.CATPart document.

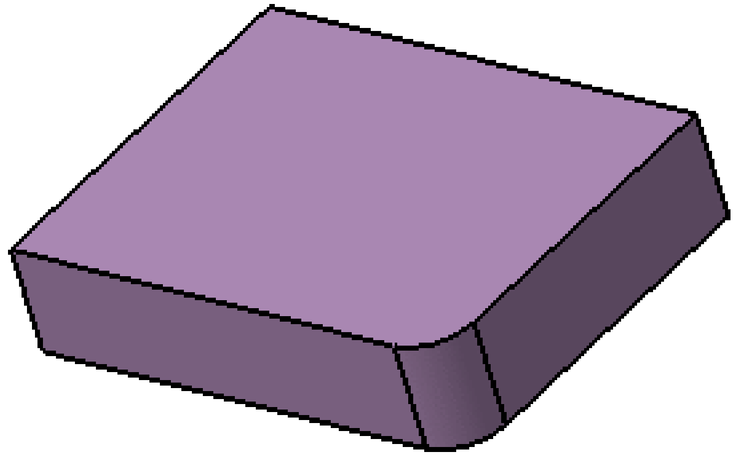
1. Click the **Variable Radius Fillet** icon .

   The Variable Edge Fillet dialog box appears.



2. Select the edge to be filleted.
3. Set the **Radius** to 10mm.

4. Click OK to create the fillet volume.



The **Trim support** option is grayed out, it cannot be used with volumes.

Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting Using A Filter chapter in the *CATIA Infrastructure User's Guide*.

Creating Variable Bi-Tangent Circle Radius Fillets Using a Spine

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

This task shows how to create a variable bi-tangent circle radius fillet on an edge or consecutive edges that do not present any tangency continuity. The propagation along the edge(s) can be done smoothly when selecting a spine along which an arc of circle is slid. Cutting the resulting fillet surface by a plane normal to the spine would result in a circle of the specified radius value.

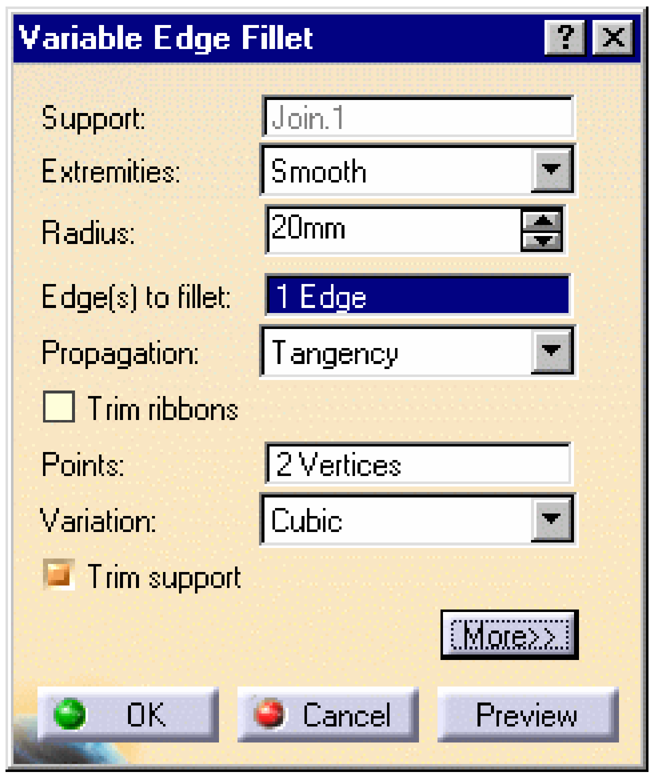
Using this type of radius may help solve twisted fillets created when using any other type of fillet.

Open the FilletVariableRadius2.CATPart document.

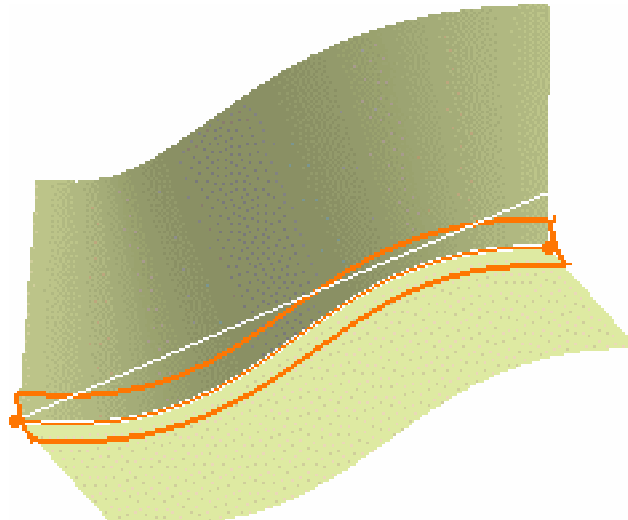
To find out more on variable radius fillets, refer to Variable Radius Fillets.

1. Click the **Variable Radius Fillet** icon .
2. Select the edge(s) to be filleted and click Preview.

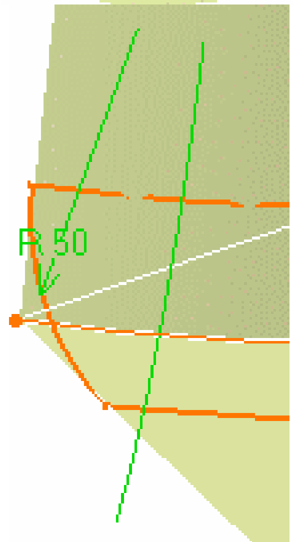
The Variable Edge Fillet dialog box appears.



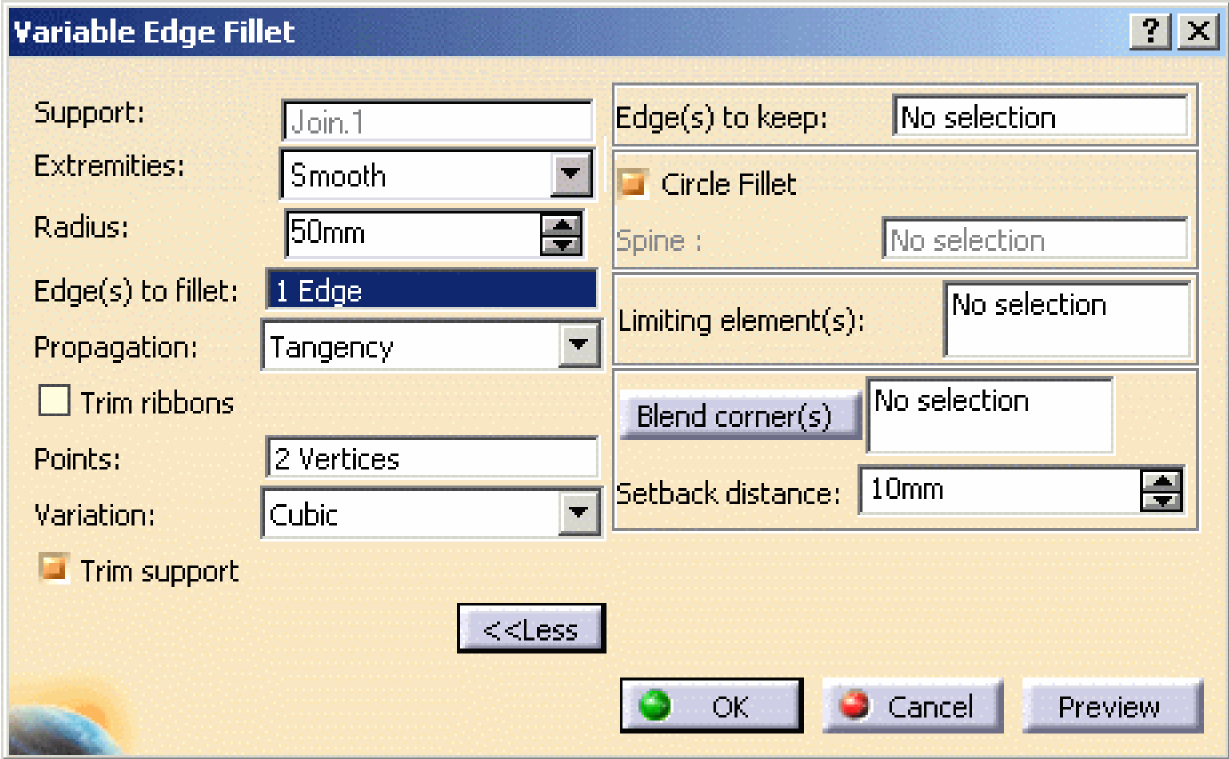
The fillet is previewed on the geometry.

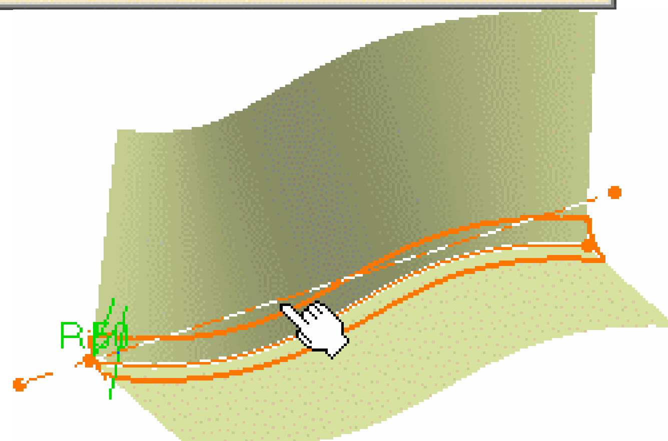


3. Change the radius value to 50mm.



4. Expand the Variable Edge Fillet dialog box and check the **Circle Fillet** option:





5. Select the line as a spine.

6. Click **OK** to create the variable fillet.

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

- The same operation without checking the Circle Fillet option would have led to the following fillet:

- You can use any curve as a spine, provided it covers all selected edges, i.e. it is longer than the set of selected edges.
- In some cases, fillets may be overlapping. The **Trim ribbons** option lets you solve this by trimming the fillets where they overlapping. For further information on this option, refer to Trimming Overlapping Fillets.
- Use the **More>>** button to display further options:
  - the edge that should not be filleted (see Edge Fillet)
  - the circle fillet using a spine (see Variable Radius Fillet Using a Spine)
  - the Blend corner(s) option (see Reshaping Corners).

Creating Face-Face Fillets

This task shows how to create a face-face fillet.
The fillet surface is obtained by rolling a sphere, which radius is larger than the distance between the selected elements, between the selected surfaces.

You generally use the Face-Face fillet command when there is no intersection between the faces or when there are more than two sharp edges between the faces.

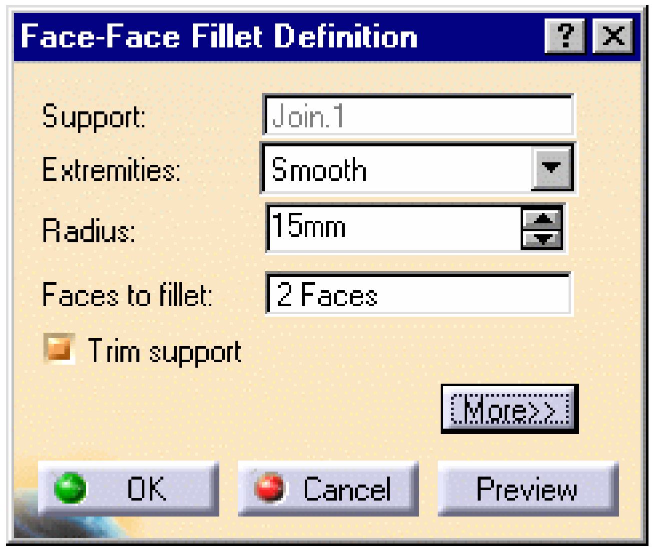
- Limiting Elements
- Hold Curve
- Filleting Volumes

Open the FaceFillet1.CATPart document.

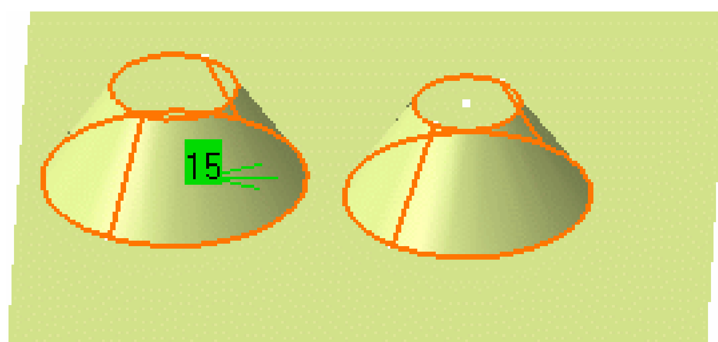
1. Click the **Face-Face Fillet** icon.

   The Face-Face Fillet Definition dialog box appears.

2. Select the two **Faces to fillet**.



3. Select the **Extremities** type, that is the relimitation mode.

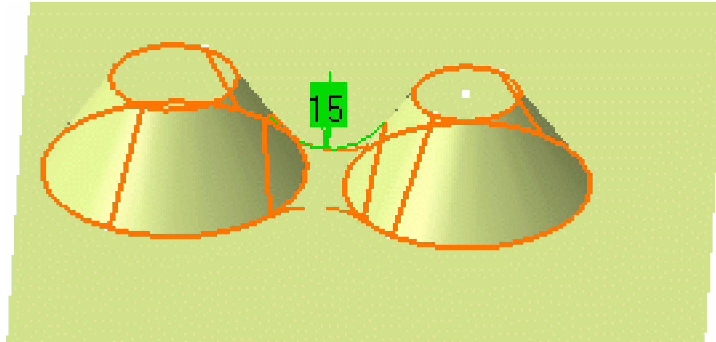


- **Straight:** no tangency constraint is imposed at the connecting point between the fillet and the initial support, generating sometimes a sharp angle
- **Smooth:** a tangency constraint is imposed at the connection between the fillet surface and the support surfaces, thus smoothing the connection
- **Maximum:** the fillet surface is limited by the longest selected edge
- **Minimum**: the fillet surface is limited by the shortest selected edge

(Refer to Shape Fillet)

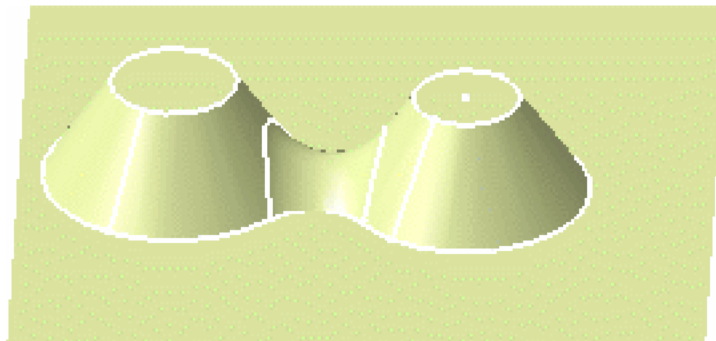
4. Enter a radius value in the **Radius** field if you are not satisfied with the default one. This value must be greater than 0.

5. Click **Preview**.



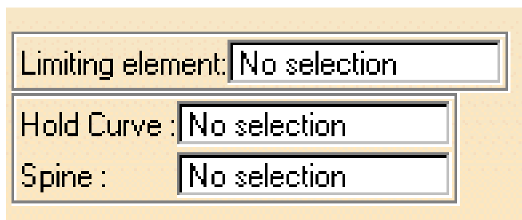
6. Click **OK**.

The faces are filleted. This fillet is indicated in the specification tree.



Check the **Trim support elements** option to relimit the support elements and assemble them to the fillet.

7. Click the **More>>** button.



Limiting Elements

While creating the fillet, you can limit it by selecting an element (plane or surface) that intersects it completely in the **Limiting element** field prior to selecting the trimming element. For further details, refer to Limiting Fillets.

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

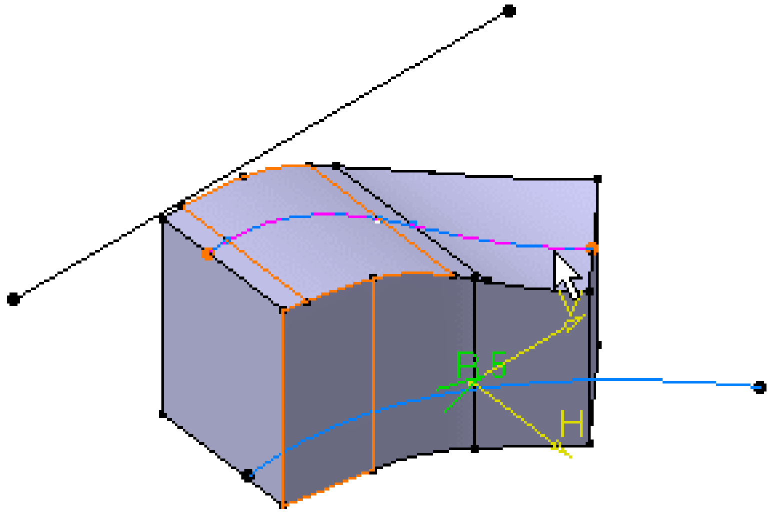
Hold Curve

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

Instead of entering a radius value, you can use a "hold curve" to compute the fillet. Depending on the curve shape, the fillet radius value is then more or less variable.

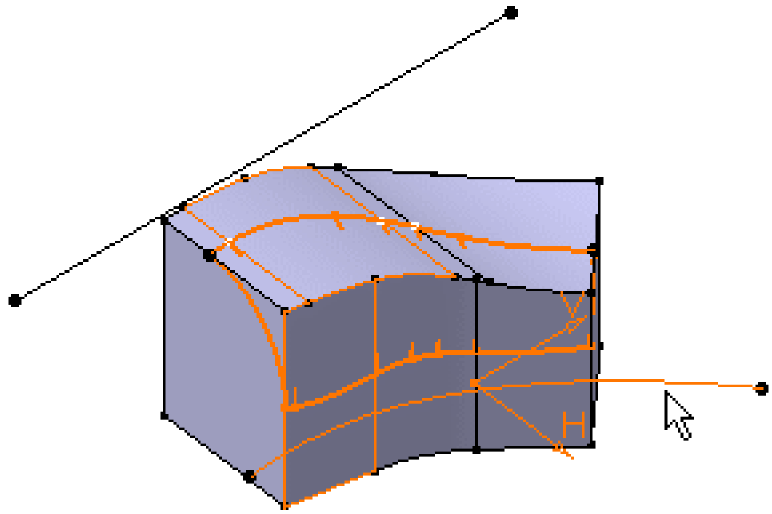
1. Select both faces as shown then Join.2 as the hold curve.

   The curve must be sketched on one of the selected faces.



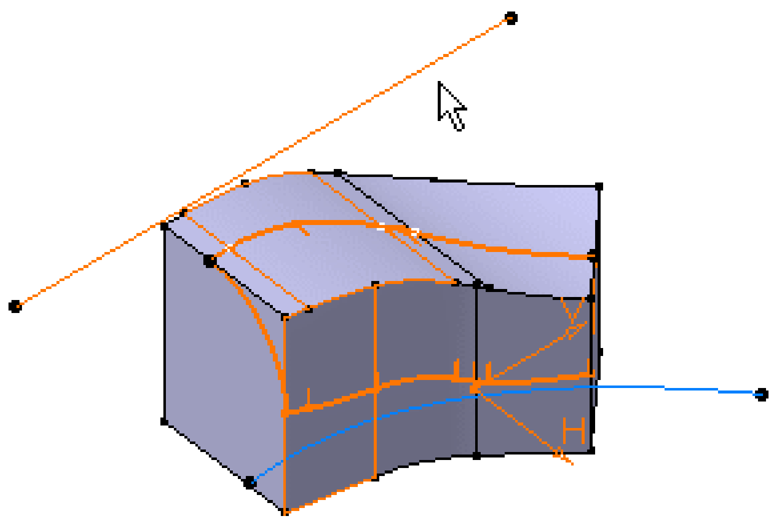
2. Select Sketch.4 as the spine. The spine provides a better control of the fillet.

To compute the fillet, the application uses circles contained in planes normal to the spine. It is then possible to control the shape of the fillet.



The spine can be a wireframe element or a sketcher element.

3. Preview the fillet.
4. Repeat the operation and select Sketch.3 as the spine.
5. Click **OK** to create the fillet.



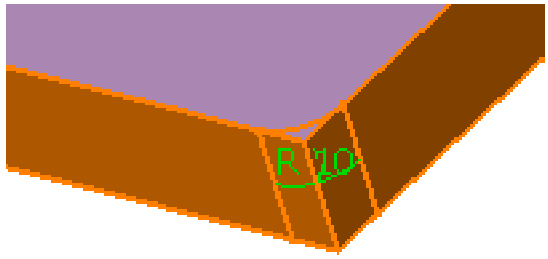
For more information, please refer to the *CATIA V5 Part Design* documentation.

Filleting Volumes

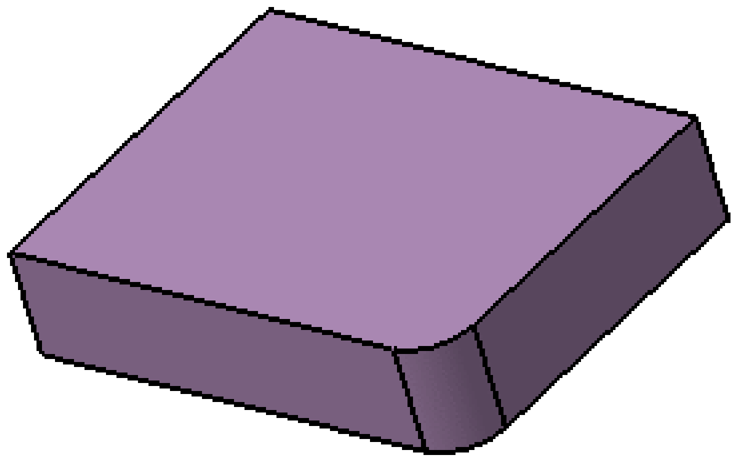
Open the FilletingVolumes1.CATPart document.

1. Click the **Face-Face Fillet** icon .

   The Face-Face Fillet Definition dialog box appears.



2. Select the two **Faces to fillet**.
3. Set the **Radius** to 10mm.
4. Click **Preview**.
5. Click OK to create the fillet volume.



Extremities and **Trim support** options are grayed out, they cannot be used with volumes.

Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting Using A Filter chapter in the *CATIA Infrastructure User's Guide*.

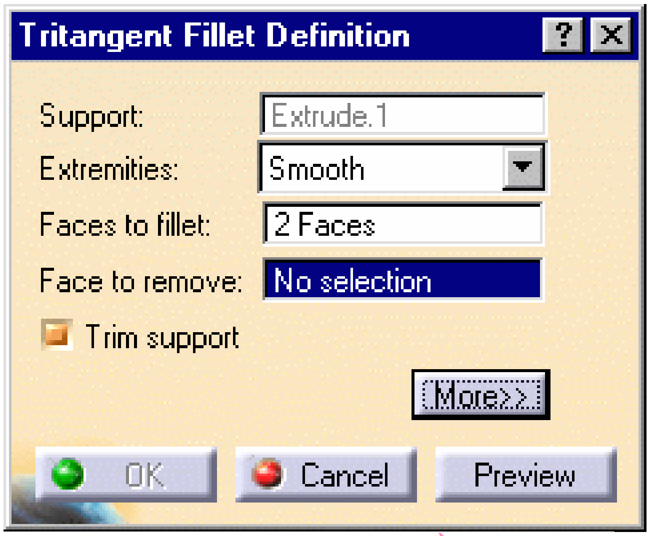
Creating Tritangent Fillets

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

This task shows how to create a tritangent fillet.

The creation of tritangent fillets involves the removal of one of the three faces selected, as the fillet surface is obtained by rolling a sphere, which radius is automatically computed to be larger than the removed surface, between the selected surfaces.

Open the Tritangent1.CATPart document.

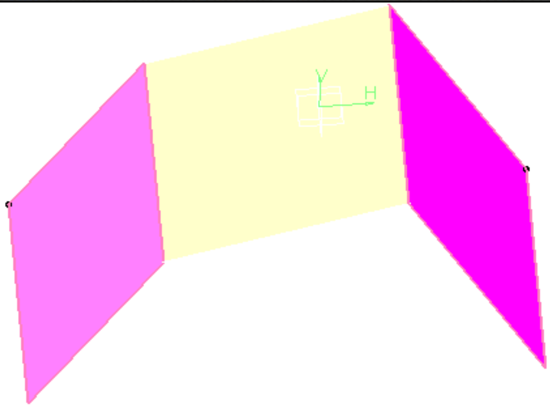


1. Click the **Tritangent Fillet**  icon.

   The Tritangent Fillet Definition dialog box appears.

2. Select the two **Faces to fillet**.

3. Select the **Extremities** that is the relimitation mode:

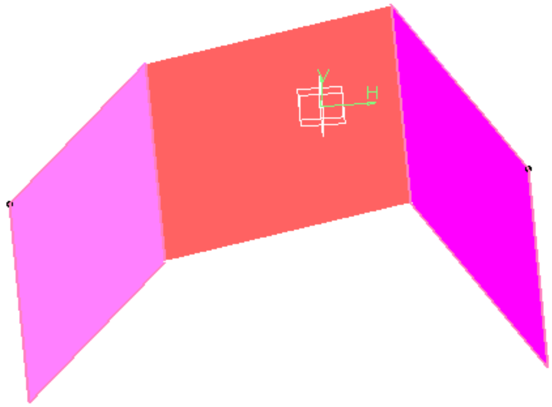


- **Straight:** no tangency constraint is imposed at the connecting point between the fillet and the initial support, generating sometimes a sharp angle.
- **Smooth:** a tangency constraint is imposed at the connection between the fillet surface and the support surfaces, thus smoothing the connection
- **Maximum:** the fillet surface is limited by the longest selected edge
- **Minimum**: the fillet surface is limited by the shortest selected edge

(Refer to Shape Fillets)

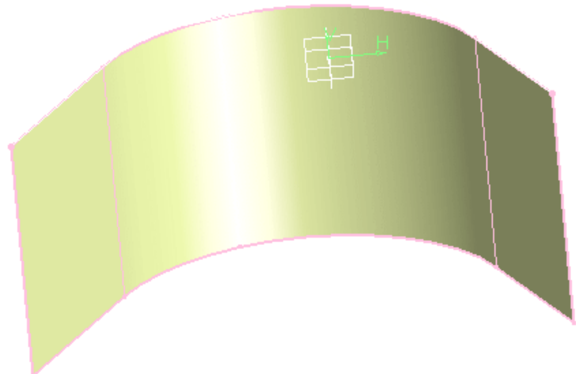
4. Select the **Face to remove**.

   The fillet will be tangent to this face.



5. Click **OK**.

The faces are filleted. The creation of this fillet is indicated in the specification tree.



- Check the **Trim support elements** option to relimit the support elements and assemble them to the fillet.
- While creating the fillet, you can limit it by selecting an element (plane or surface) that intersects it completely. This capability is available when clicking the More>> button, and clicking within the **Limiting element** field prior to selecting the trimming element. For further details, refer to Limiting Fillets.

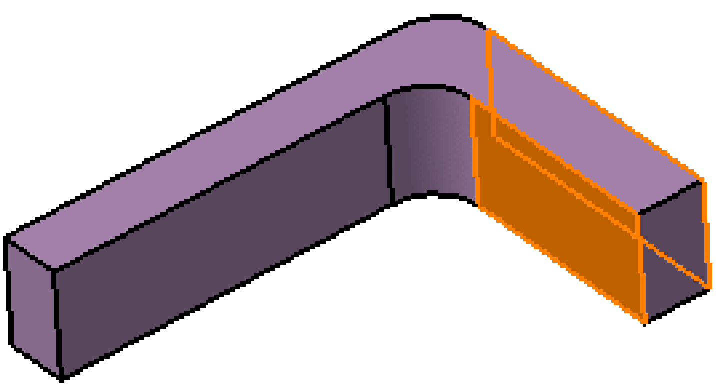
Filleting Volumes

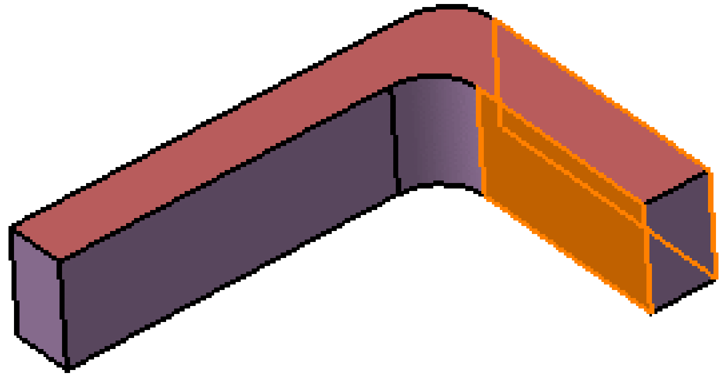
Open the FilletingVolumes2.CATPart document.

1. Click the **Tritangent Fillet**  icon.

   The Tritangent Fillet Definition dialog box appears.

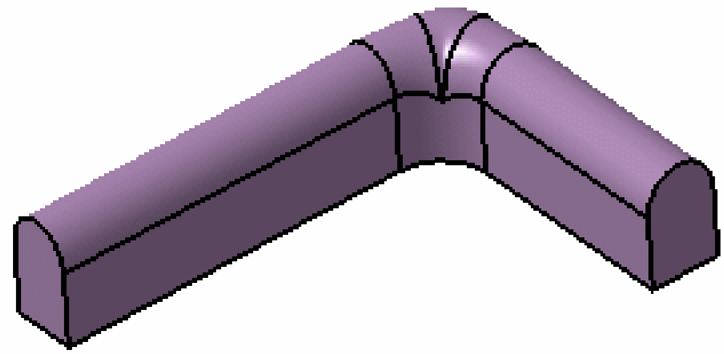
2. Select the two **Faces to fillet**.





3. Select the **Face to remove**.

   The fillet will be tangent to this face.



4. Click OK to create the fillet volume.

Extremities and **Trim support** options are grayed out, they cannot be used with volumes.

Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting Using A Filter chapter in the *CATIA Infrastructure User's Guide*.

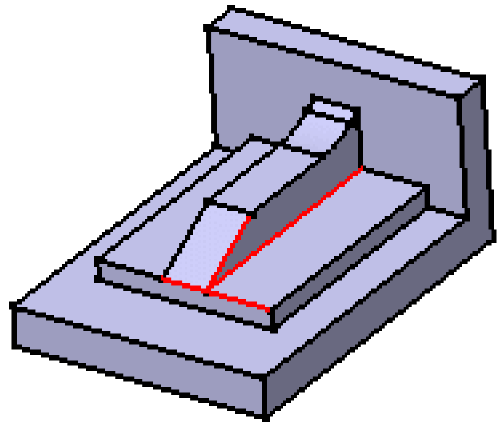
Reshaping Corners

Sometimes, when filleting, you can see that corners resulting from the operation are not satisfactory. This capability lets you quickly reshape these corners.

Open the BlendCorner1.CATPart document.

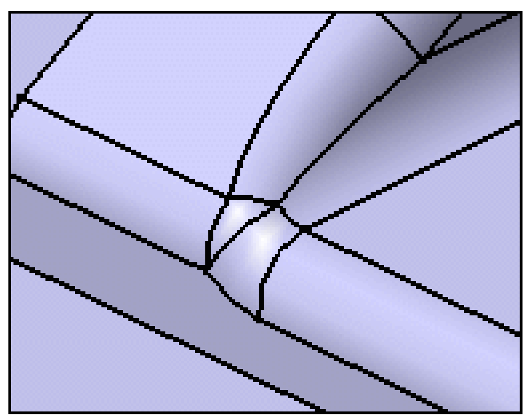
1. Click the **Edge Fillet** icon and fillet the edges as shown using 5 mm as the radius value.

This option is available through the Variable Radius Fillet command too.



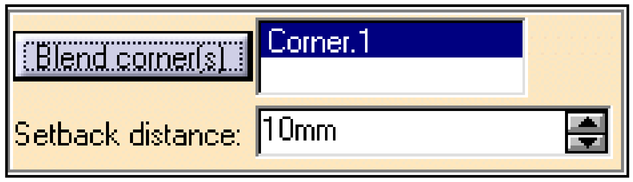
Be careful when selecting edges as the order of selection affects the final shape of the fillet. This explains why you may sometimes encounter error messages when filleting.
To obtain the shape we need for our scenario, please select the edges counter-clockwise.

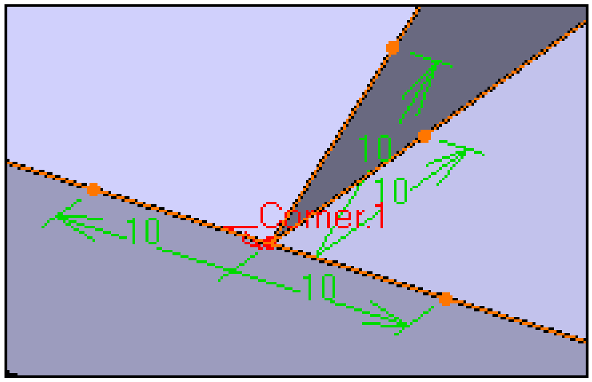
2. Taking a closer look at the corner, you can notice that the edges need to be rounded again.



3. After launching the Edge Fillet dialog box to edit the fillet, click the **More>>** button to access additional options.

4. Click the **Blend corner(s)** button to detect the corner to reshape. In our example, only one corner is detected. The application shows it in the geometry area (3D text).



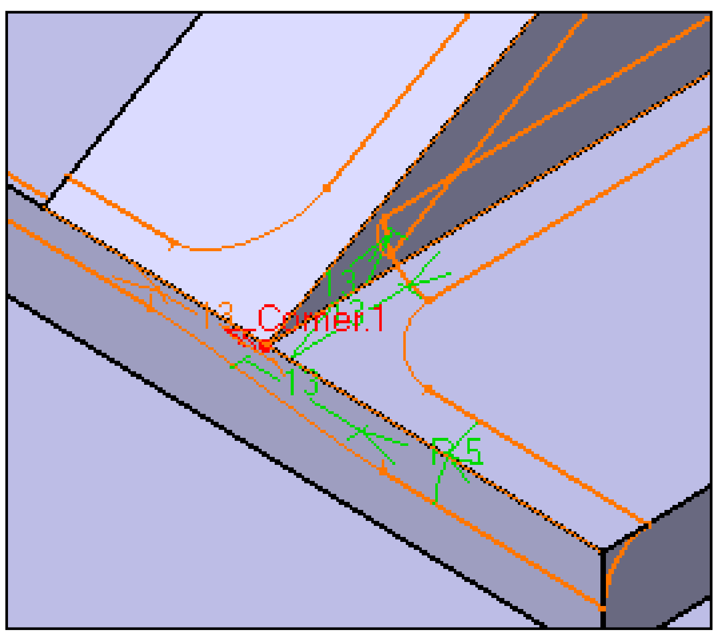


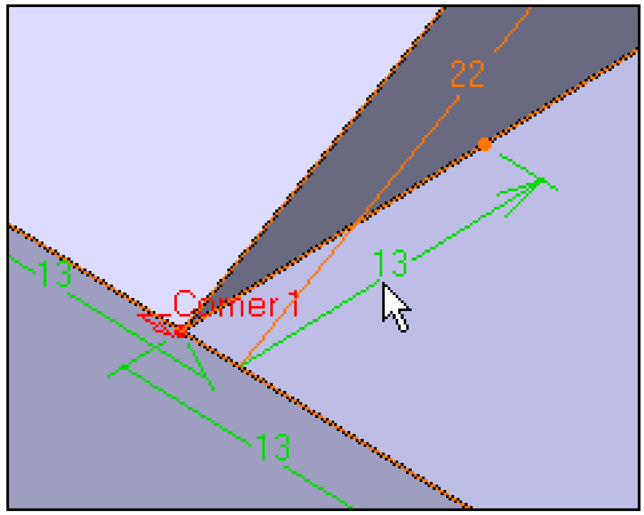
When the application detects several corners, it is not possible to reshape just a few of them: all of them will be edited.

The setback distance field determines for each edge a free area measured from the vertex along the edge. In this area, the system adds material so as to improve the corner shape.

5. Enter a value in the setback distance field. For example, 13.

6. Click **Preview** to examine the result.

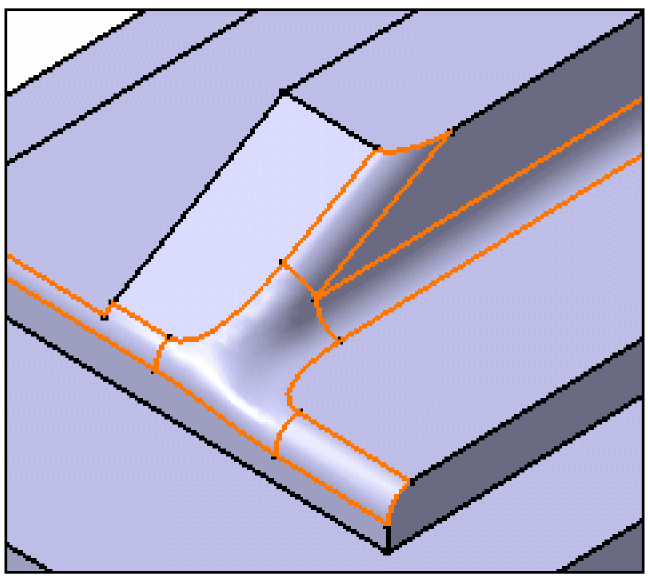




To edit the distance for the top edge, click 13 and enter 22 as the new value in the Setback distance field.

7. Repeat the operation for the edge below using the same distance value.

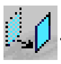
8. Click OK to confirm the operation. The corner is reshaped.

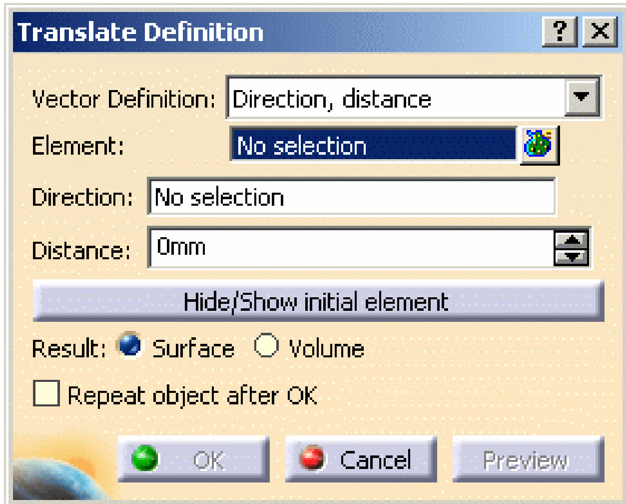


Translating Geometry

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 Tools Palette.

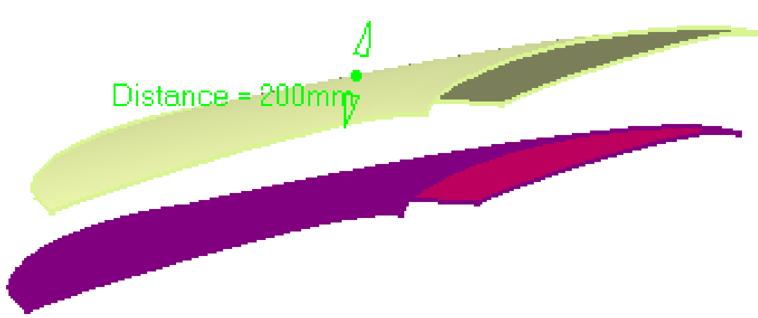
2. Select the **Element** to be translated.

3. Select the **Vector Definition**.

Direction, distance

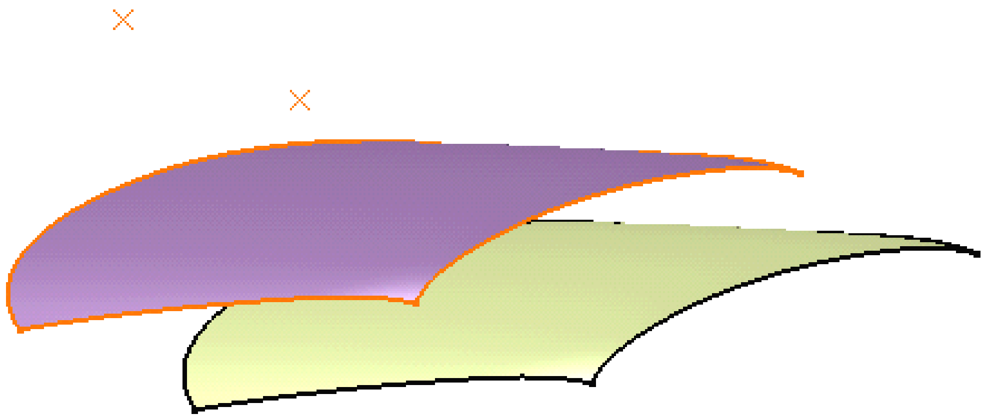
1. Select a line to take its orientation as the translation direction or a plane to take its normal as the translation direction.

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.

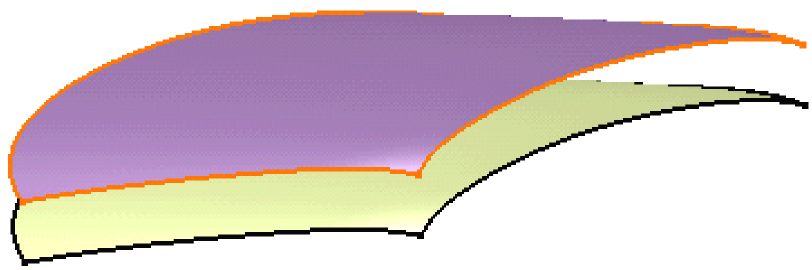
Point to Point



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.

- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  Note that the switch between surface and volume is greyed out when editing the feature.
  To have further information about volumes, please refer to the corresponding chapter.

Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting using a Filter chapter in the *CATIA Infrastructure User's Guide*.

- 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.

- 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.

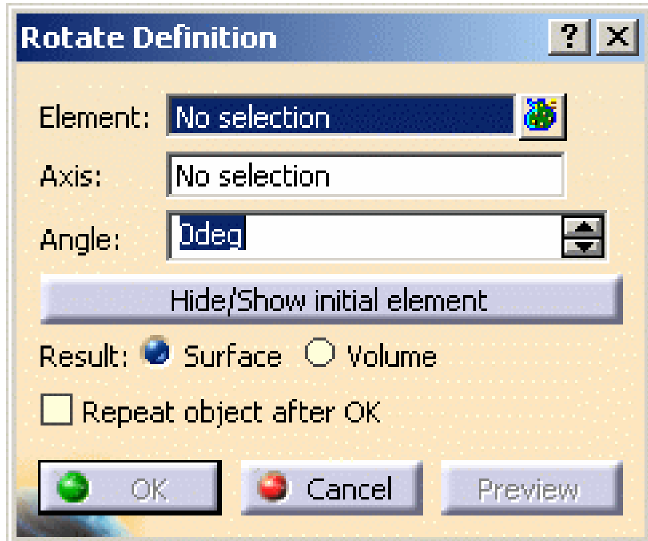
The following capabilities are available: Stacking Commands, Selecting Using Multi-Output, Measure Between and Measure Item.

Rotating Geometry

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

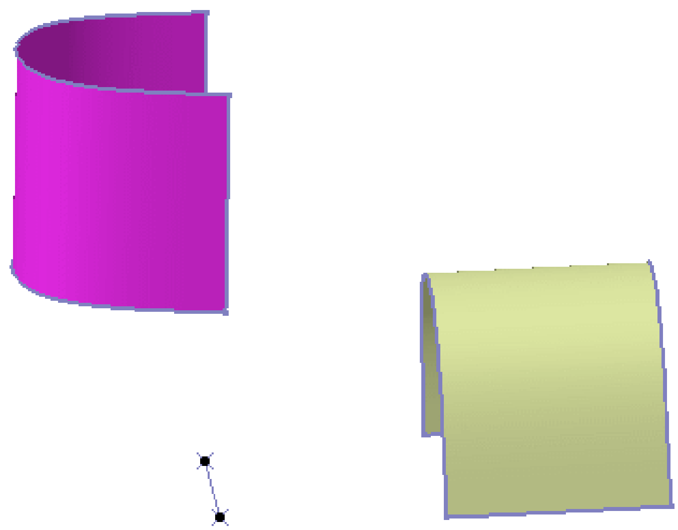
Open the Transform1.CATPart document.

1. Click the **Rotate** icon .



The Rotate Definition dialog box appears as well as the Tools Palette.

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**.



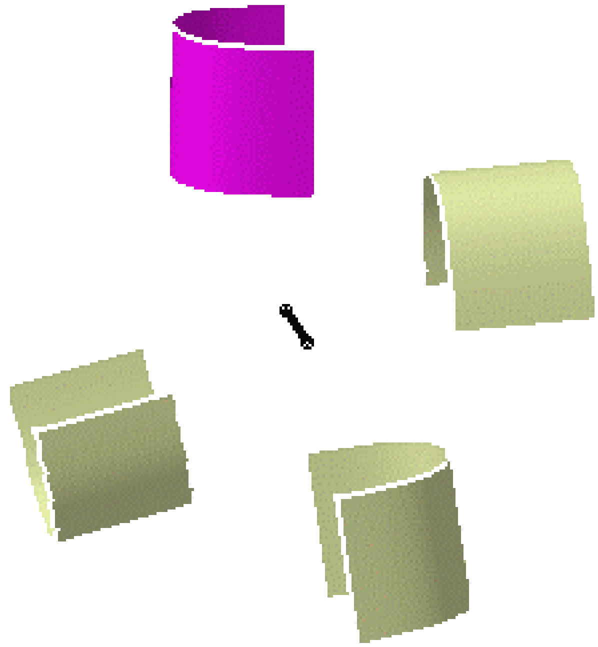
5. Click **OK** to create the rotated element.

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

- Use the **Hide/Show initial element** button to hide or show the original element for the translation.
- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  Note that the switch between surface and volume is greyed out when editing the feature.
  To have further information about volumes, please refer to the corresponding chapter.

Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting Using A Filter chapter in the *CATIA Infrastructure User's Guide*.

- 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 rotated, providing it was previously created.

The element is identified as Rotate.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.

Performing a Symmetry on Geometry

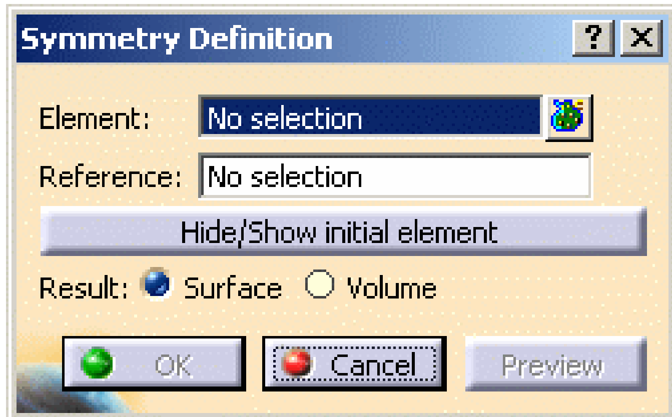
This functionality is P2 for FreeStyle Shaper, Optimizer, and Profiler.

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

Open the Transform1.CATPart document.

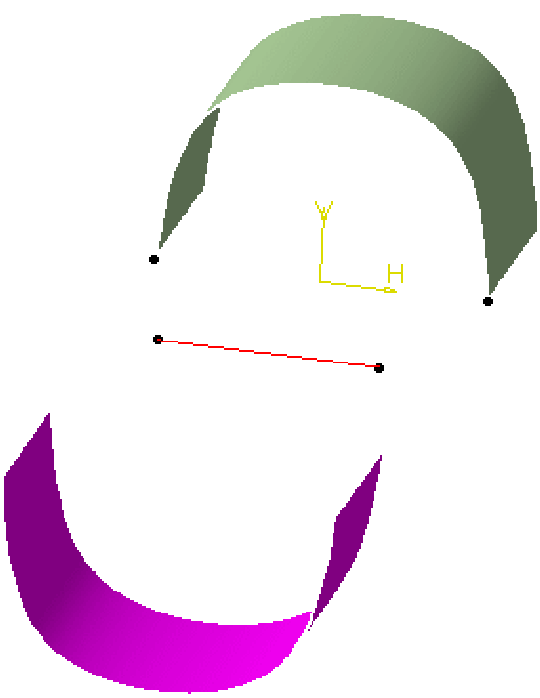
1. Click the **Symmetry** icon.

The Symmetry Definition dialog box appears as well as the Tools Palette.

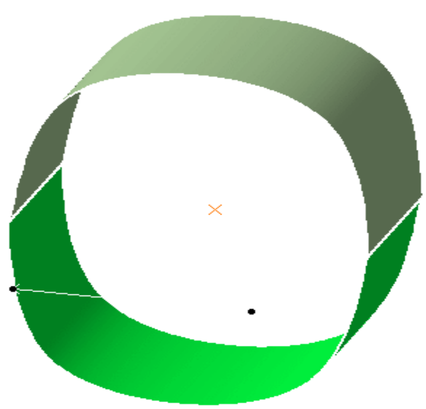


2. Select the **Element** to be transformed by symmetry.
3. Select a point, line or plane as **Reference** element.

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 **Hide/Show initial element** button to hide or show the original element for the translation.
- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  Note that the switch between surface and volume is greyed out when editing the feature.
  To have further information about volumes, please refer to the corresponding chapter.

- You can select an axis system as the **Element** to be transformed, providing it was previously created.
  The element is identified as Symmetry.xxx in the specification tree, however the associated icon is the axis system's .

Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting using a Filter chapter in the *CATIA Infrastructure User's Guide*.

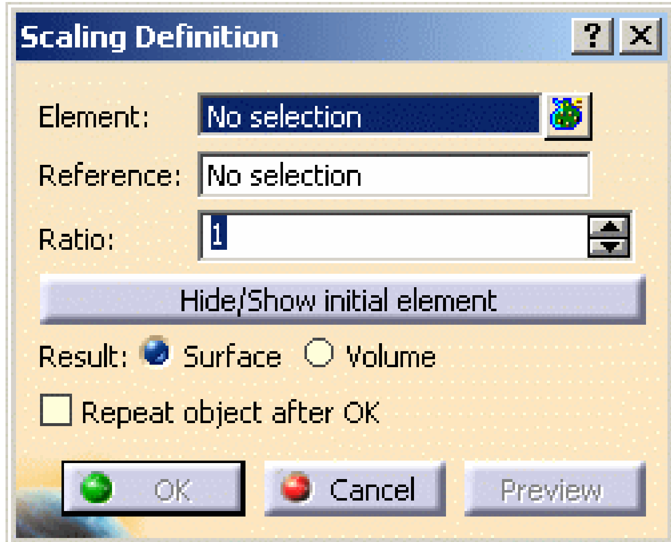
The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.

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 .



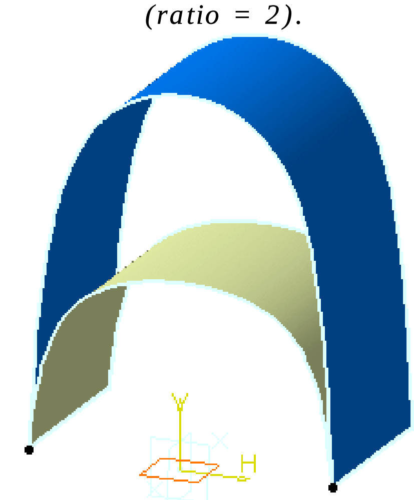
The Scaling Definition dialog box appears as well as the Tools Palette.

2. Select the **Element** to be transformed by scaling.

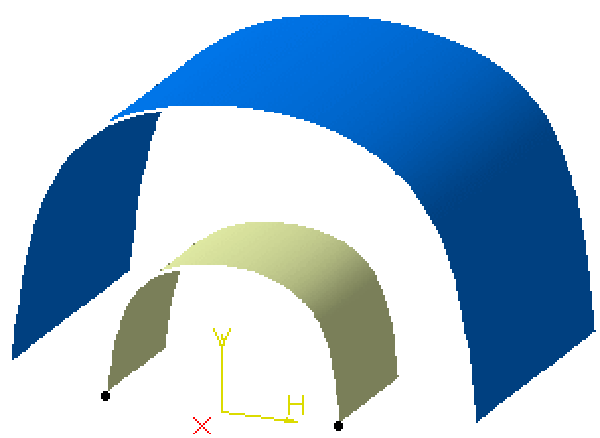
3. Select the scaling **Reference** point, plane or planar surface.

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).



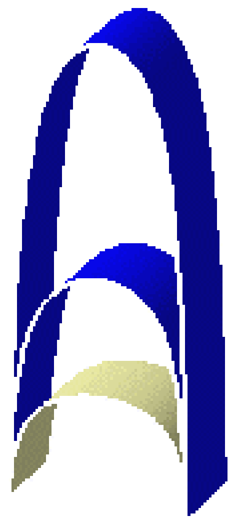
5. Click OK to create the scaled element.

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

- Use the **Hide/Show initial element** button to hide or show the original element for the translation.
- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  Note that the switch between surface and volume is greyed out when editing the feature.
  To have further information about volumes, please refer to the corresponding chapter.

Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting using a Filter chapter in the *CATIA Infrastructure User's Guide*.

- 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.



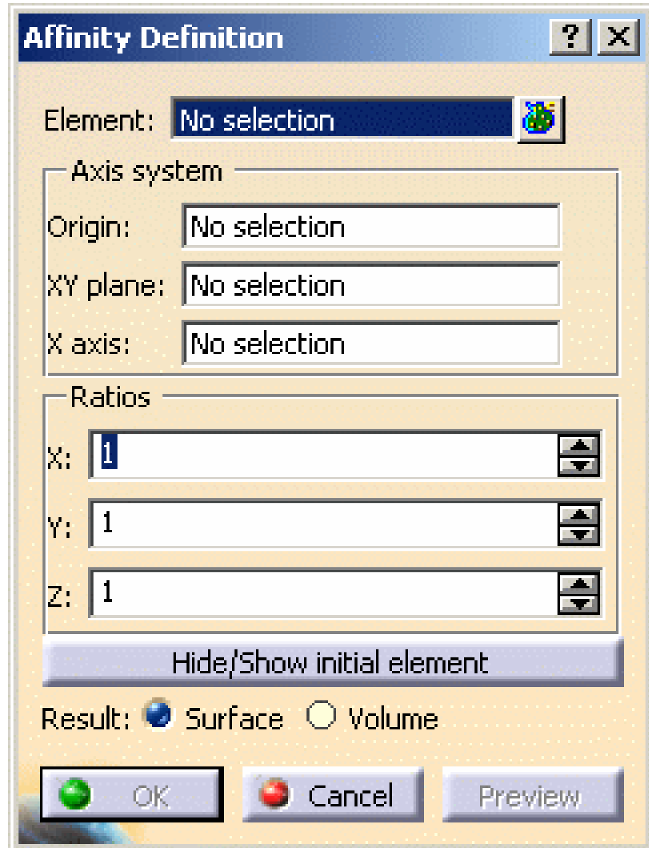
The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.

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 .



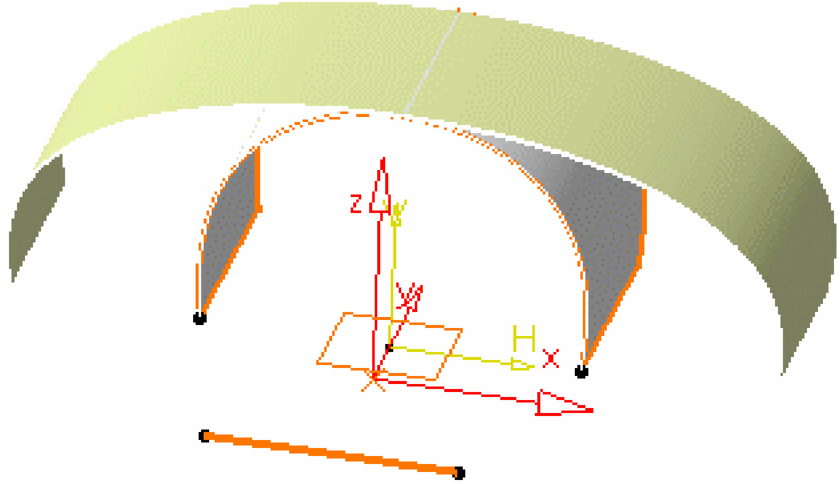
The Affinity Definition dialog box appears as well as the Tools Palette.

2. Select the **Element** to be transformed by affinity.
3. Specify the characteristics of the **Axis system** to be used for the affinity operation:

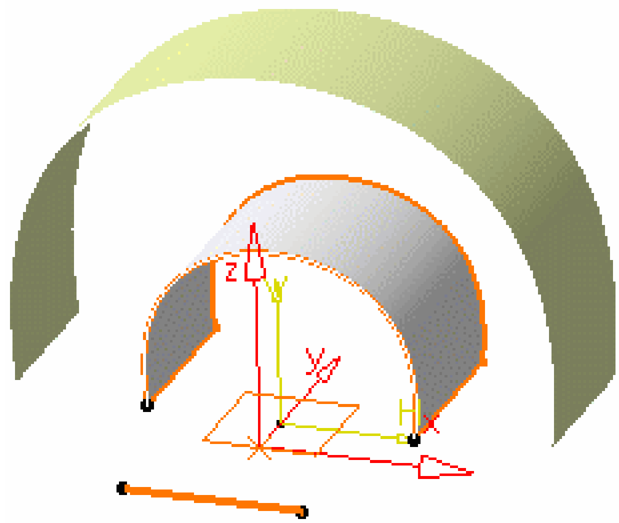
- the **Origin** (Point.1 in the figures below)
- the **XY plane** (the XY plane in the figures below)
- the **X axis** (Line.1 in the figures below).

4. Specify the affinity **Ratios** by entering the desired **X**, **Y**, **Z** values.

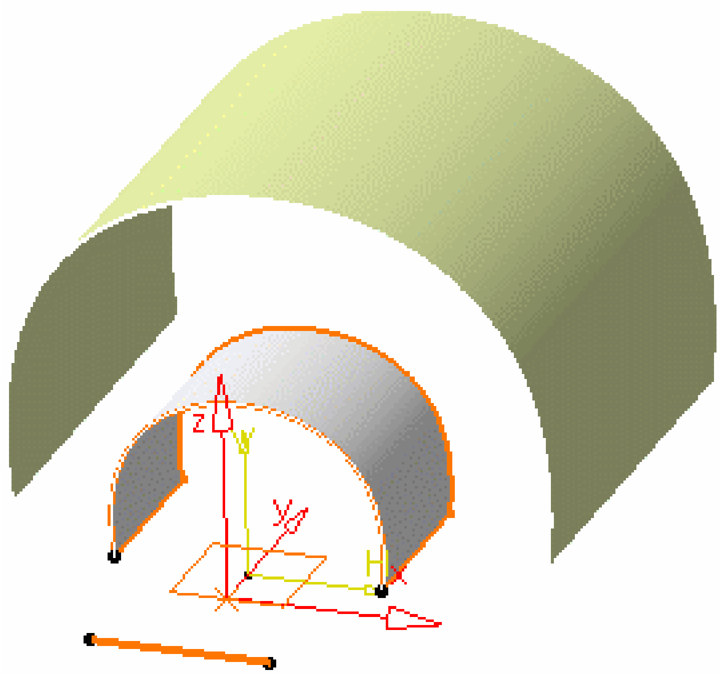
The figure below illustrates the resulting affinity with ratios X = 2, Y =1 and Z=1.



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 **Hide/Show initial element** button to hide or show the original element for the translation.
- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  Note that the switch between surface and volume is greyed out when editing the feature.
  To have further information about volumes, please refer to the corresponding chapter.

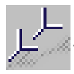
Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting using a Filter chapter in the *CATIA Infrastructure User's Guide*.

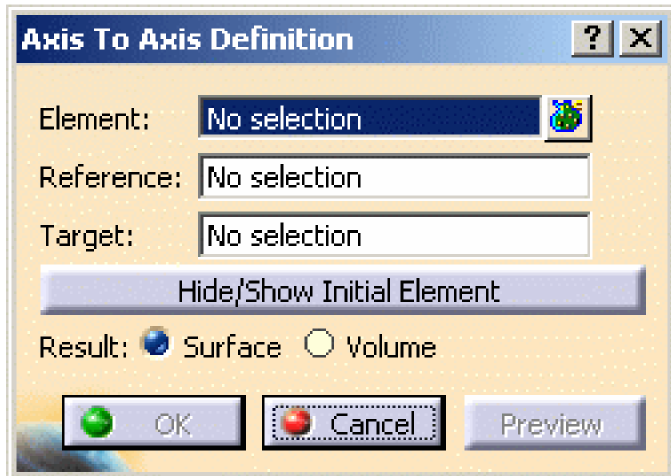
The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.

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.

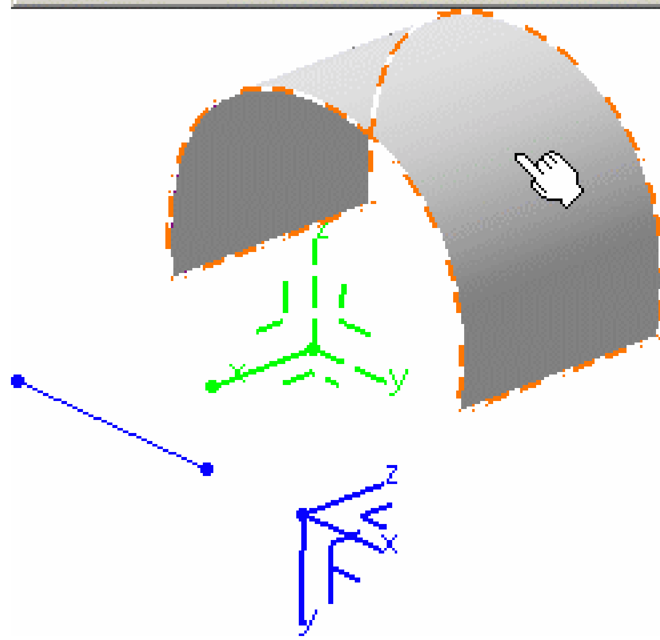
Open the Transform2.CATPart document.

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

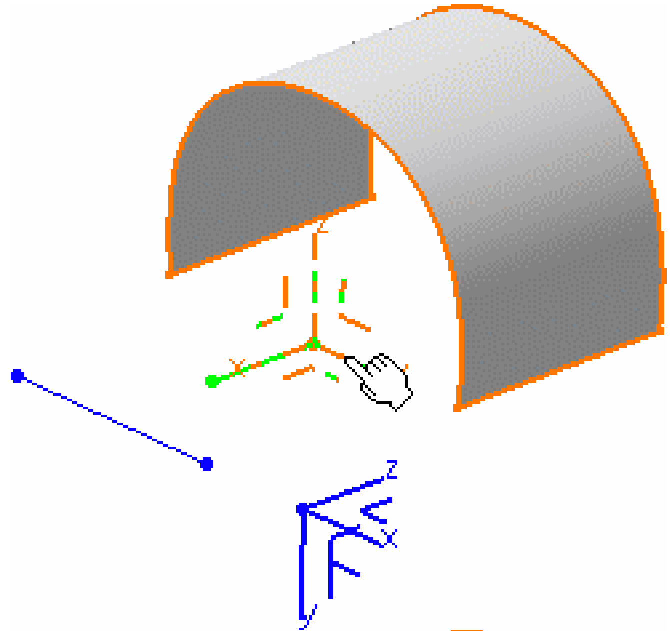


The Axis to Axis Definition dialog box appears as well as the Tools Palette.

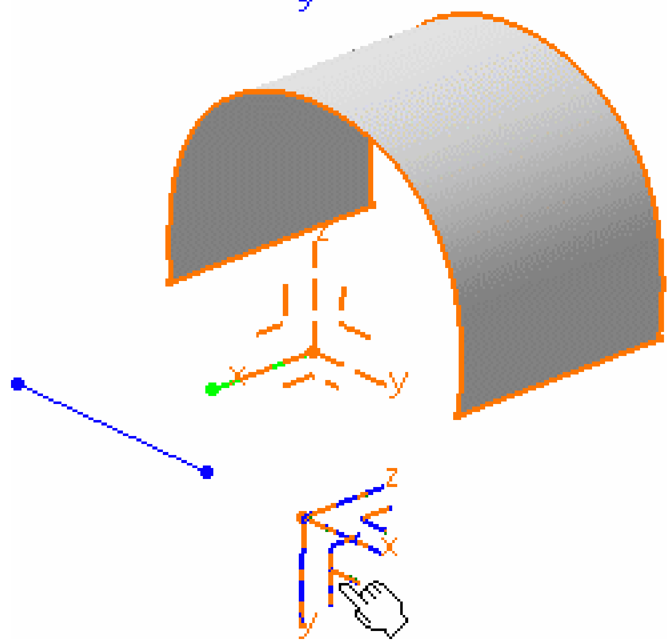
2. Select the **Element** to be transformed into a new axis system.



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



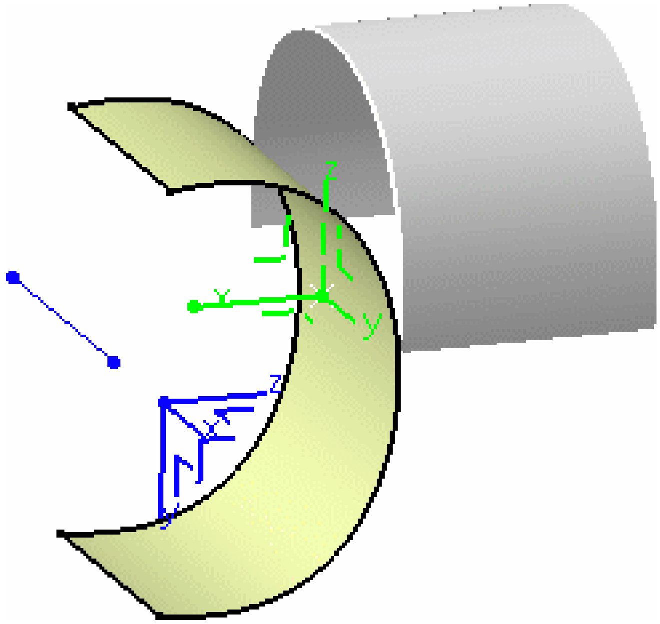
4. Select the **Target** axis system, that is the one into the element should be positioned.



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 **Hide/Show initial element** button to hide or show the original element for the translation.

- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  Note that the switch between surface and volume is greyed out when editing the feature.
  To have further information about volumes, please refer to the corresponding chapter.

- You can select an axis system as the **Element** to be transformed, providing it was previously created.
  The element is identified as Axis to axis transformation.xxx in the specification tree, however the associated icon is the axis system's .

Note that the selection of the feature prevails over the selection of the sub-element.
To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar.
For further information, refer to the Selecting using a Filter chapter in the *CATIA Infrastructure User's Guide*.

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

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.

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

4. 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.

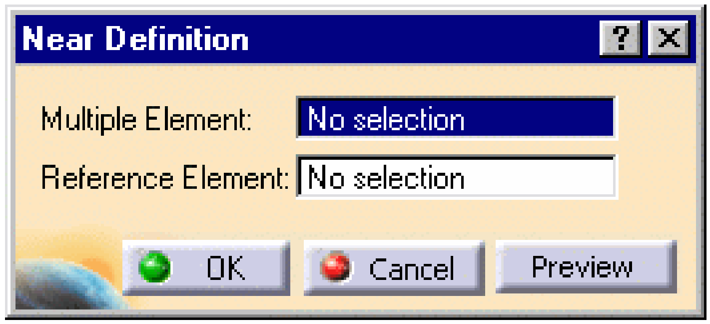
Creating the Nearest Entity of a Multiple Element

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 from the samples directory.

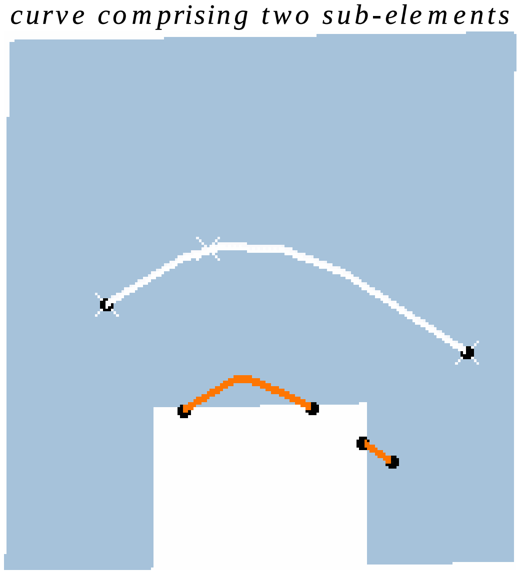
1. Select the **Insert -> Operations -> Near** command.

   The Near Definition dialog box appears.

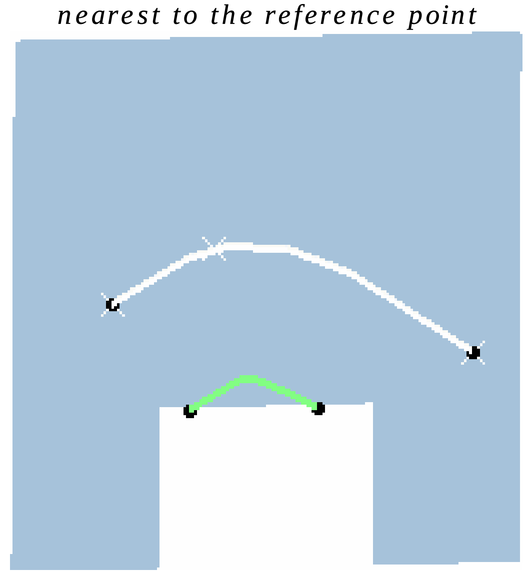


2. Select the element that is made up from several sub-elements.
3. Select a reference element whose position is close to the sub-element that you want to create.

The example below shows a parallel curve comprising two sub-elements



The example below 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.

Creating Laws

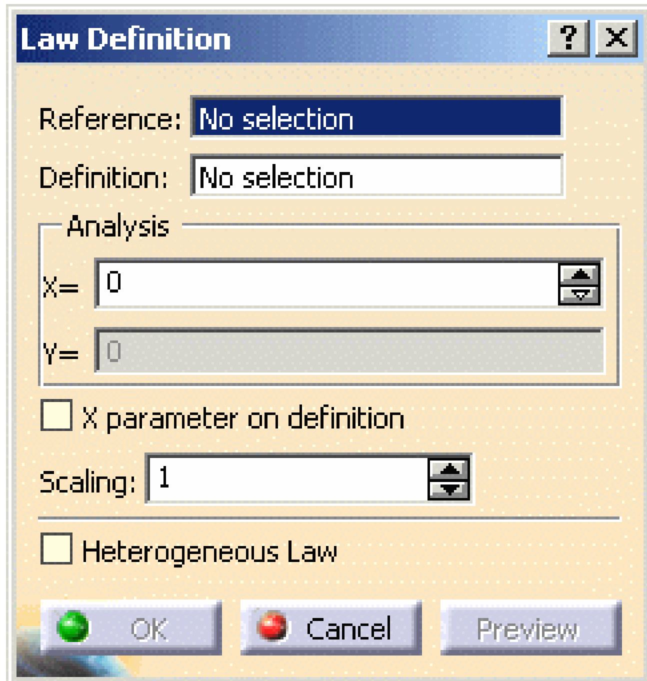
This command is only available with the Generative Shape Design 2 product.

This task shows how to create evolution laws within a .CATPart document, to be used later on when creating Generative Shape Design elements, such as swept surfaces, or parallel curves.

Open the Law1.CATPart document.

1. Click the **Law** icon.

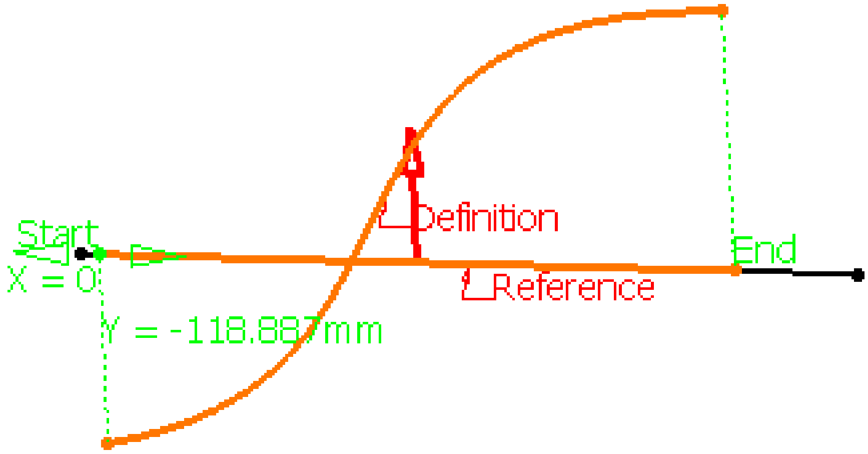
   The Law Definition dialog box appears.



2. Select the reference line.

3. Select a definition curve.

The law is computed as the distance between points on the reference line and their matching points onto the curve.



Laws can be created using negative values.
The intersection between the reference line and the definition curve is taken into account to change the law evaluation sign.
The direction lets allows you to choose which side of the reference line must be considered as positive.

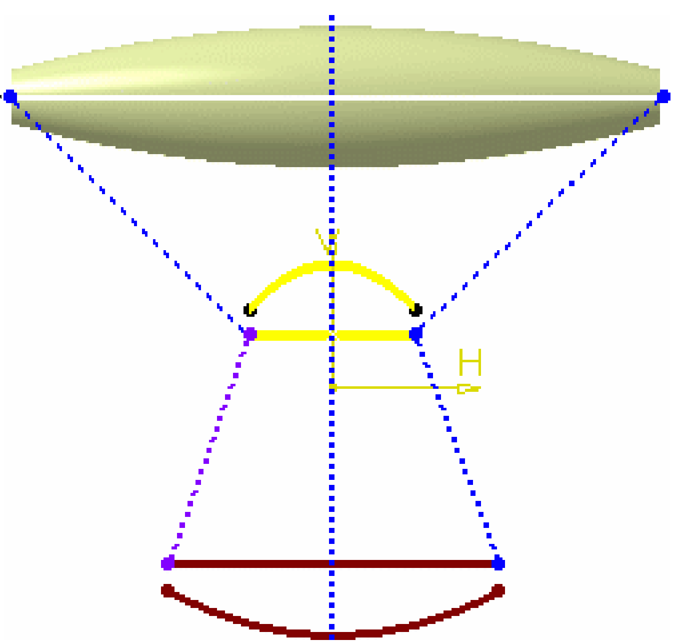
If the **X parameter on definition** option is checked, the X parameter is displayed on the selected curve and represents the percentage of the curvilinear abscissa on this curve. The law is computed by projecting the start point normally onto the reference line.

You can analyze the law using the manipulator,, or specifying a value in the X field. This parameter represents the percentage of the curvilinear abscissa on this curve. The law is computed by projecting the start point normally onto the reference line.
The Y field indicates the distance between any point on the reference line and its matching point on the selected curve.

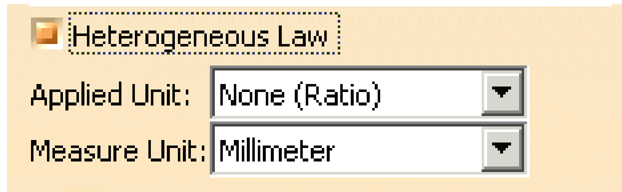
4. Define the law amplitude by entering a value or using the graphic manipulators in the **Scaling** field.

When the law is applied to a geometric element, the latter usually is not of the same length as the reference line. Therefore a linear mapping is applied between the reference line and the element the law is applied to, resulting in a scaling of the law.

In the illustration, the law is applied to a circular sweep (top) and to a parallel curve (bottom). The dotted lines represent the linear mapping between the law (middle) and the two elements to which it is applied.



5. Click the **Heterogeneous Law** button to define the applied law unit (none for ratio law; degree, radian, or grade for angle law) and the distance measure units (current unit by default).
Two conversions will be performed during the law evaluation:



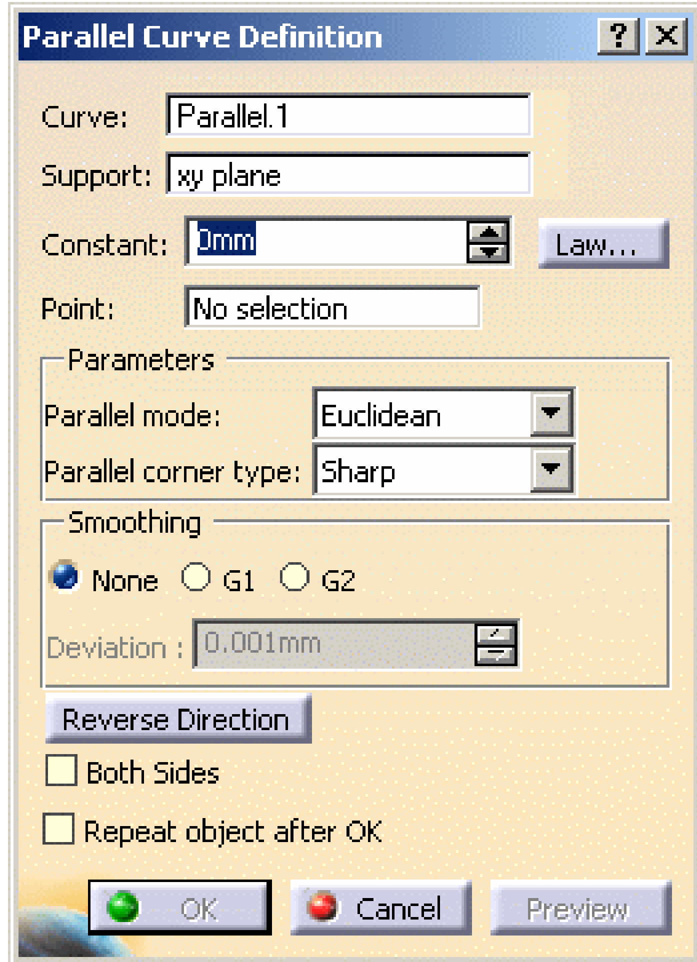
- conversion from the model unit (millimeters) to the stored measure unit
- conversion form the stored applied law unit to the model unit (degrees) in case of an angle

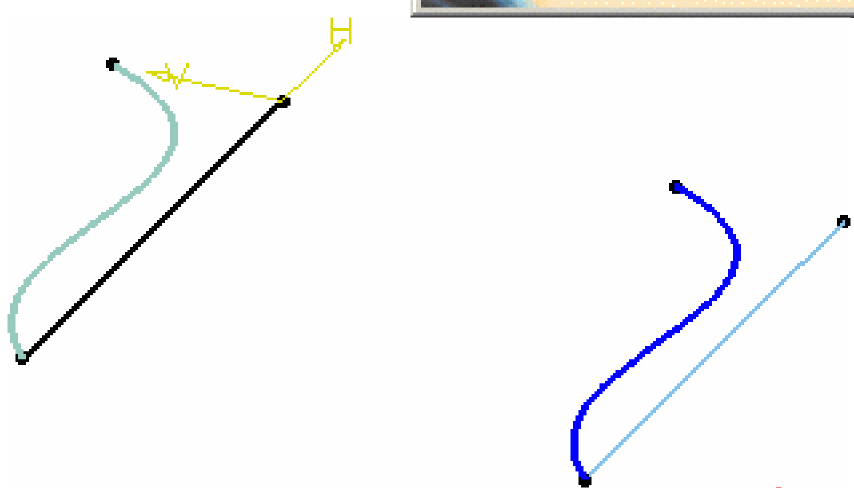
6. Click OK to create the law.

The law (identified as Law.xxx) is added to the specification tree.
It is now ready for use in the creation of other Shape Design elements.

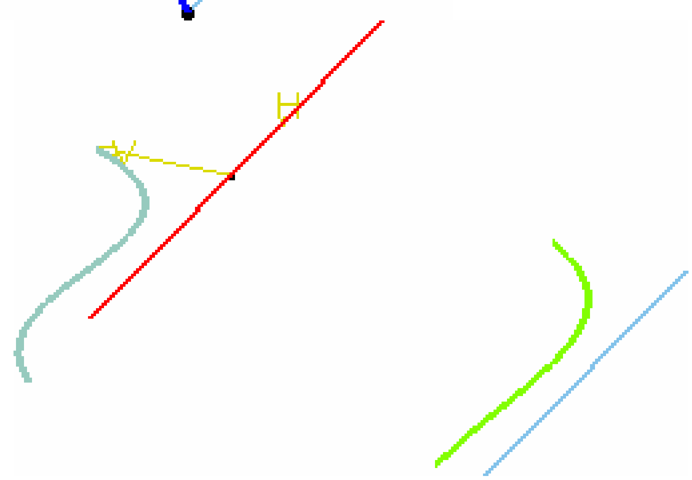
7. Click the **Parallel Curve** icon .
8. In the Parallel Curve Definition dialog box, click the **Law...** button.
9. Select the Law.1 from the specification tree.
10. Click OK.

The law is applied to the selected element.

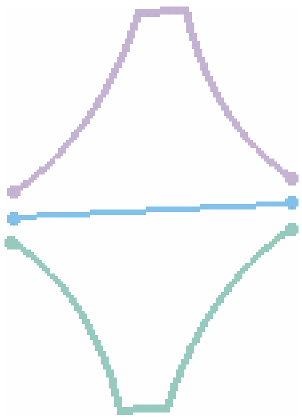




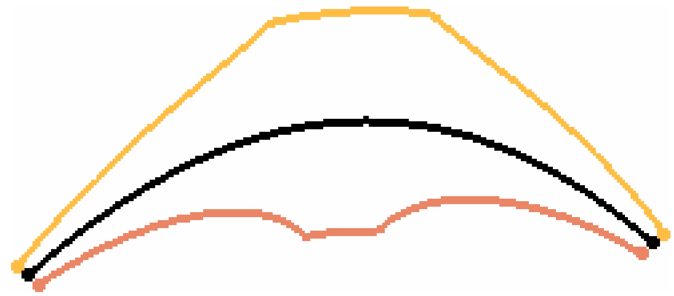
- When the reference line and definition curve do not present the same length, only the common area is used to compute the law.



- Check the **Both sides** option to generate a parallel curve symmetrically on each side of the selected curve.
  Note that depending on the geometry, the elements may not appear symmetrical. They are if the curve is a line, otherwise, the resulting curves' shape may differ:



Resulting parallel curves when a line is selected



Resulting parallel curves when any curve is selected

- When the **X parameter on definition** option is unchecked, the selected curve should not present several intersections with the plane normal to the reference line. If there are several intersections, the law cannot be evaluated and cannot be applied when creating geometric elements.

- Laws created using the Knowledge Advisor product, being mathematical formulas, can be used with Generative Shape Design's operators, such as swept surfaces, or parallel curves for example.
  For further information, refer to the *Knowledge Advisor's User's Guide*, *Basic Tasks*, Creating and Using a Knowledge Advisor Law.
  Note that laws created with the **Law** icon of Generative Shape Design product, can be referenced by laws created with Knowledge Advisor product.

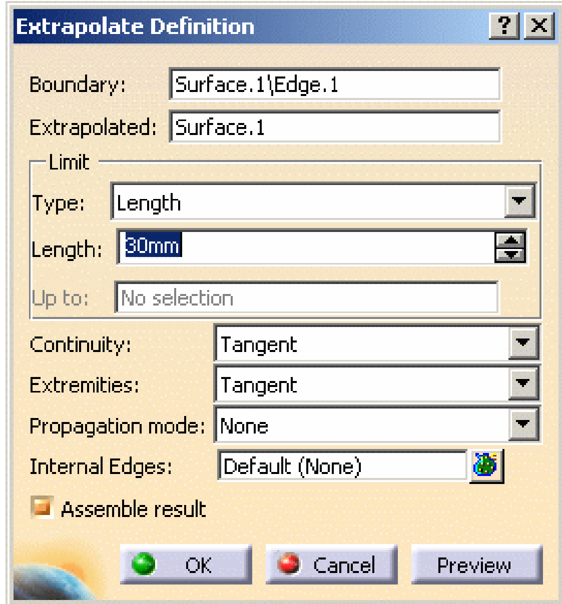
Extrapolating Surfaces

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**.

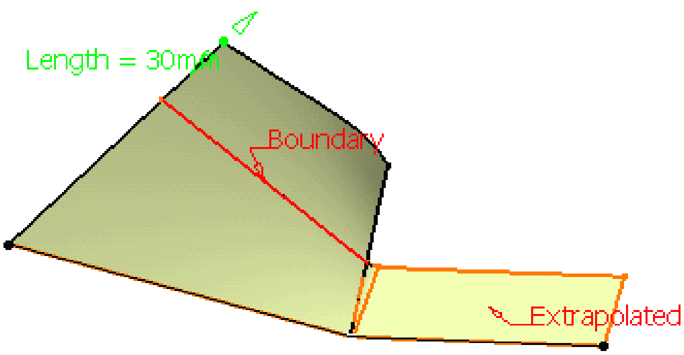
3. Select the surface to be **Extrapolated**.

4. Specify the **Limit** of the extrapolation by either:

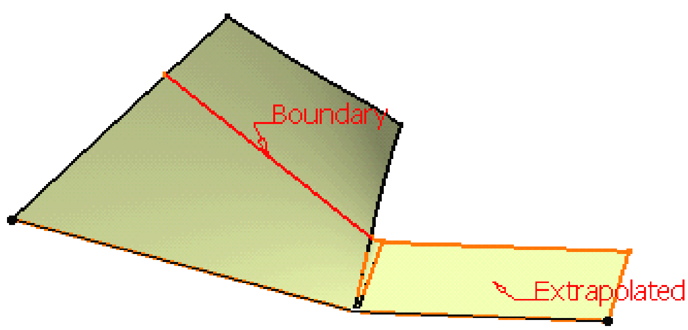
- entering the value of the extrapolation length
- selecting a limit surface or plane
- using the manipulators in the geometry.

5. Specify the **Continuity** type:

- **Tangent**
- **Curvature**



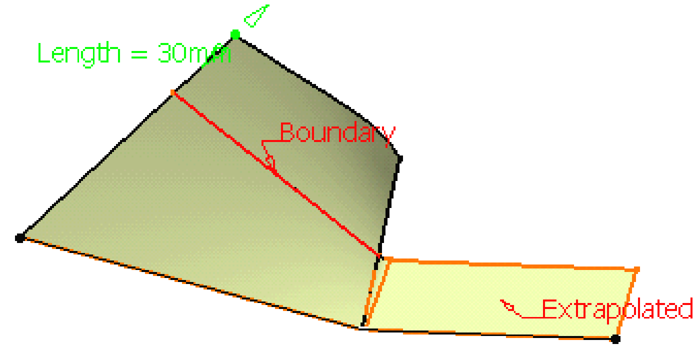
Tangent



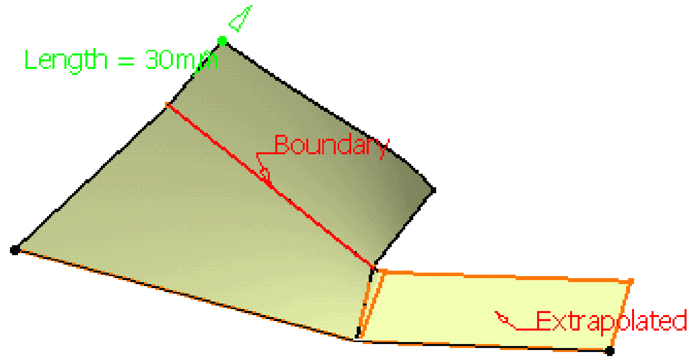
Curvature

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.



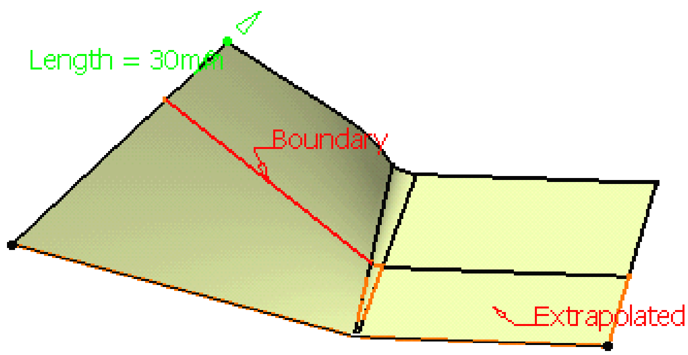
Tangent



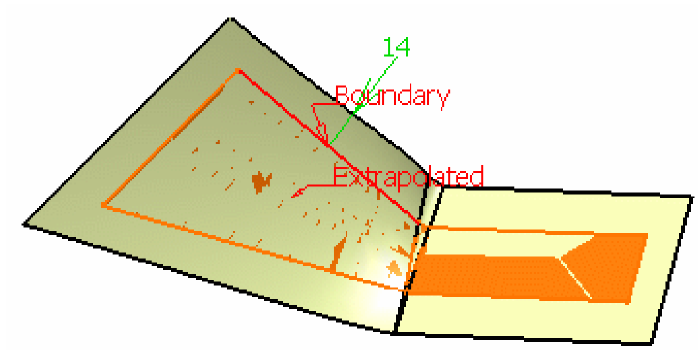
Normal

7. Specify the **Propagation** type:

- **Tangency continuity** to propagate the extrapolation to the boundary's adjacent edges.
- **Point continuity** to propagate the extrapolation around all the boundary's vertices.



Tangent continuity



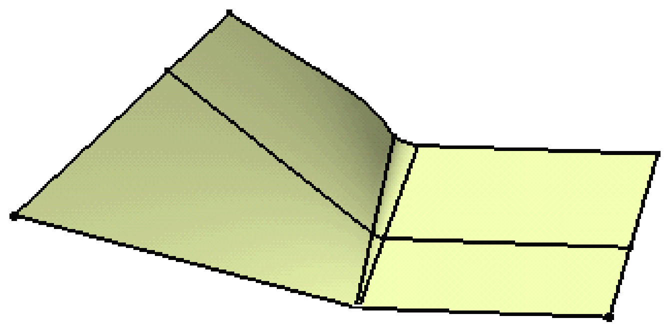
Point continuity

8. Check the **Assemble result** option if you want the extrapolated surface to be assembled to the support surface.

This option is now also available with the Curvature continuity.

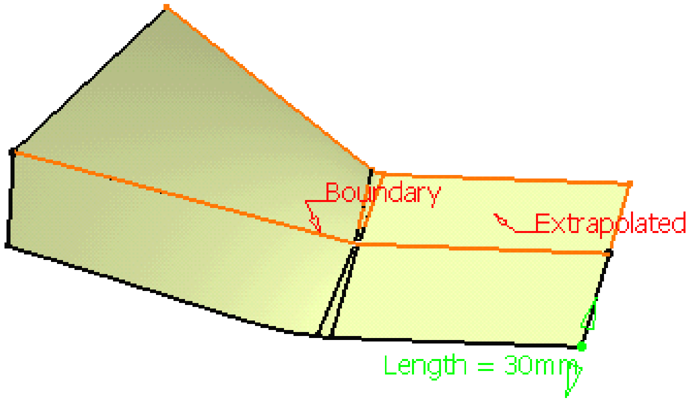
9. Click **OK** to create the extrapolated surface.

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

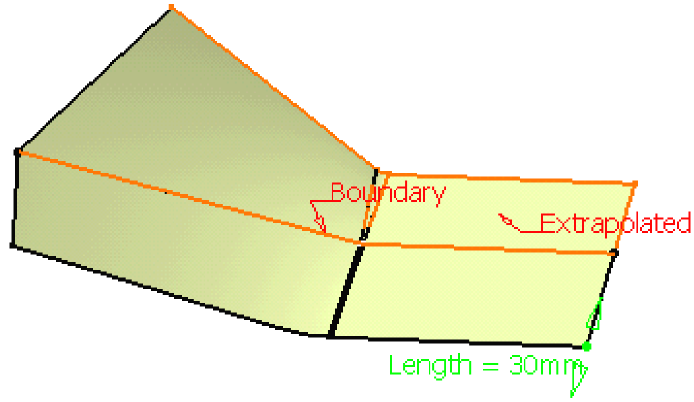


The **Internal Edges** option enables to determine a privileged direction for the extrapolation. You can select one or more edges (in the following example we selected the edge of Surface.1) that will be extrapolated in tangency. You can also select a vertex once you have selected an edge in order to 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 not available 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**, and the **Internal Edges** options are not available with the **Curvature** continuity type.

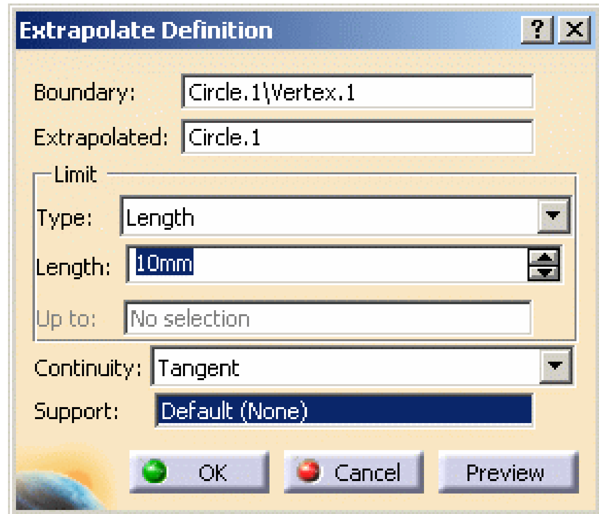
Extrapolating Curves

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.



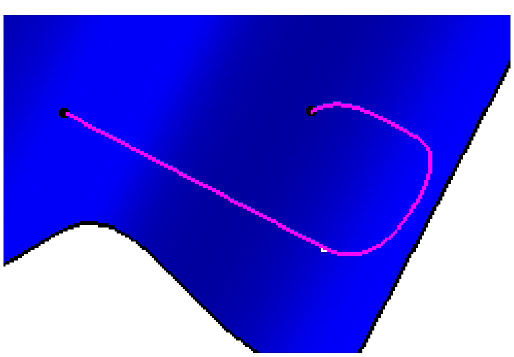
2. Select an endpoint on a curve.
3. Select the curve to be **Extrapolated** (it can be a wire, an edge, a curve or a line)
4. Select the extrapolation type:
   - **Length**: enter the value in the **Length** field or use the manipulators in the 3D geometry.

     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.

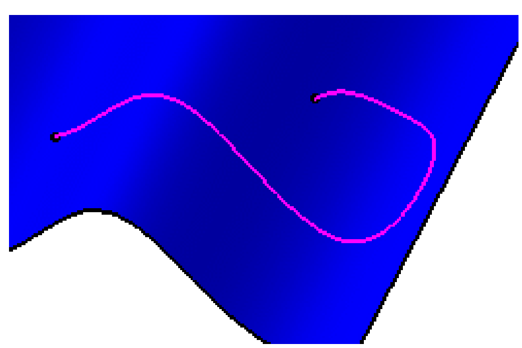
- **Up to**: the **Up to** field is enabled. Select a curve belonging to the same support as the curve to be extrapolated (surface or plane).

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.

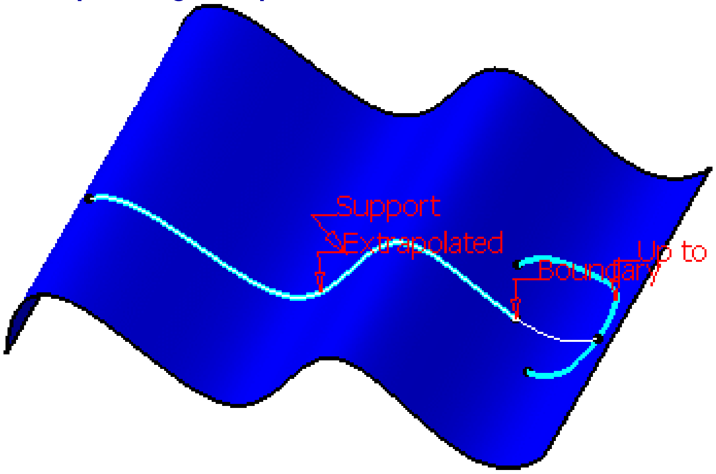
     This option is not available with the Up to type with a support.



Length extrapolation in Tangent mode

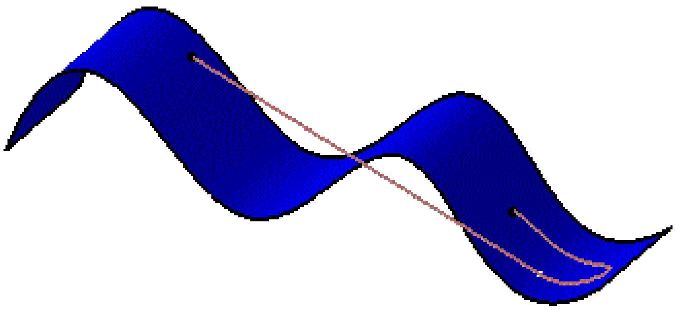


Length extrapolation in Curvature mode

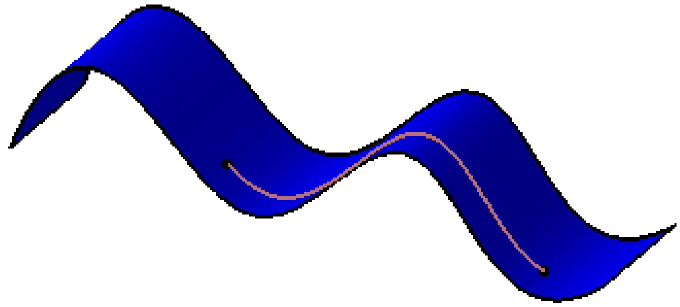


Up to extrapolation in Tangent mode

If needed and if 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.

Editing Surfaces and Wireframe Geometry

Generative Shape Design provides powerful tools for editing surfaces and wireframe geometry.

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

Select using a filter: select an object, and use the Selection Filter toolbar to manage element types and modes.

Replace elements: select the element to be replaced, choose the **Replace...** contextual command, then select the replacing element.

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.

Manage the orientation of geometry: double-click a line or a plane in the specification tree, and change the Angle value.

Move elements from a geometrical set: select the element, use the **Change Body** contextual menu, select the element before which it should be inserted

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

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

Deactivate elements: select the element to be deactivated, choose the Deactivate contextual menu and choose to deactivate its children as well, if needed.

Isolate geometric elements: select the element to be isolated, choose the xxx object -> Isolate contextual menu.

Edit parameters: click Preview while creating the element, or, if the element is already created, select it and choose the xxx object -> Edit Parameters contextual menu.

Upgrade features: select the elements to be upgraded, choose the xxx object -> Upgrade contextual menu.

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** menu item from the contextual menu
- Select the element then choose the **Edit -> xxx.object -> Definition** command
- Double-click 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.

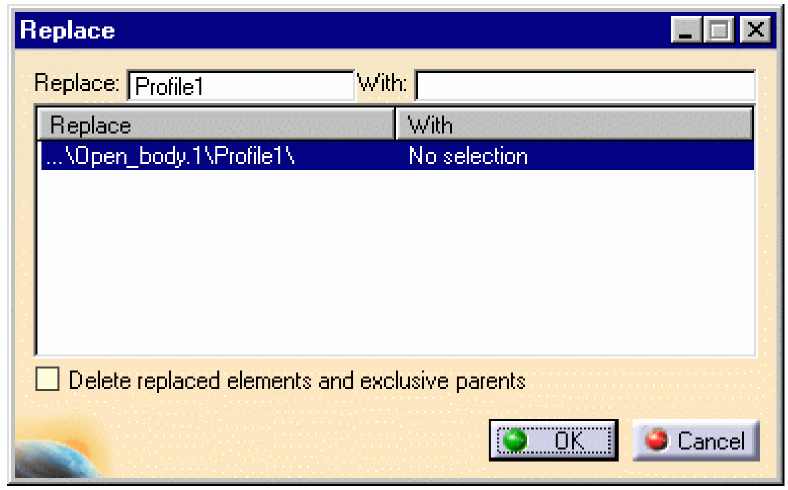
Replacing Elements

This task shows how to replace a geometric element by another.
This may be useful when a modification occurs late in the design as the whole geometry based onto the element that is replaced is updated according to the new specifications coming from the replacing elements.

Open the Replace1.CATPart document.

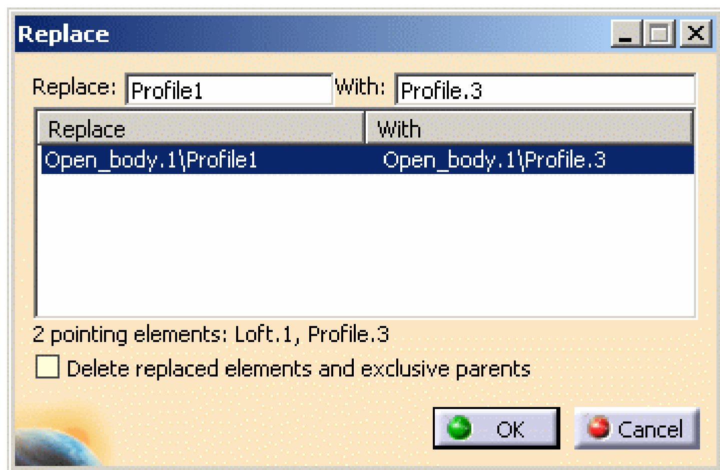
1. Right-click the pink curve (Profile.1) and choose the **Replace** contextual menu.

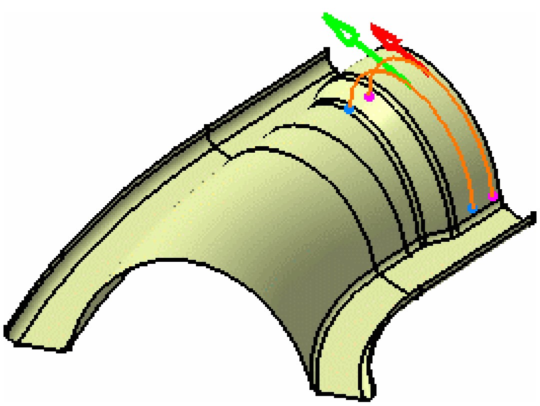
The Replace dialog box appears.



2. Click the With field and select the blue curve in the geometry.

The Replace dialog box is updated accordingly and the geometry displays the curves orientation.

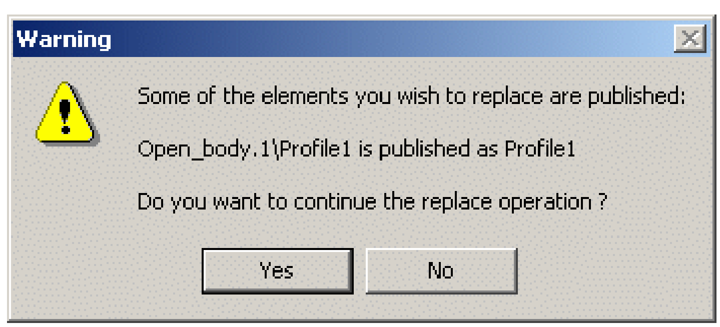




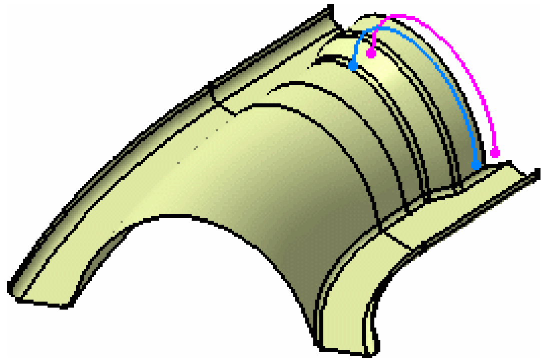
You can check the **Delete replaced elements and exclusive parents** if you do not need these elements for later operations.

A warning message is issued if the element to replace is a published element.

- Click Yes to replace the published element
- Click No to cancel the operation and close the Replace dialog box.



In our example, we published Profile 1.



3. Click **OK** to validate the replacement.

   The geometry is updated accordingly.

Creating Elements From An External File

You can create points, curves, and multi-sections 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 multi-sections 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 a Geometrical Set or an Ordered Geometrical Set.

1. Open the ElementsFromExcel.xls file from the Samples directory into Excel, and enable the macros.

The document looks like this:

	A	B	C
1	StartMulti-SectionsSurface		
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	EndMulti-SectionsSurface		
22	End		

It contains:

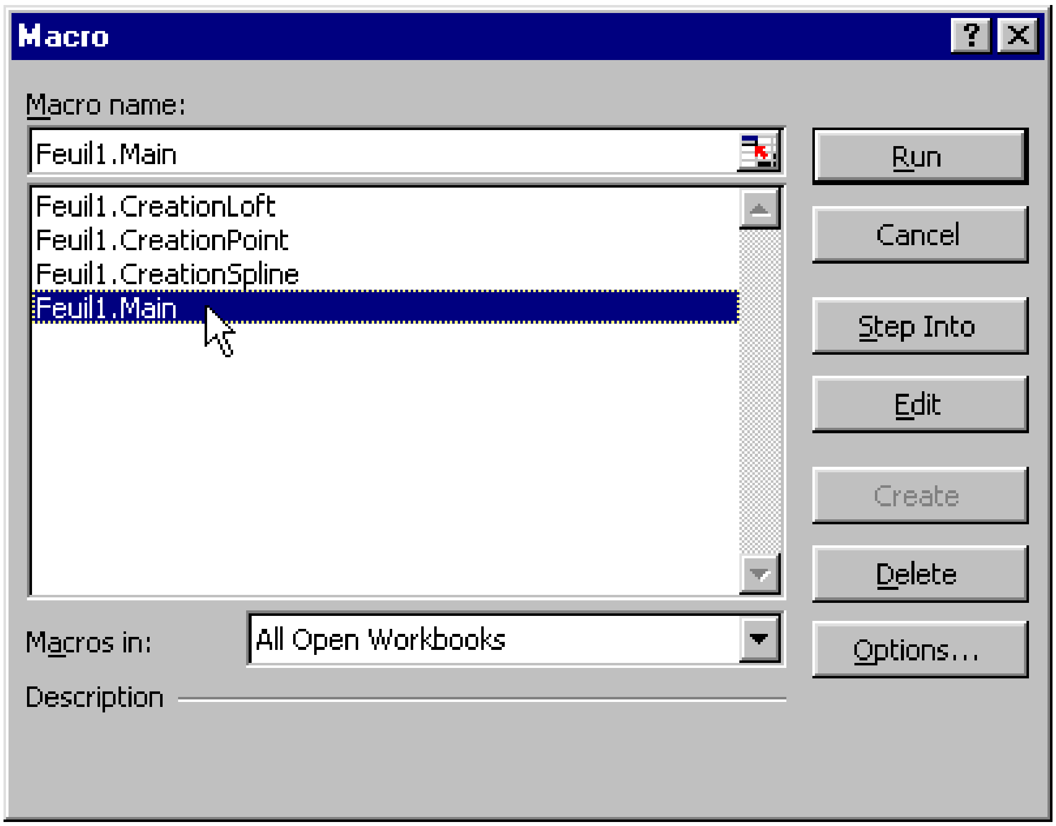
- instructions, such as StartMulti-SectionsSurface and EndMulti-SectionsSurface, StartCurve and EndCurve between which other instructions or numerical data are given.
- numerical data that are point space coordinates: X, Y, Z respectively from the left to the right
- a final End instruction

In the above example, a multi-sections surface 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.

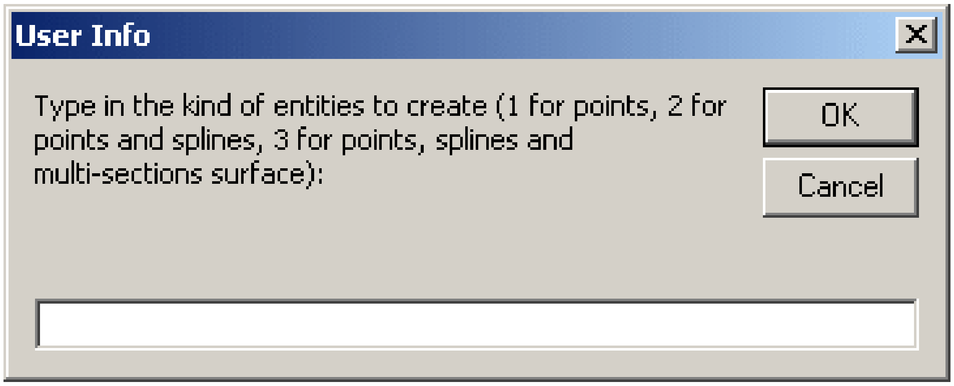
2. From Excel, select the **Tools -> Macro -> Macros** menu item.



The Macro dialog box is displayed.

3. Select the Feuil1.Main macro and click Run.

The User Info dialog box is displayed.

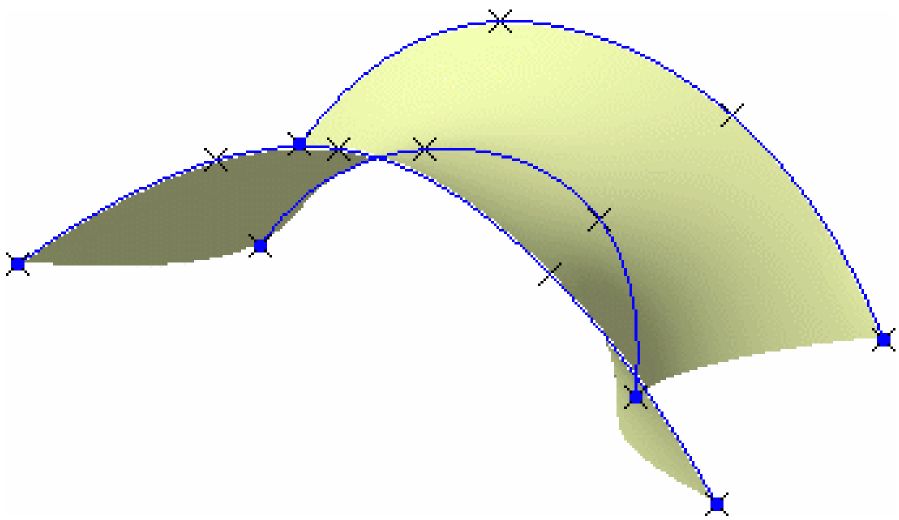


4. Key in the type of element to be generated:

- 1: to generate only the point(s)
- 2: to generate the points and the curve(s)
- 3: to generate the points, curves and multi-sections surface(s)

5. Click OK.

The elements (points, curves, and multi-sections surface) are created in the geometry. The specification tree is updated accordingly.



- The Generative Shape Design or Wireframe and Surface workbench needs not to be loaded, provided a CATIA session is running and a .CATPart document is loaded.
- The curve definition is limited to 500 points, and the multi-sections surface definition to 50 splines, with the delivered macro. This can be modified using the Excel macro edition capabilities.

Selecting Implicit Elements

There are many ways of selecting geometrical elements either in the geometry as described in the *CATIA Infrastructure User's Guide*, Selecting Objects section, or in the specification tree.

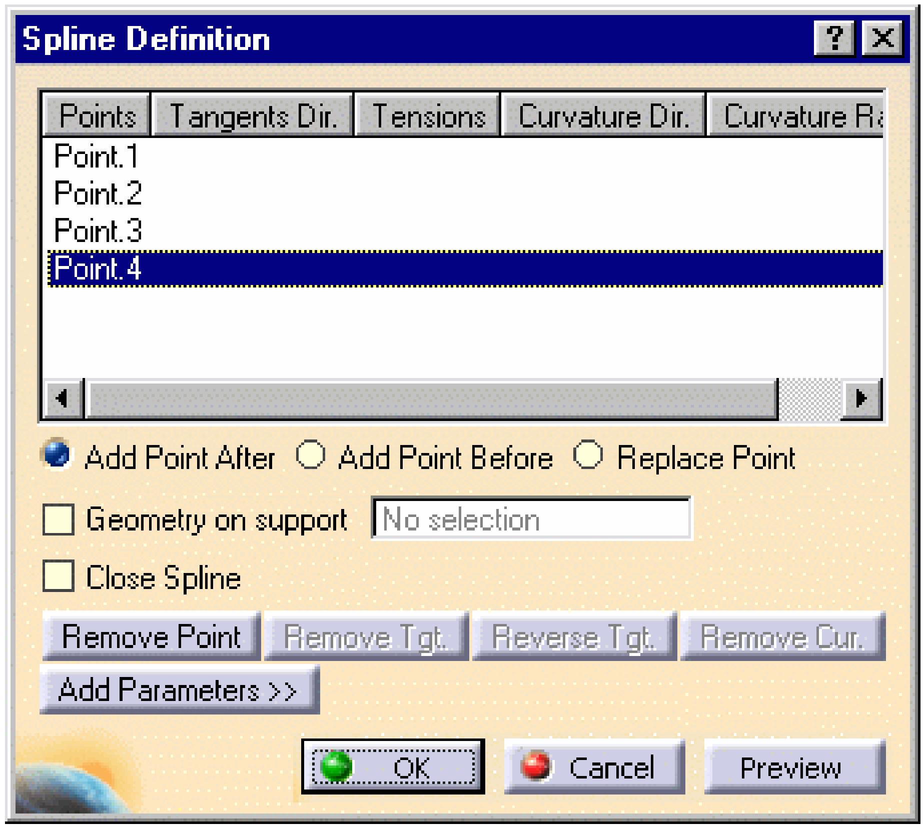
However, specific to wireframe and surface elements are some implicit elements, such as the axis of a cylinder, or the vertex of a cone for instance, participating in the creation of a feature yet not directly selectable as a separate element.

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.

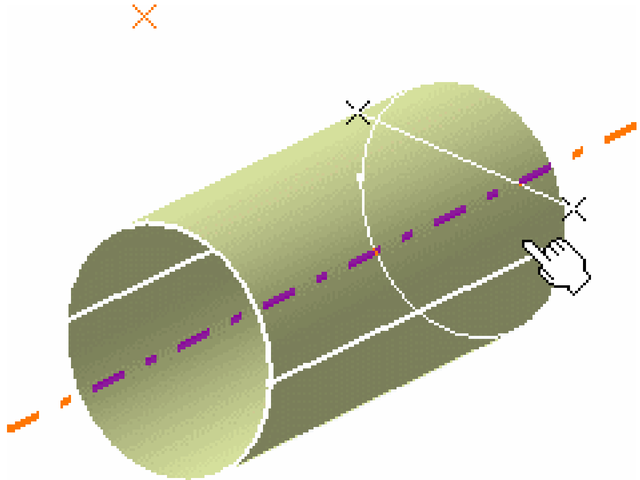
The Spline Definition dialog box looks like this:



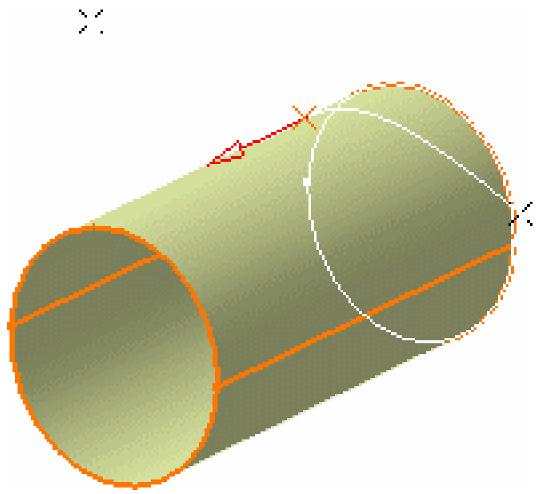
2. Select Point.3 from the list, to impose a tangency constraint on this point.

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.

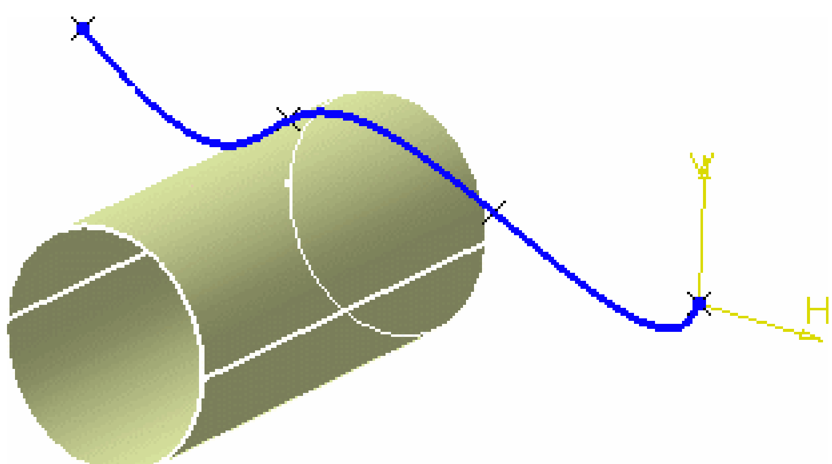


4. Click anywhere on the cylinder's surface, still holding the Shift key pressed down.



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.



Managing the Orientation of Geometry

This task shows you how to manage the orientation of modified geometry.

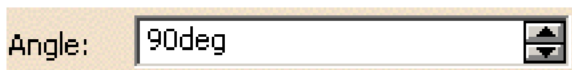
This capability is only available with the Line and Plane functionalities.

Open the Orientation1.CATPart document.

1. Double-click Line.2 in the specification tree.

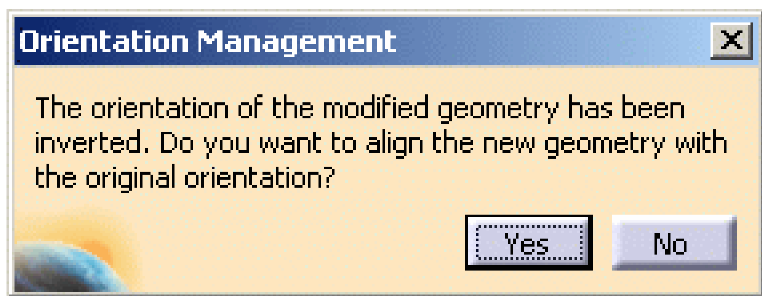
The Line dialog box is displayed.

2. Change the Angle value from 180deg to 90deg.



3. Click OK to validate the modification.

The Orientation Management dialog box is displayed.



- Click **Yes** to align the modified geometry with the original orientation.

  An inverse element is created that replaces the original line or plane (here Line.2). The inversion is proposed according to the following criteria: the normal vectors to the planes or the tangent vectors to the lines, before and after edition, have a null or negative scalar product.

- Click **No** to align the modified geometry with new orientation.

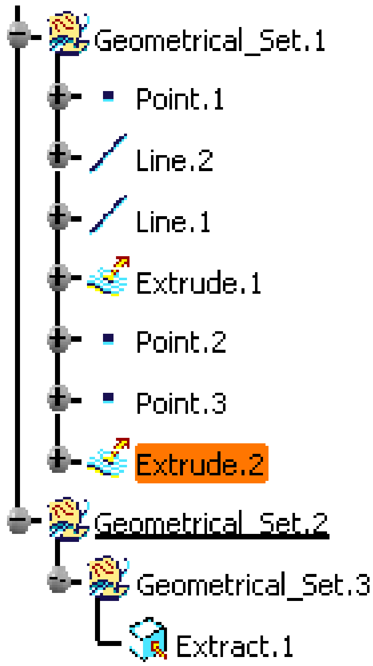
Refer to " Inverting the Orientation of Geometry" to have further information about the Inverse functionality.

Moving Elements From a Geometrical Set

This task shows how to move any element from a Geometrical Set to another location within another body (Hybrid Body or Geometrical Set).

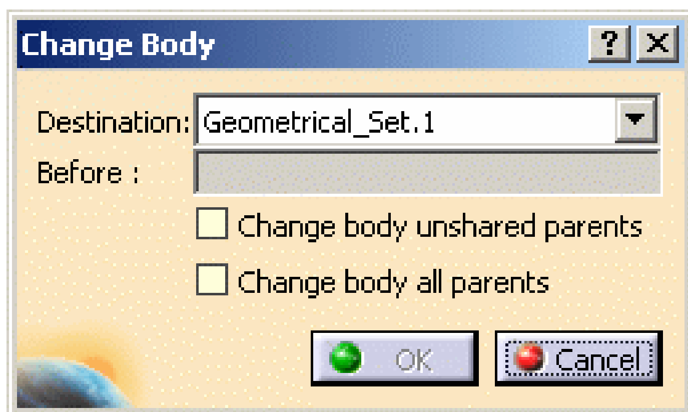
Open any .CATPart document containing several geometrical elements.
You can also open the GeometricalSets2.CATPart document.

1. From the specification tree, select the element then choose the **xxx.object -> Change Geometrical Set...** item from the contextual menu.



Multi-selection of elements of different types is supported. However, note that in this case, the contextual menu is not available, and that you can access this capability using the **Edit** menu item.

The Change Body dialog box is displayed.

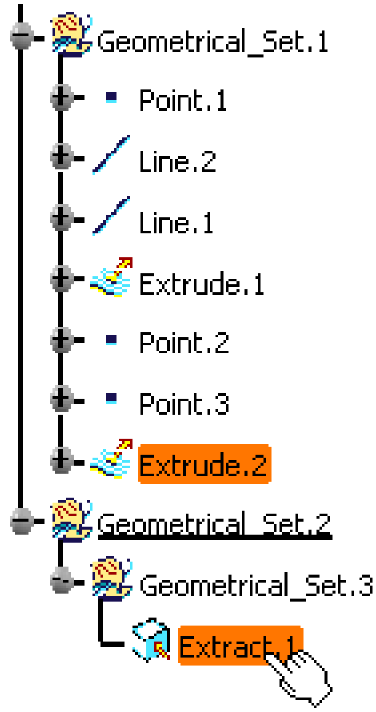


2. Select the **Destination** Body for the selected element.

   Here we selected Geometrical_Set.3.

You can do so by selecting the Body in the specification tree, or using the drop-down list from the dialog box.

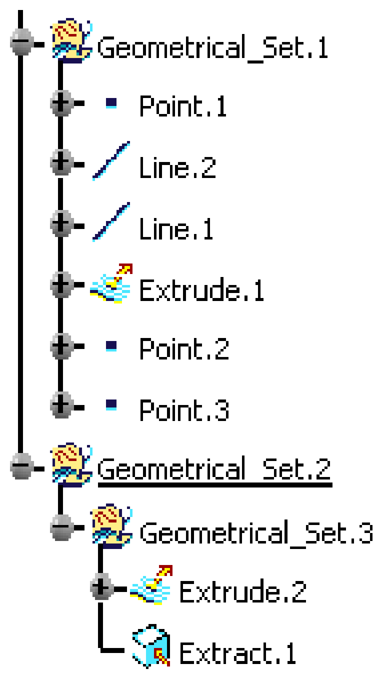
3. Select the element above which the one you already selected is to be inserted.



You can directly select this positioning element. In this case the **Destination** field of the Change Body dialog box is automatically updated with the Body to which this second element belongs.

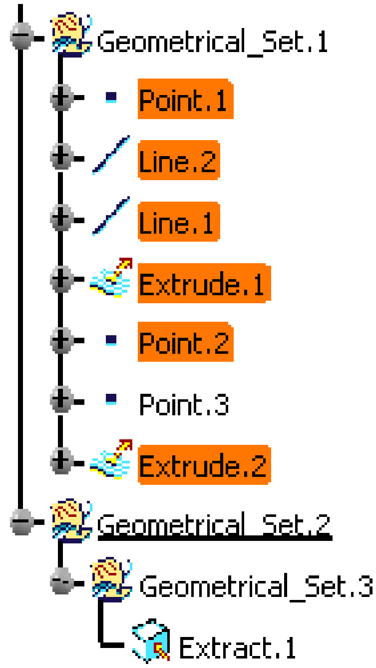
4. Click OK in the dialog box.

The element selected first is moved to its new location in the specification tree, but geometry remains unchanged.

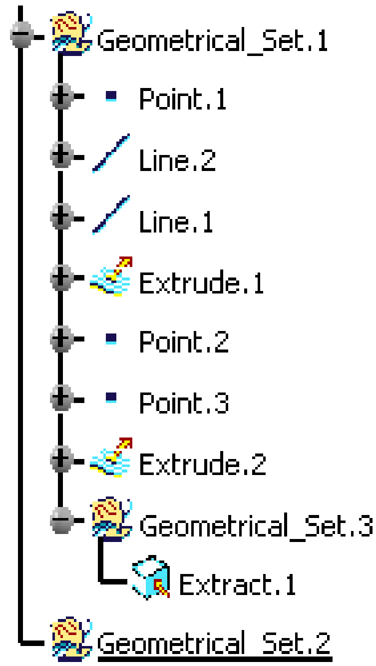


- Check the **Change body unshared parents** option to move all parents of the first selected element to its new location, provided these parents are not shared by any other element of the initial body.
  In this case, all the unshared parents are highlighted prior to the move.

- Check the **Change body all parents** option to move all parents of the first selected element to its new location, regardless of whether these parents are used (shared) by any other element of the initial body.
  In this case, all the parent elements are highlighted prior to the move.



- You can move a whole branch, i.e. a whole body and its contents, at a time.
  Here we moved Geometrical_Set.3 last in Geometrical_Set.1.



See also Managing Geometrical Sets.

Copying and Pasting

This task shows how to copy and paste geometrical set 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 Geometrical Set entity in the tree where you want to paste the selected elements.
4. Select the **Edit -> Paste** command.

The elements are copied into the target Geometrical Set.

- 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, extracts (basic and multiple edges), and fillets for example.
  In this case, you may also consider using PowerCopies.

Deleting Surfaces and Wireframe Geometry

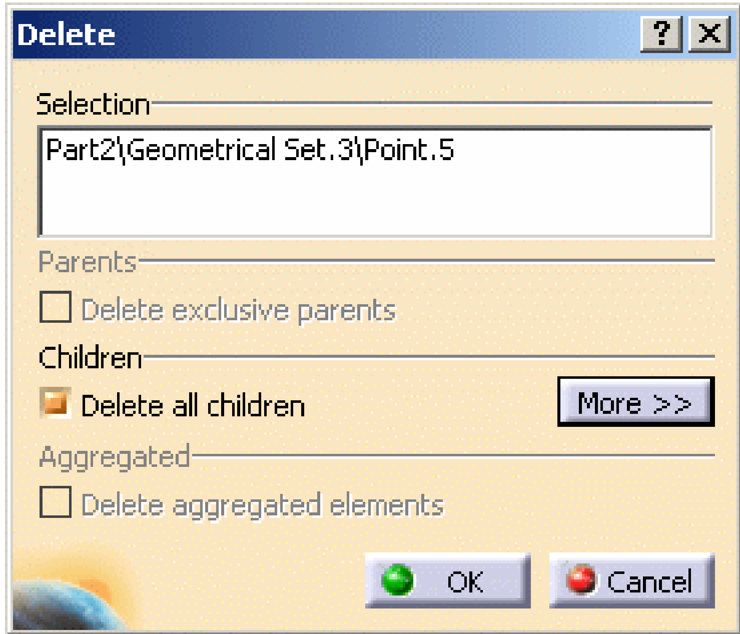
Deleting a Specific Element

This task shows how to delete geometry from your design.

Open the Delete1.CATPart document.

1. Select the entity you want to delete.
2. Select the **Delete** command either from the **Edit** menu or the contextual menu.

The Delete dialog box appears.



3. Set the desired options for managing the deletion of parent and children entities.

Two options are available:

- **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
- **Delete all children**: deletes the geometry based upon the element to be deleted, in other words, dependent elements
- **Delete aggregated elements**: deletes the geometry based upon the elements aggregated to the element to be deleted

4. Click **OK** to validate the deletion.

While deleting elements you may want to replace children of the deleted elements. To do this, click the More>> button to display the Advanced Children Management area, and refer to the Replacing Elements task as the replacement principles are the same.
For further information, refer to "Deleting Features" in the *Part Design User's Guide*.

Deleting Useless Elements

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

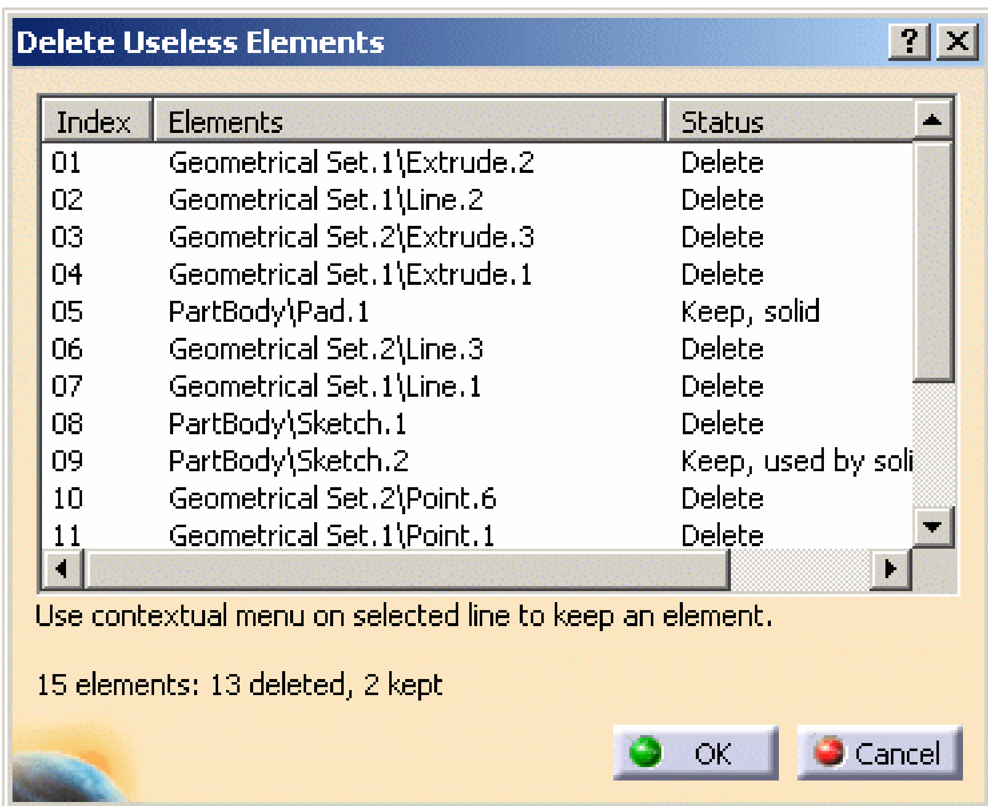
This task shows how to delete all un-referenced elements, i.e. not participating in the creation of other geometrical elements.

Please note that the command would delete all the elements of the part, if you do not wish all elements to be deleted, you have to specify it.

Open the Delete1.CATPart document.

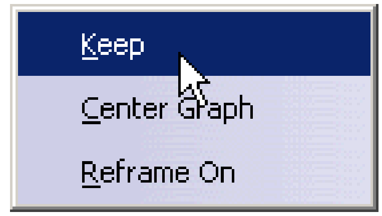
1. Choose the **Tools -> Delete useless elements** menu item.

The Delete Useless Elements dialog box appears.
It lists all the wireframe and surface elements, datum or not, that are present in the document or in other documents when working in context (in a CATProduct document referencing CATPart documents).

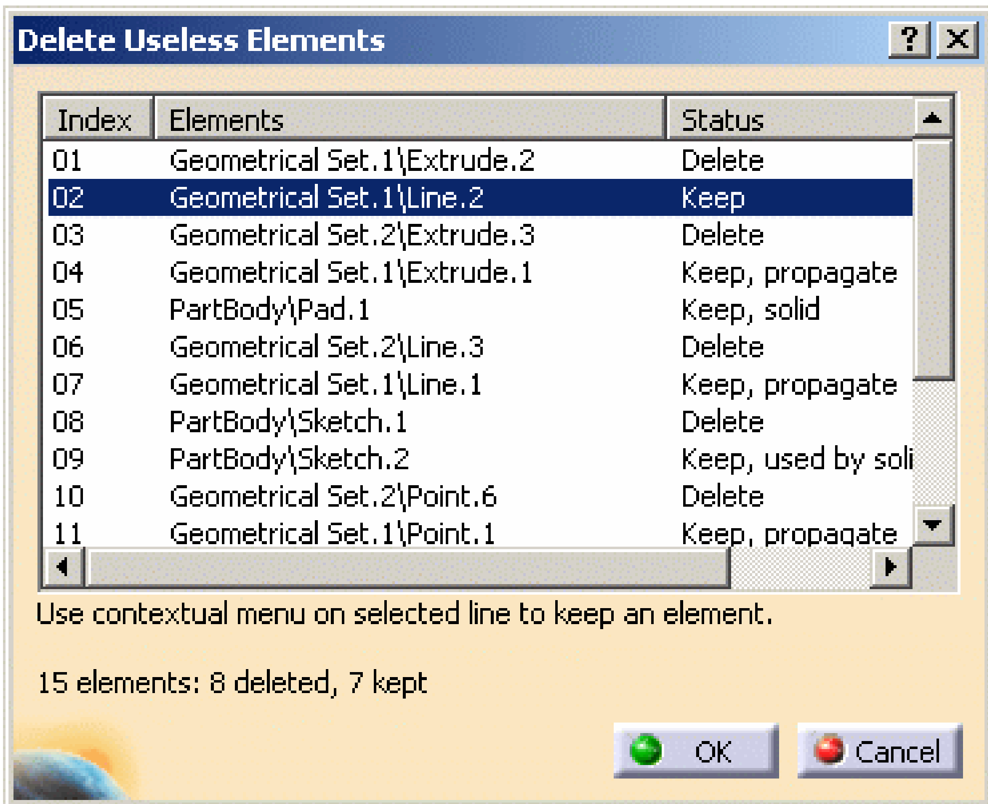


When an element is used by a Part Design feature, its status is **Keep, used by solid**, meaning it cannot be deleted.

2. Select an element you wish to keep from the list, and using the contextual menu, choose the Keep menu item.



The list of un-referenced elements is automatically updated, indicating a new status for the selected element (**Keep**) and which elements are to be kept as a consequence (**Keep, propagate** status).
In the bottom left corner of the dialog box, the global status is also updated.



3. Click OK to confirm the deletion of all elements listed with the **Delete** status.

- Use the Delete contextual menu item, to delete an element which status is either **Keep** or **Keep, propagate**.
  Also available in the contextual menu, are the **Center Graph** and **Reframe On** items.
- Bodies, whether Geometrical Sets, Ordered Geometrical Sets or PartBodies located directly below the main Part are not displayed in the list, as when creating a new document, they are necessarily empty of geometric elements, and it does not make sense to delete them.

Deactivating Elements

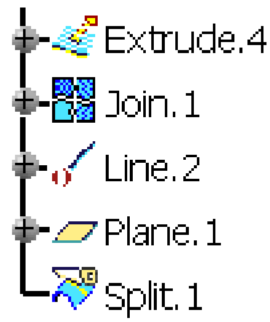
This task shows how to inactivate a geometric element.
This may be useful when, in a complex part, a branch of the part should not be affected by an update, or is not updating correctly for instance.
This capability will let you work on the other elements present in the document while ignoring a specific element.
Deactivated elements are identified by the () symbol in the specification tree. Also refer to Symbols Reflecting an Incident in the Geometry Building.

Open the Join3.CATPart document.

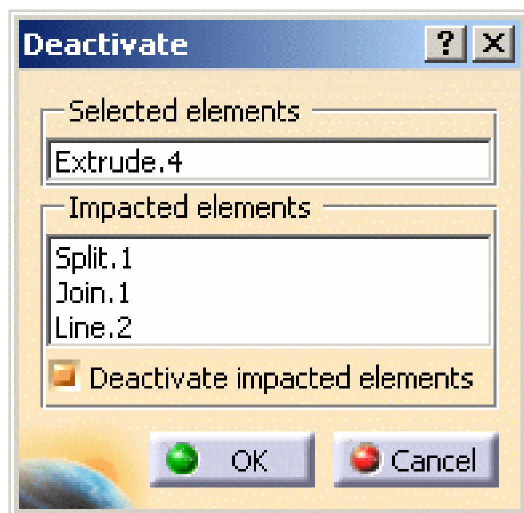
1. Right-click the element to be deactivated from the specification tree, and choose the **XXX object -> Deactivate** contextual command.

   Here we select Extrude.4.

- If the selected element does not have any children, it is directly deactivated. This is indicated by a symbol in the specification tree:



- If the selected element has children, the Deactivate dialog box appears, listing the elements to be deactivated, and their children as affected elements.



In this case, the geometry is displayed in red, as if needing an update.

There are two deactivation modes:

- **Copy mode**: the deactivation is performed on the modification operation of the feature (providing a modification of a feature of same dimension). When selected, the feature can be seen in the 3D geometry.
  Here are the features concerned by this mode:
  - Projection
  - Curve Smooth
  - Blend (with Trim option only)
  - Corner (with Trim option only)
  - Shape Fillet (with Trim option only)
  - Connect Curve (with Trim option only)
  - Parallel Curve
  - Offset
  - Variable Offset
  - Rough Offset
  - 3D Curve Offset
  - Split (on the Element to cut)
  - Trim (on the Element to cut)
  - All transformations in creation and modification modes
  - Extrapolating Surfaces (with Assemble Result option only)
  - Extrapolating Curves (with Assemble Result option only)
  - Join (copy of the first element)
  - Healing (copy of the first element)
  - Combine
  - Develop
  - Wrap Curve
  - Wrap Surface
  - Bump
  - Shape Morphing

- **Destructive mode**: the deactivation makes the feature unusable. When selected, the feature cannot be seen in the 3D geometry.
  Here are the features concerned by this mode:
  - Line
  - Plane
  - Circle
  - Reflect Line

- Spiral
- Spline
- Helix
- Intersection
- Extrude
- Revolution
- Cylinder
- Sweep (except tangent sweeps with trim option)
- Multi-Sections Surface

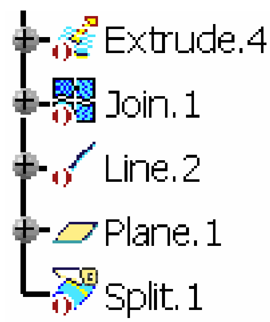
You are advised to propagate the deactivation by checking the **Deactivate impacted elements** button when it is a destructive one.

For instance, the corner using the trim option replaces the tangency curve. Therefore the deactivation is performed in copy mode. However, the corner without the trim option is a curve creation operation. Therefore, the deactivation is performed in destructive mode.

When elements are imported using multi-part links (external references) or using a Copy-Paste As result with link, the deactivation concerns the link, not the feature. As a consequence, the feature can still be selected.

2. Click OK.

The selected element and its children are deactivated.
The **( )** symbol is displayed in the specification tree, and the corresponding geometry is hidden.



It is not possible to deactivate datum elements as they do not have an history. Indeed, a deactivation would destroy their geometry and a reactivation would therefore be impossible.

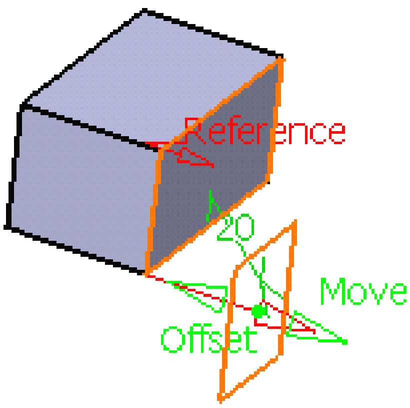
- To re-activate the elements, right-click their name in the specification tree and choose the **XXX object -> Activate** contextual command.
- Multi-selection is available, i.e. you can select several elements to be deactivated at a time. In this case the Deactivate dialog box will show a list of the selected elements.

Isolating Geometric Elements

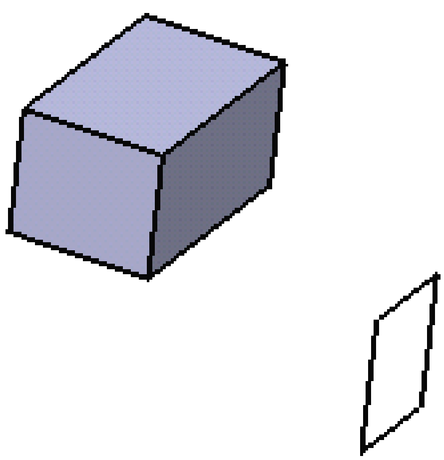
This task shows you how to isolate a geometric element, that is how to cut the links the feature has with the geometry used to create it.

To perform this task, create a plane using an offset of 20mm from a pad's face.

1. Prior to isolating the plane, note that if you edit the offset value...



...you can obtain this kind of result:



2. Right-click the plane as the element you want to isolate. The element you can isolate can be:
   - a plane
   - a line
   - a point
   - a circle

3. Select the **xxx object -> Isolate** command from the contextual menu.
   The geometrical link between the plane and the face is no longer maintained. This means that the face is no longer recognized as the reference used to create the plane, and therefore, you can no longer edit the offset value.

   The way the plane was created is ignored. You can check this by double-clicking the plane: the Plane Definition dialog box that appears indicates that the plane is of the explicit type.

   In the specification tree, the application indicates isolated elements via a red symbol in front of the geometrical element.

An isolated feature becomes a datum feature. For more information, refer to Creating Datums.

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:

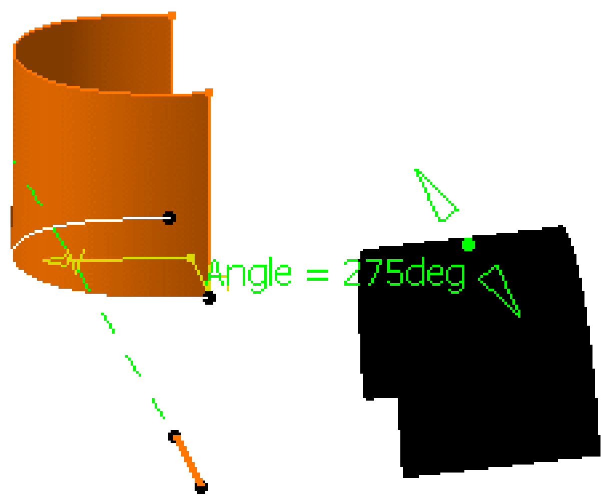
Operator	Type	Sub-Type	Parameter displayed
Bump			Length, Deformation, Distance (Maximum distance along the deformation direction from the deformed surface)
Circle	Center and Radius		Radius, Start Angle, End Angle
	Center and Point		Start Angle, End Angle
	Two Points and Radius		Radius
	Bitangent and 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 Length: Pitch Length: Height
Line	Angle/Normal to Curve		Angle
	Point-Point		

	Point-Direction Angle-Normal to Curve Tangent to Curve Normal to Surface Bisecting		Length : Start, End Infinite Start Point: End Infinite End Point: Start Infinite: /
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
	Angle and Radius		Length: Start Radius Length: End Radius
	Angle and Pitch		Length: Start Radius Length: Pitch
	Radius and Pitch		Length: Start Radius Length: End Radius Length: Pitch

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

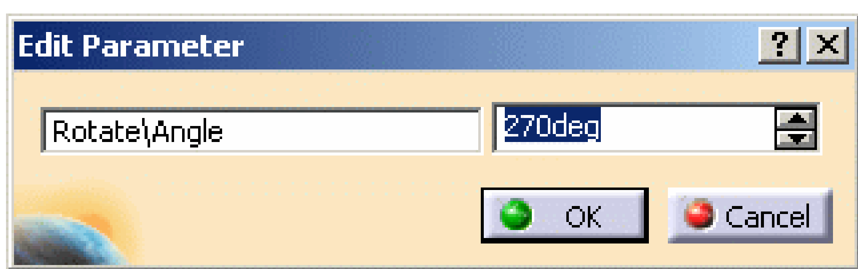
Create any of the features above.
Let's take an example by performing a rotation.

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.

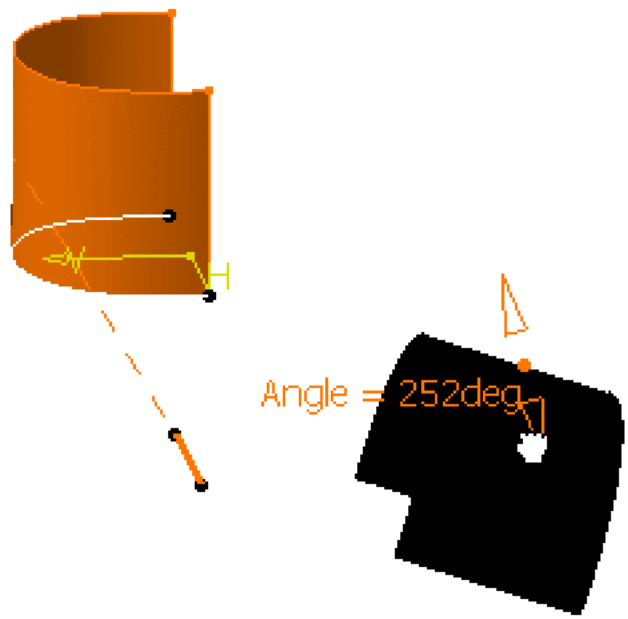
The Parameter Definition dialog box appears.



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**.

Upgrading Features

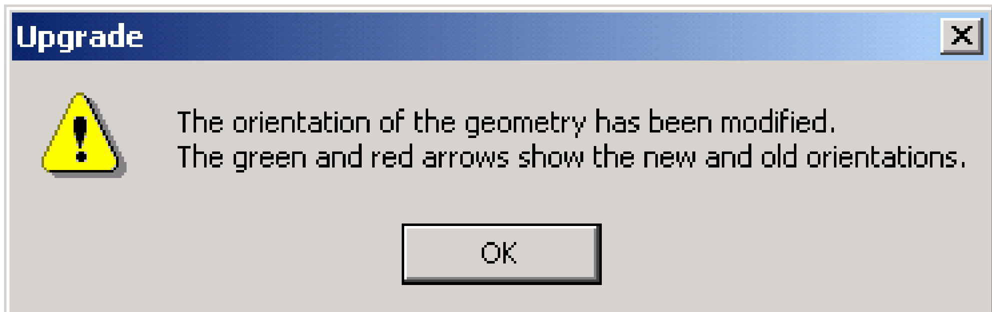
This task aims at improving the upgrade of a feature by manually upgrading it.

Open the Upgrade1.CATPart document.

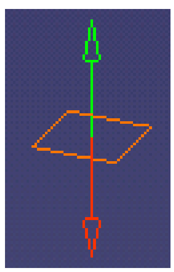
1. Right-click the feature that needs to be upgraded.
2. Select the **Upgrade** item from the contextual menu.

   In our scenario, Plane.1 needs to be upgraded.

May the orientation of the element be modified by the upgrade, a warning message is issued.
If no dialog box is displayed, it means that the orientation of the geometry is unchanged.



In the 3d geometry, a red and a green arrow show the orientation before and after the upgrade.



Upgrading a feature enables to create its 3D parameters.
See Editing Parameters.

- Note that only the algorithm linked to the topology and the geometry of the feature is upgraded.
- This capability is not available with sketches.

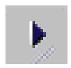
Using Tools

Generative Shape Design provides powerful tools to help you manage and analyze your surfaces and wireframe geometry.

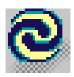
Display parents and children: select the feature under study, the **Tools -> Parent / Children...** command and use the diverse contextual commands to display parents and children.

Scan Toolbar

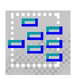

Quick selection: click the icon, and select the element.

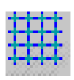

Scan the part and define local objects: Select the the **Edit -> Scan or Define in Work Object...** command, click the buttons to move from one local feature to the other, then the **Exit** button.

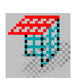
Tools Toolbar

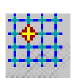

Update your design: select the element and click the icon or use the contextual menu

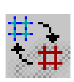

Define an axis-system: set the origin and X, Y, and Z directions

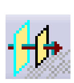

Use the historical graph: select an element, click the icon and work in the graph

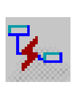

Work with a support: click the icon and select a plane or surface as support element


Work with a 3D support: click the icon and select a define the 3D support type: Reference or Local


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


Work on support activity: define the default current support to be selected when a working support is required


Create plane systems: select a plane system type, a direction, and an origin.


Create datums: click the icon to deactivate the History mode


Perform a temporary analysis: select this mode once you enter the Offset command in order to check the connections between curves or surfaces

Insert elements: click the icon to automatically insert a new element after its main parent

Keep the initial element: click either icon to retain or not the element on which you are performing an operation.

Display only the current body: click the icon to display only the features of the current geometrical set (or ordered geometrical set) and therefore greatly improve the application performances whenever you edit these features.

Instantiate PowerCopies: click this icon to manage and browse catalogs.

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.

Select bodies using the body selector: click the combo, choose a body, release the combo

Analysis Toolbar

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

Perform a draft analysis: select the surface, set the analysis mode and color range parameters, and manipulate the surface

Perform a surfacic curvature analysis: select the surface, set the analysis mode and color range parameters, and manipulate the surface

Perform a curvature analysis: select a curve or surface boundary, specify the curvature comb parameters (spikes number and length, orientation, etc.)

Apply a dress-up: set the display options then apply or remove the visualization options on selected elements.

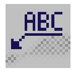
Display information on elements: click the icon and select any element

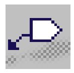
Constraints Toolbar

Create constraints: select the element to be constrained

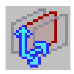

Create constraints on several elements: multi-select element and check the adequate options

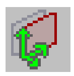
Annotations Toolbar

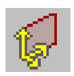

Create a text with leader: click this icon, select a face and enter your text in the dialog box


Create a flag note with leader: 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

View/Annotation Planes


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

Apply Material Toolbar


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

Miscellaneous

Apply a thickness: select a surface, the **Tools -> Thin Parts Attribute...** command and define the thickness

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


Manage groups: choose the Create Group contextual menu on a geometrical set and select the group's elements

Edit groups: choose the Edit Group contextual menu on a group

Collapse/Expand groups: choose the Collapse/Expand Group contextual menu on a group

Move groups: choose the Change Body contextual menu and select a new geometrical set

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.

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

Manage multi-result operations: select the element(s) to keep in case the result in not connex

Displaying Parents and Children

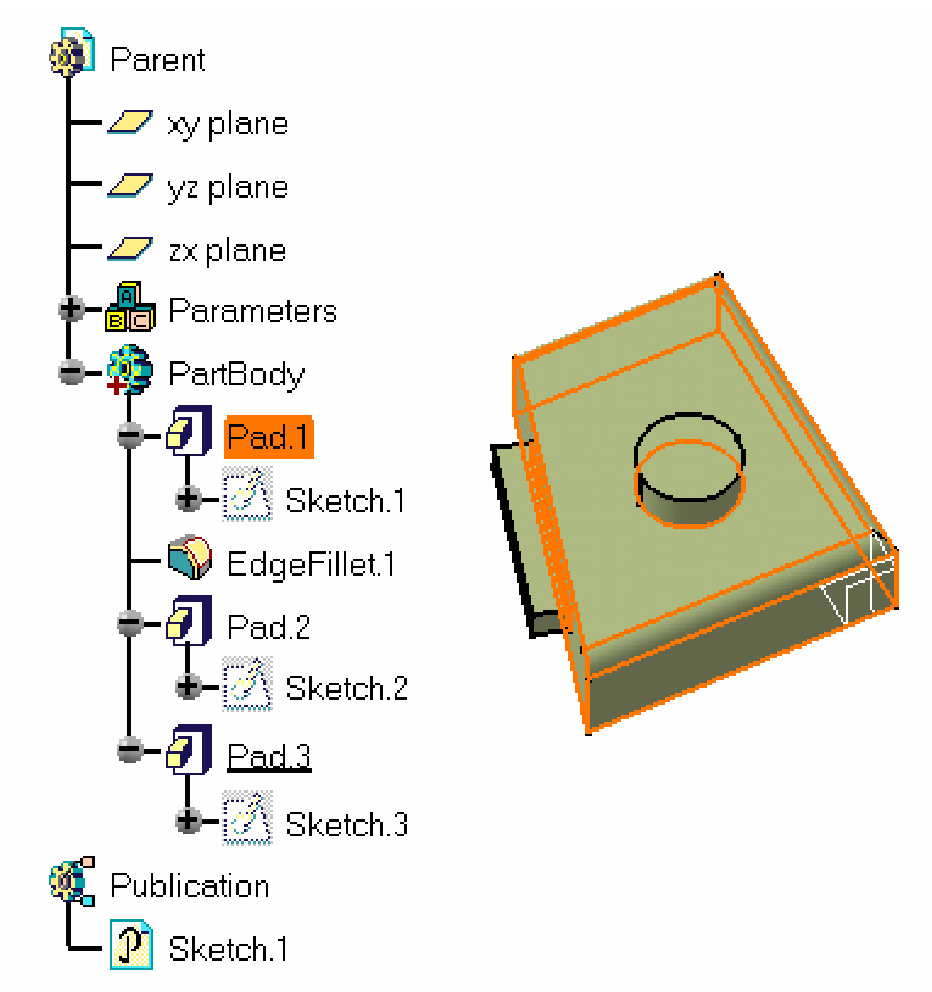
The **Parent and Children** command enables you to view the genealogical relationships between the different components of a part.

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.

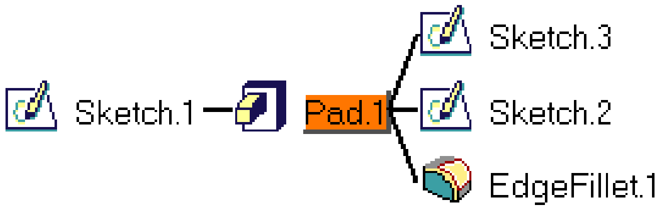
Open the Parent_R9.CATPart document.

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



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

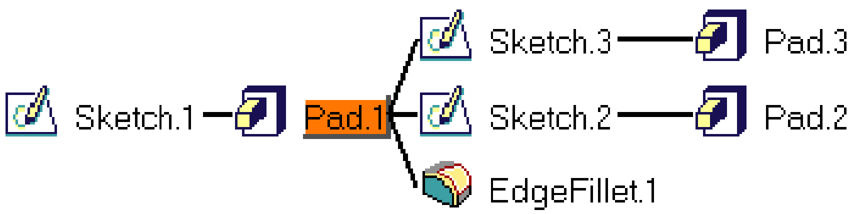
   A window appears containing a graph. This graph shows the relationships between the different elements constituting the pad previously selected.



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.



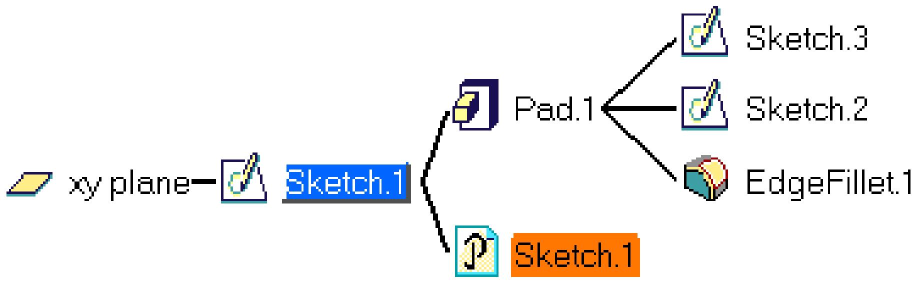
Here is the exhaustive list of the diverse contextual commands allowing you to hide parents and children. These commands may prove quite useful whenever the view is overcrowded.

- **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.

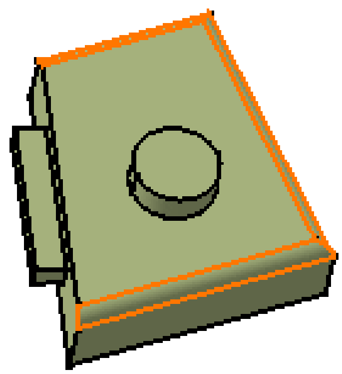
We can see that **Sketch.1** has been created on **xy plane**.

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



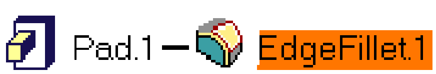
5. Now, select **EdgeFillet1** in the graph.

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.

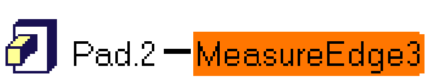


- Double-clicking on the components alternately shows or hides parents and children.
- The **Edit** contextual command can be accessed from any element. For example, right-click EdgeFillet.1 and select Edit. The Edge Fillet dialog box appears. You can then modify the fillet. When done, the Edge Fillet 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.

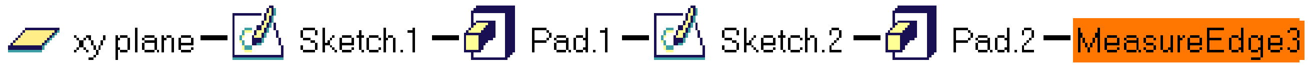
8. Select the **Tools -> Parent/Children...** command.

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.



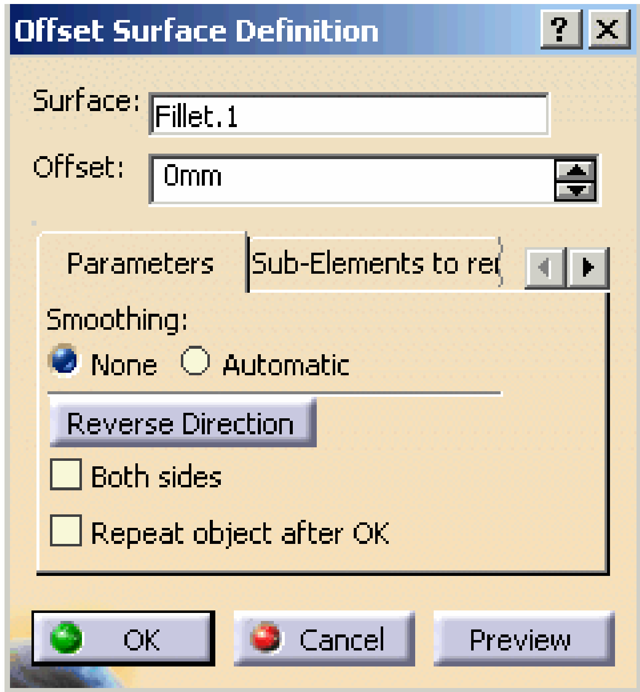
Quick Selection of Geometry

This task shows how to access rapidly to sub-elements in the geometry without scrolling in the specification tree and while already being in a command. You simply identify the generating element of the final element, without necessary trace the parents, especially if the generating element is not visible.

Open the QuickSelect1.CATPart document.

1. Click the **Offset** icon  to perform an offset of the Extrude.2 surface.

   The Offset Surface Definition dialog box appears.



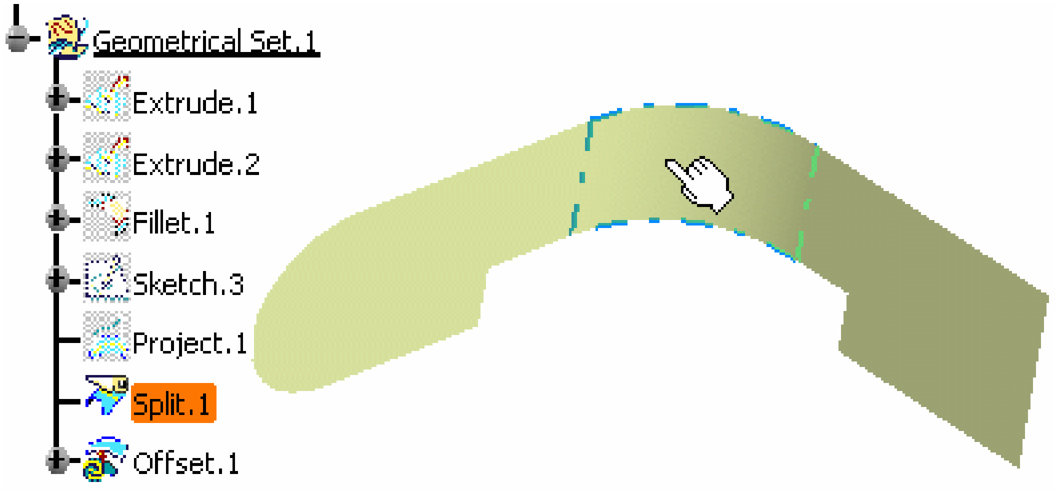
When you want to select the Extrude.2 surface as the **Surface** to offset, you notice that the root surface is not visible in the 3D geometry as it is in no show.

In order to retrieve this surface, you can use the Quick Select shared capability.

2. Click the **Quick Select**  icon.

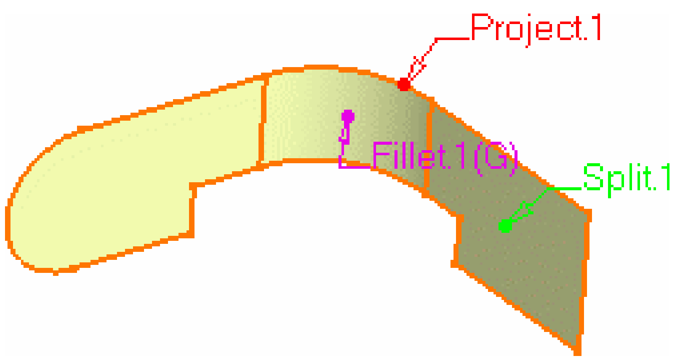
3. Move the pointer over the geometry.

   Just like in the regular selection mode, the element is highlighted in the geometry area, and the object name is highlighted in the specification tree. Moreover, the identity of the pre-selected element is displayed in the status bar:

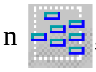


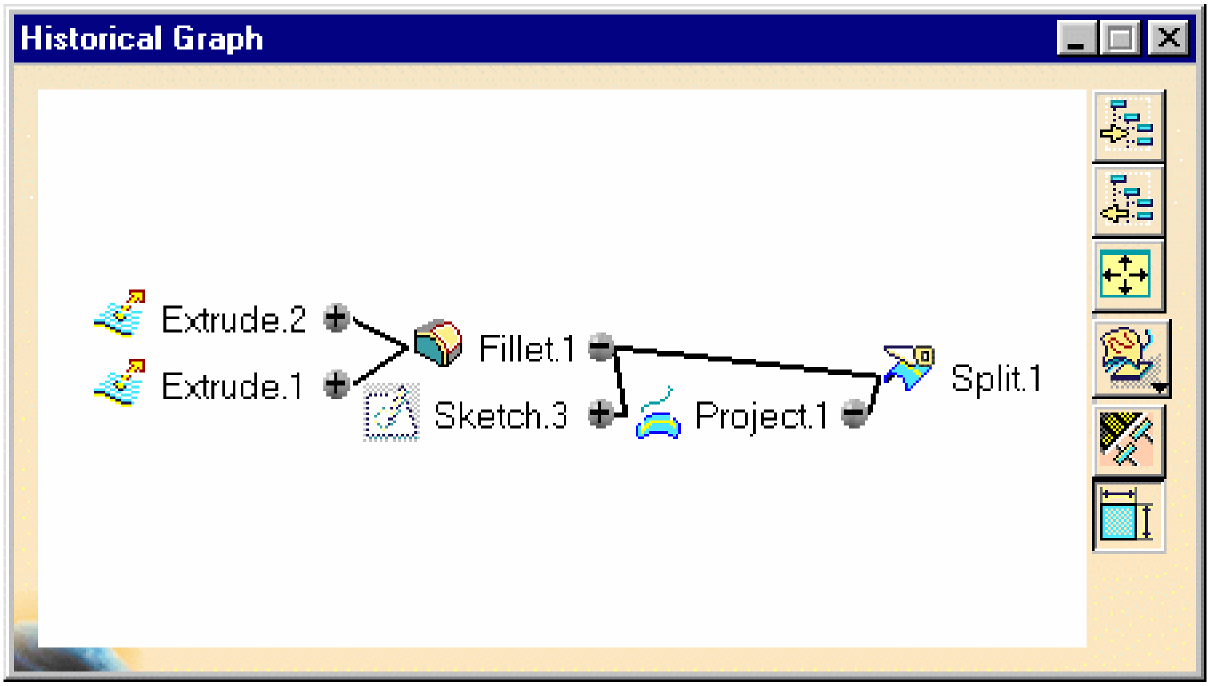
4. Click the element (Split.1).

Information is displayed on the whole geometry:



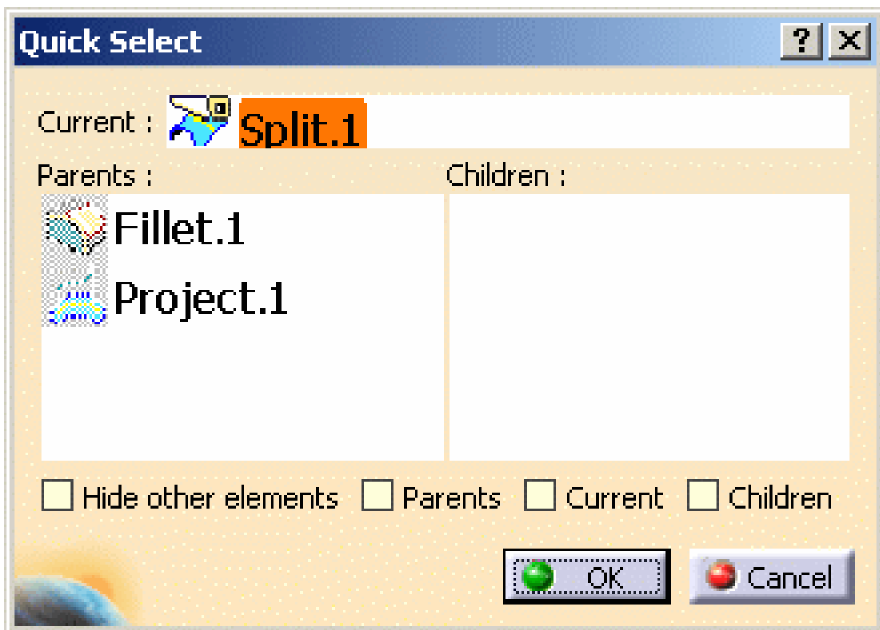
- in **green**: the feature selected using the standard selection tool
- in **red**: its direct parents
- in **purple**: the "generating" element, that is the feature generating the underlying surface/curve where you initially selected the element.

If you display the element's graph using the the **Show Historical Graph** icon , you can better relate the elements to its "parents:

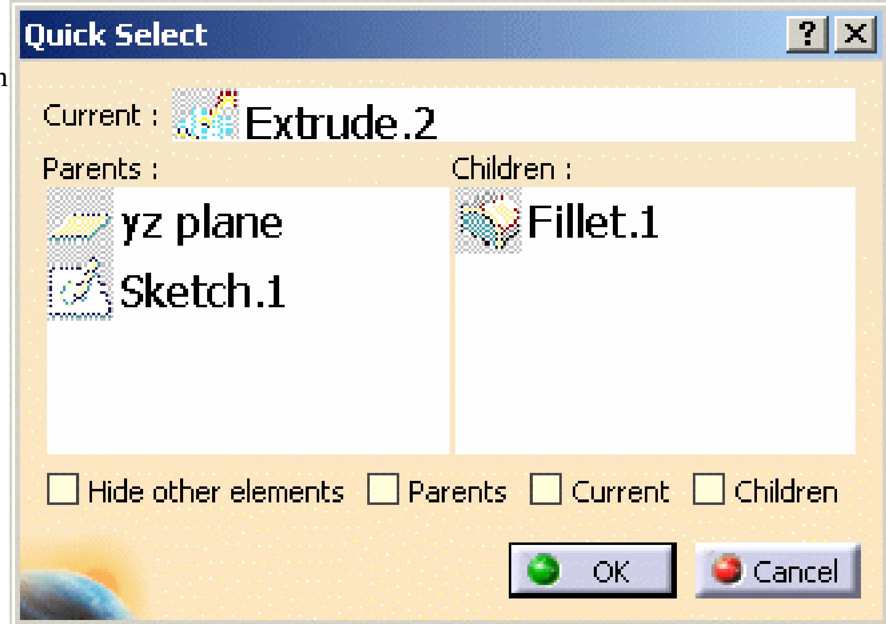


- The **Split.1** is the last generated element, to the left of the graph, and is displayed in green.
- The Project.1 is its direct parent, as shown in the graph and is displayed in red.
- The Fillet.1 is another direct parent, but is also the generating one, as it is the first element that unite other independent elements (the extruded surfaces) that lead to the creation of the split. Therefore it is displayed in purple, the precedence being given to the generating element over the direct parent.

Along with the information onto the geometry, the Quick Select dialog box is displayed: it indicates which element has been selected, as well as its parents, and children where applicable.



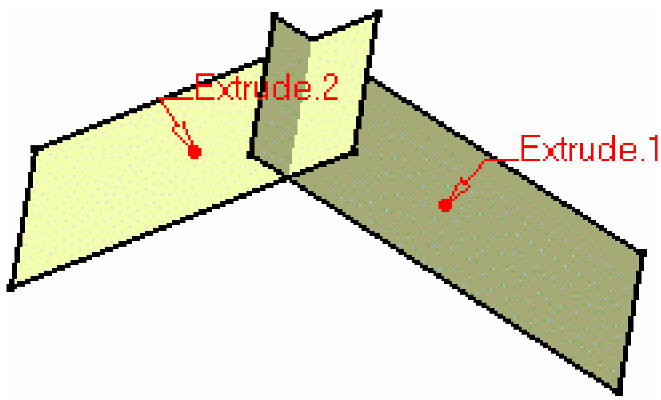
Within the dialog box, navigate in the Parents / Children graph in order to retrieve the root surface: select the Fillet.1 element as Quick Select, then the Extrude.2 element. The latter is set as the current element.



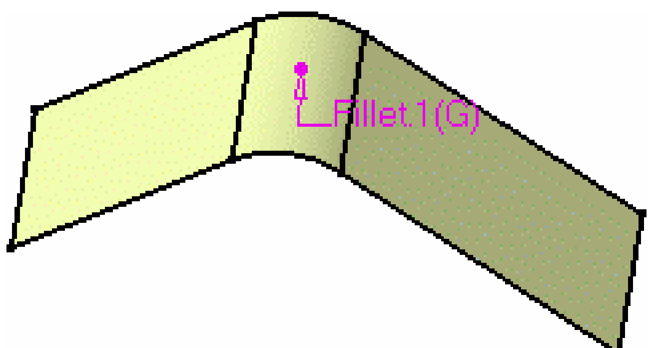
The Quick Select dialog box is updated accordingly:

The contextual menu is available on the current element, displaying standard commands such as Reframe On, Delete, Replace, etc.

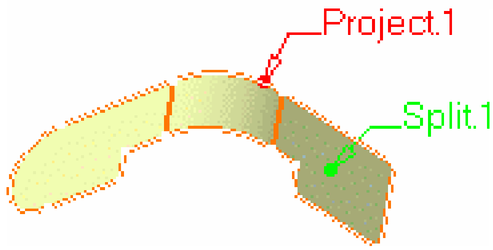
5. Use the check buttons at the bottom of the dialog box to display or hide a number of elements in the geometry.



- If you check the **Hide other elements** and the **Parents** button, you see this:



- If you check the **Hide other elements** and **Current** (i.e. the only filleted surface) buttons, you see this:

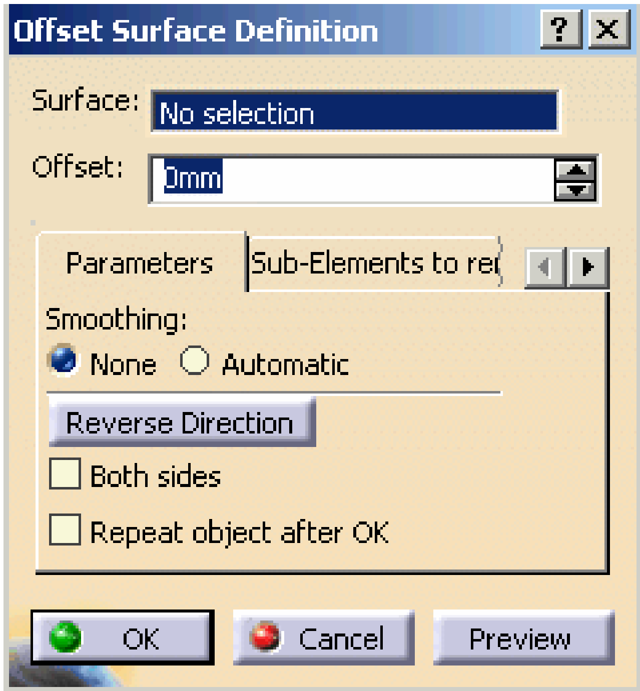


- If you check **Children** button only (i.e. the projection and the split), you see this:

6. Click OK in the dialog box.

The Quick Select dialog box closes and you return to the Offset Surface Definition dialog box.

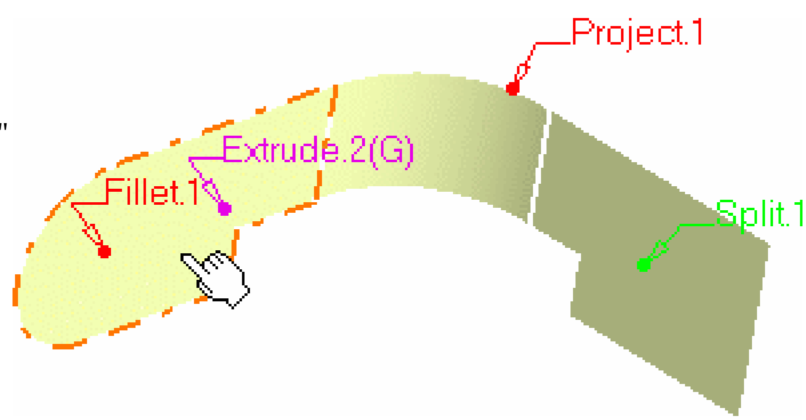
The **Surface** field is valuated with the Fillet.1 surface you previously selected.



7. Specify the **Offset**.

8. Click OK to create the offset surface.

- You can double-click on any arrow, not necessarily the generating parent as shown above, to edit any of the elements.
- You can also edit any of the elements by using the contextual menu available on all elements from the Quick Select dialog box, as well as from the texts in the geometry.
- You can select another "final" element directly in the geometry, without having to reselect the **Quick Select**  icon.



- Click in space to deselect any geometry and reset the quick selection without deselecting the icon.

Scanning a Part and Defining In Work Objects

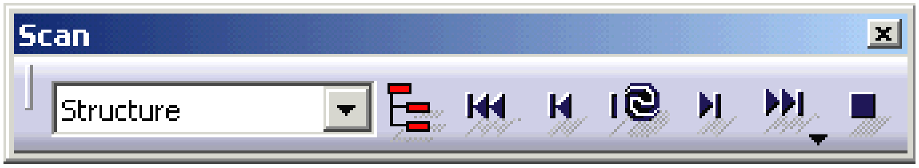
This task shows how to scan the part and define a current object without taking the complete part into account. Therefore, it is useful for the analysis of the better understanding of the part design.
Both geometrical sets and ordered geometrical sets can be scanned.

Open the Scan1.CATPart document.

1. Select the **Edit -> Scan or Define in Work Object...** command or click the icon from the Select toolbar.

The Scan toolbar appears enabling you to navigate through the structure of your part. Moreover, the part can be updated feature by feature.

You actually need to click the buttons allowing you to move from one current feature to the other. Sketch elements are not taken into account by the command.

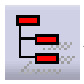


2. Select the Scan mode to define the way of scanning:

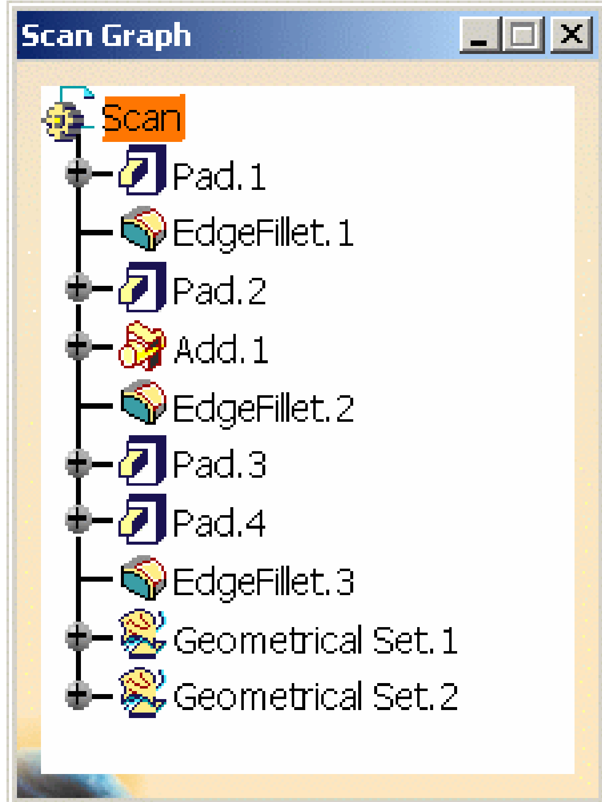
Structure

All features of the part are now scanned in the order of display in the specification tree.
The current position in the graph corresponds to the in work object.

Internal elements of sketches, part bodies and bodies, ordered geometrical sets, and elements belonging to a geometrical set are not taken into account by this mode.

1. Click the **Display Graph** icon .

The Scan Graph dialog box appears and displays all the features belonging to Scan1 part.



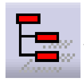
Update

All features of a part are scanned in the order of the update (which is not necessarily the order of the specification tree).
The current position in the scan graph does not correspond to the in work object: indeed the underlined object in the graph is not necessarily the one underlined in the specification tree.

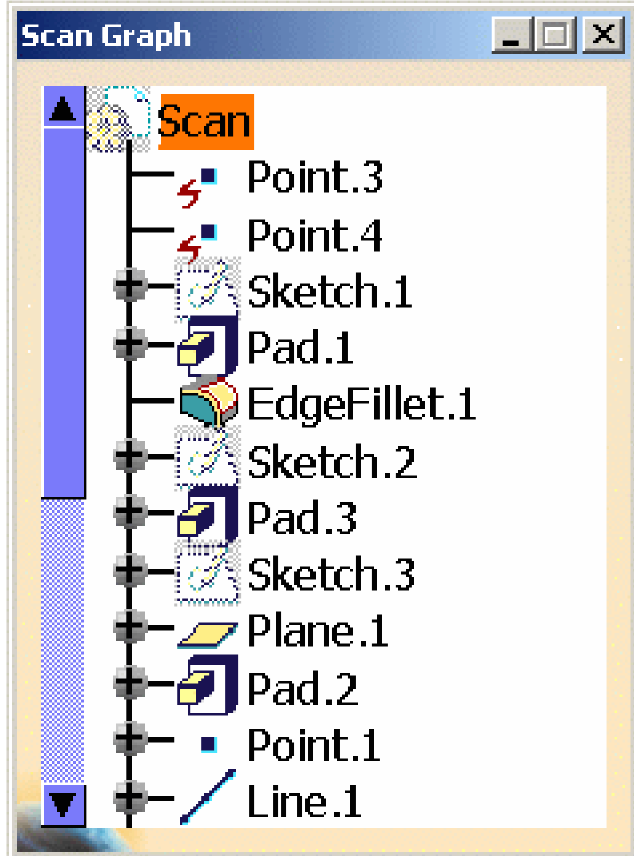
- Datum features appear first; geometrical sets, ordered geometrical sets and deactivated features do not appear in the Scan Graph.
- The part is put in no show, so is its 3D display, in order to build a new 3D display that contains the same features but in a different order.

Deactivated features appear in the Scan Graph.

As a consequence, if a geometrical set or an ordered geometrical set is in no show, it is ignored and its elements are considered as being in show.
To put the contents of this geometrical set or ordered geometrical set in no show, use the **Geometrical_Set.x object -> Hide components** contextual command.
Refer to the Hiding/Showing Geometrical Sets or Ordered Geometrical Sets and Their Contents chapter for further information.

1. Click the **Display Graph** icon .

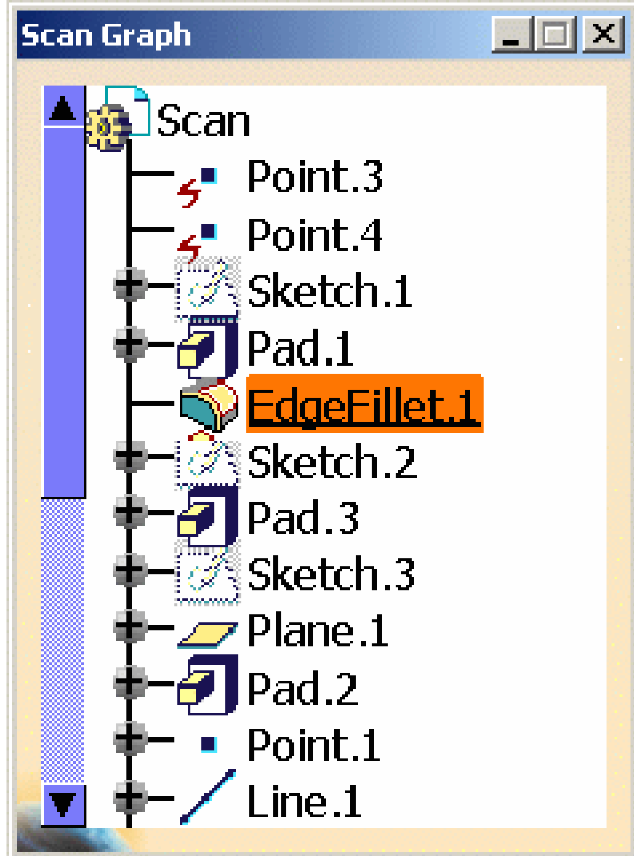
The Scan Graph dialog box appears and displays all the features belonging to Scan1 part.



3. Select a feature in the Scan Graph or in the specification tree. The application highlights the feature in question in the specification tree as well as in the geometry area and make it current.

   In our example, we chose EdgeFillet.1.

- A preview of the current object's parents is available: surfaces appear as transparent and edges as dotted yellow lines.
- If a parent of the in work object is in no show, it is temporarily shown when its child is the in work object.



4. Click the **Previous** arrow to move to the previous feature, that is Pad.1

5. Click the **First** arrow to move to the first feature, that is Point.4 (the last datum point).

In case there are several datum features, the application highlights the last one as there are all scanned at the same time.

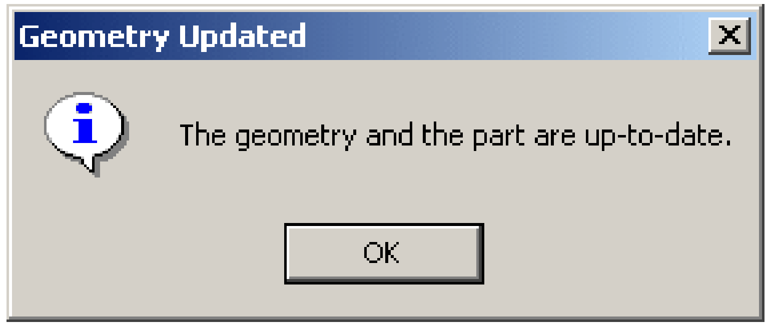
6. Click the **Next** arrow to move to the next feature, that is Sketch.1.

Scanning **Next** and **Previous** skip datum and deactivated features.

7. Click the **Last** arrow to move to the last feature, that is Point.2.

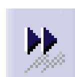
Moving to the next or last feature enables to update elements that are not up-to-date.

8. Click the **First to Update** icon to move to the first element to be updated and consequently update it.

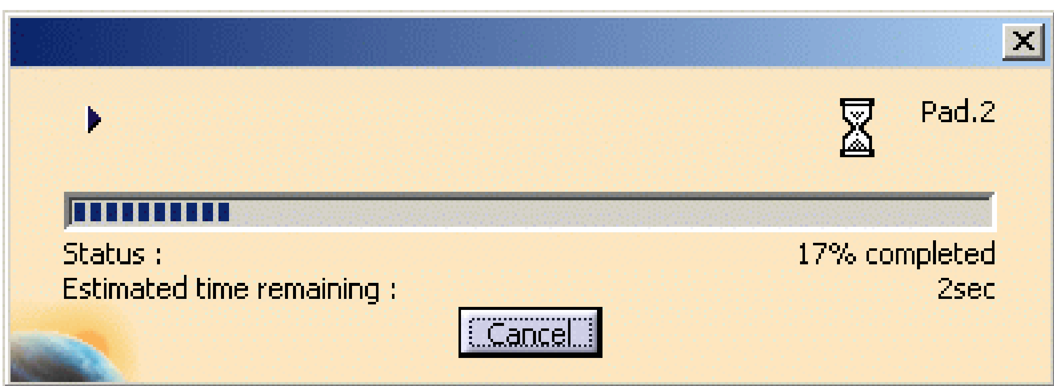


If both geometry and part are up-to-date, an information panel appears.

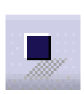
9. Click this icon again to find the next element to be updated and so on until an information panel appears to inform you that both geometry and part are up-to-date.

10. Click the **Play Update** icon  to replay the update of the geometry.

A progression bar is displayed, while the scenario is being replayed.



In case there are update errors, the replay stops at the first error. The Update Error dialog box opens.

11. Click the Exit button  to exit the command.

In the geometry area and the specification tree, the application highlights the current object.
If the object was in no show, it is put in show as long as it stays current.

Defining a feature as current without scanning the whole part is possible using the **Define in Work Object** contextual command on the desired feature. This feature is put in show if needed, and keeps its status even if another feature is defined as the in work object.

When clicking a sub-element in the 3D geometry, it is in fact the feature used to generate this sub-element which is selected as the in work object. Likewise, this feature is edited when double-clicking a sub-element.

To display 3D parameters attached to Part Design features, check the **Parameters of features and constraints** option in the **Tools -> Options -> Infrastructure -> Part Infrastructure -> Display**.

Updating Your Design

This task explains how and when you should update your design.

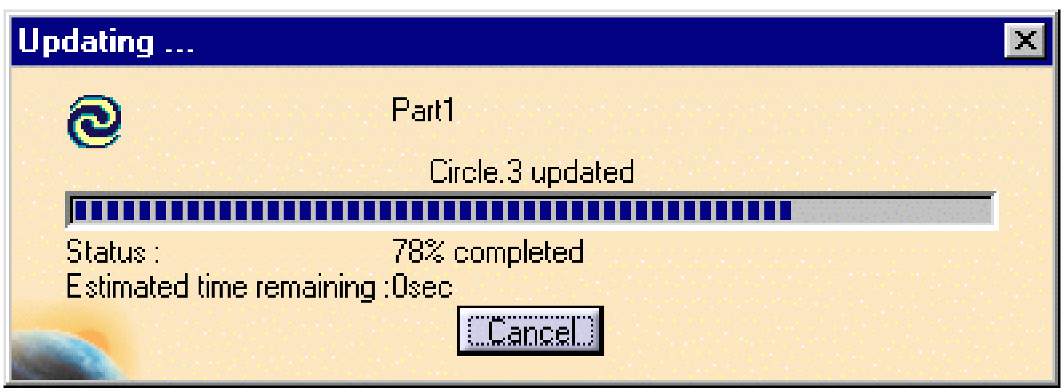
The point of updating your design is to make the application take your last operation into account. Indeed some changes to geometry or a constraint may require rebuilding the part. To warn you that an update is needed, CATIA displays the update symbol next to the part name and displays the corresponding geometry in bright red.
To update a part, the application provides two update modes:

- **automatic update**, available in **Tools -> Options -> Mechanical Design -> Assembly Design -> General** tab. If checked, this option lets the application update the part when needed.
- **manual update**, available in **Tools -> Options -> Mechanical Design -> Assembly Design -> 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.

Non-updated wireframe and surface elements are displayed in red.

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 -> Infrastructure -> Part Infrastructure**, **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

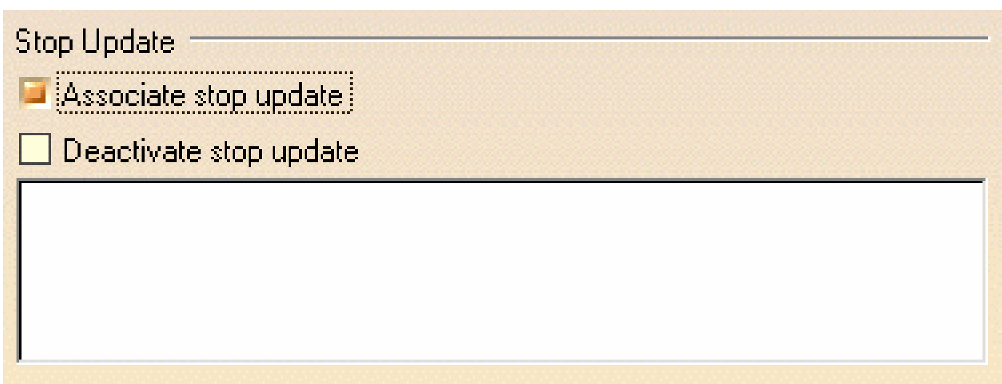
This task explains how to update a part and interrupt the update operation on a given feature by means of a useful message you previously defined.

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.

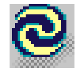
2. From the **Mechanical** tab, check the **Associate stop update** option.



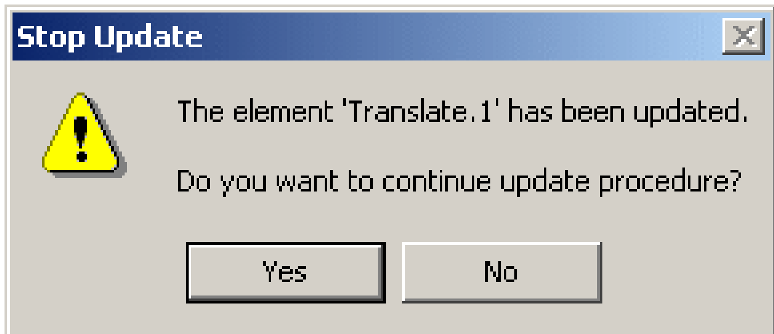
3. Enter the text to be displayed when the updating process will stop when reaching this element.

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.

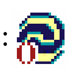
5. Whenever it is needed, click the **Update** icon  to update the whole part.

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.

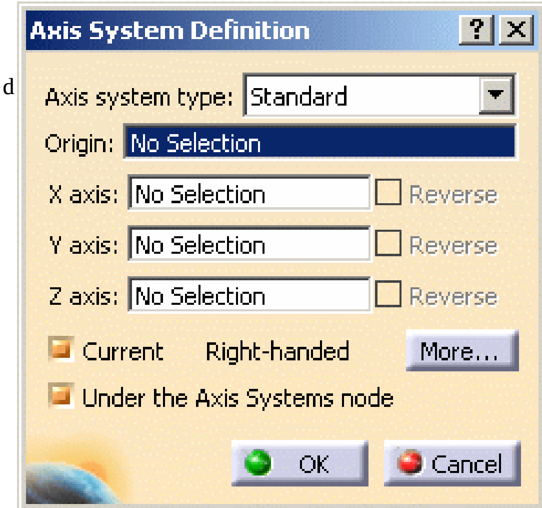
Defining an Axis System

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.

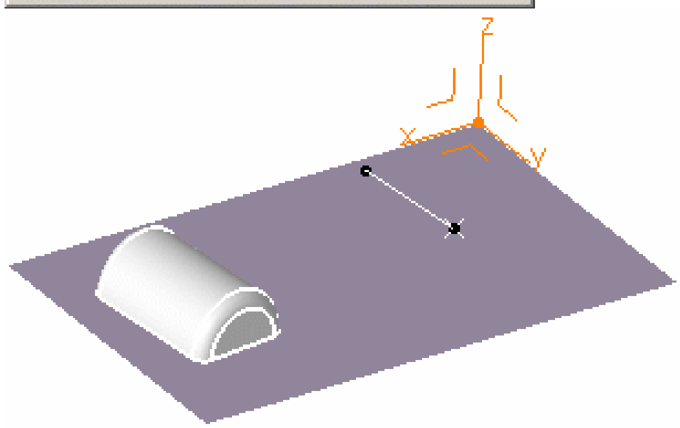
Open the PowerCopyStart1.CATPart document.

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. For instance, you can start by selecting the vertex 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:



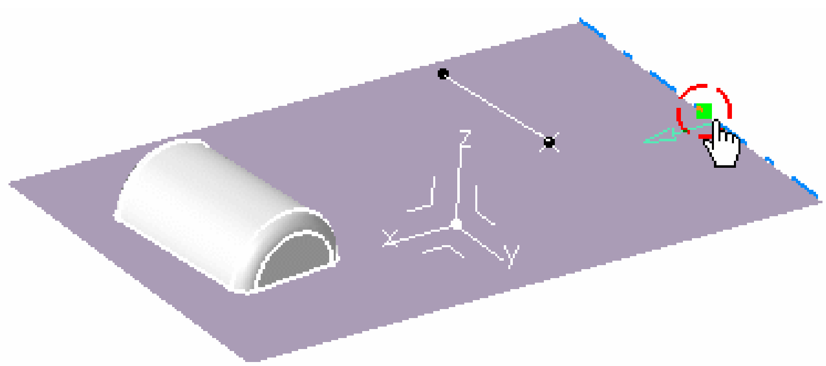
It can be right or left-handed. This information is displayed within the Axis System Definition dialog box.

You can choose from different types of axis system:

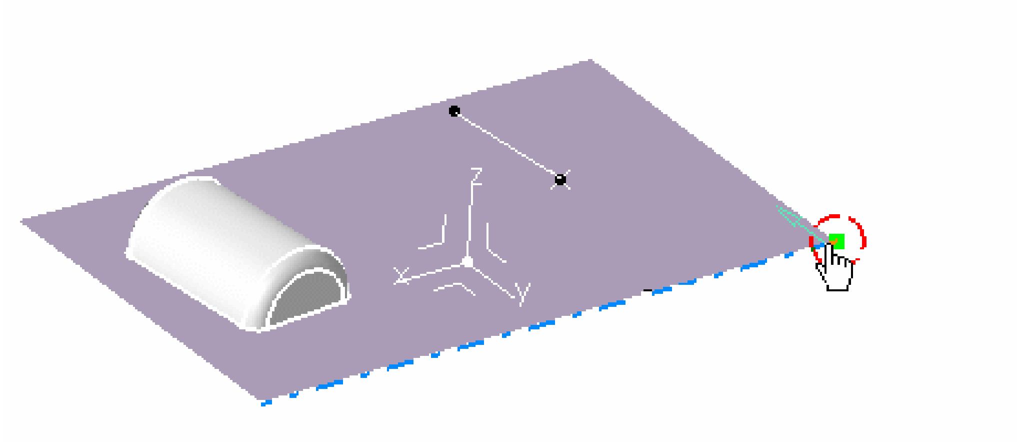
Origin

Instead of selecting the geometry to define the origin point, you can use one of the following contextual commands available from the Origin field:

- **Create Point**: for more information, refer to Points
- **Coordinates**: for more information, refer to Points
- **Create Midpoint**: the origin point is the midpoint detected by the application after selection of a geometrical element.



- **Create Endpoint**: the origin point is the endpoint detected by the application after selection of a geometrical element

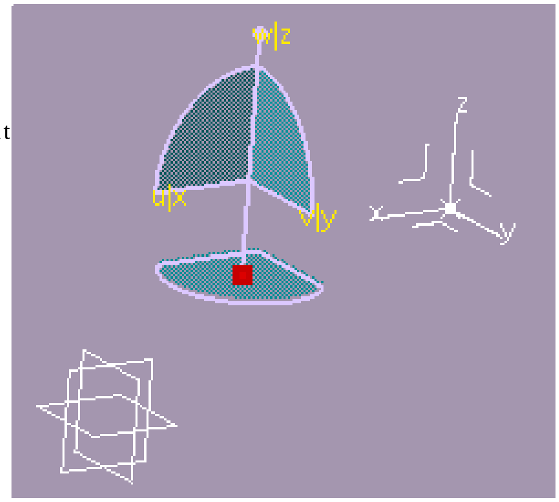


Axis System Types

You can choose from different types of axis systems:

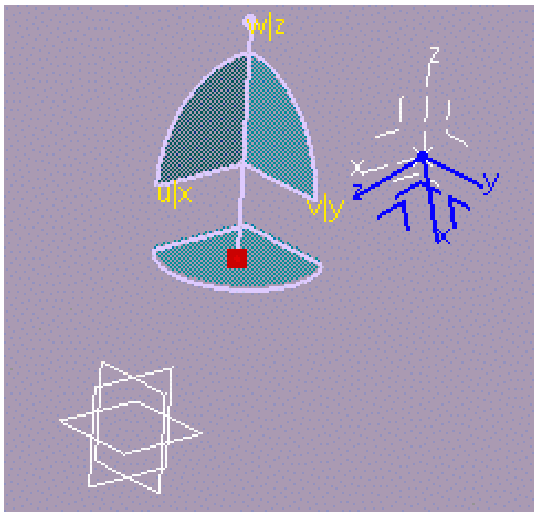
- Standard: defined by a point of origin and three orthogonal directions (by default the current directions of the compass)

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.

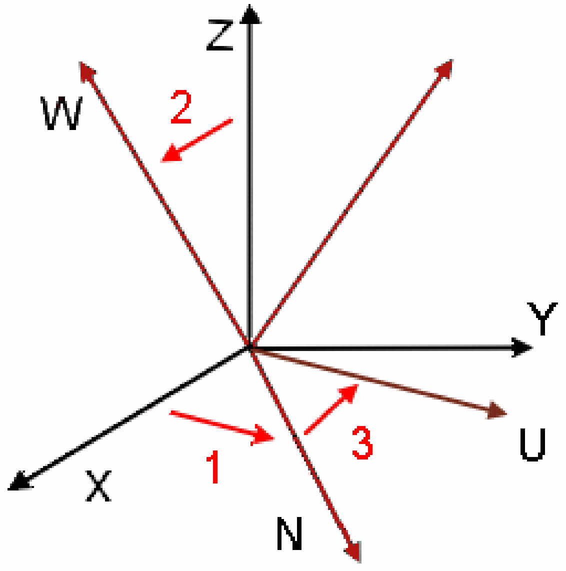


- Euler angles: defined by three angle values as follows:

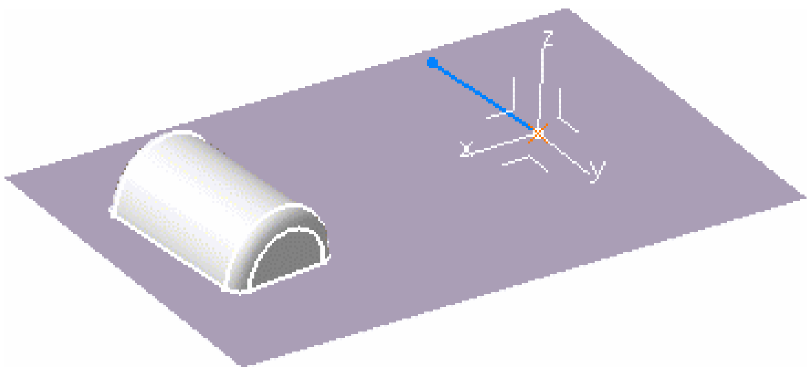
Angle 1 = (X, N)
a rotation about Z transforming vector X into vector N.

Angle 2 = (Z, W)
a rotation about vector N transforming vector Z into vector W.

Angle 3 = (N, U)
a rotation about vector W

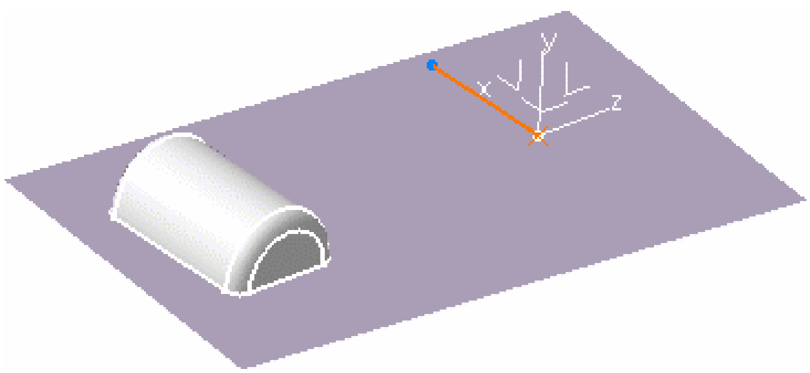


2. Select the point as shown to position the origin of the axis system you wish to create. The application then computes the remaining coordinates. Both computed axes are then parallel to those of the current system. The axis system looks like this:

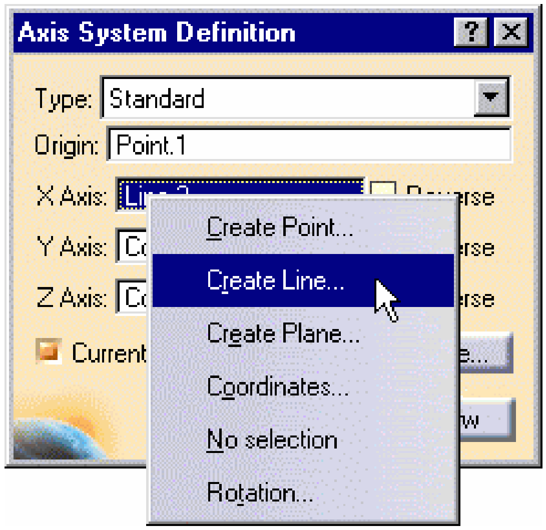


3. If you are not satisfied with x axis, for instance click the X axis field and select a line to define a new direction for x axis.

The x axis becomes collinear with this line.

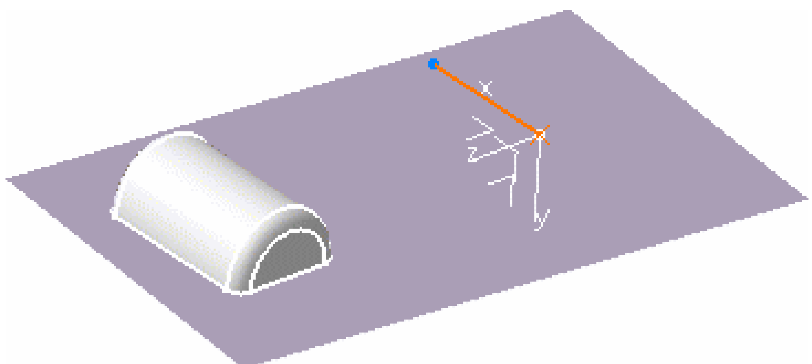


- It can be a line created along the surface edge, for example, using the **Create Line** contextual menu on the selection field, and selecting two surface vertices.
  Similarly you can create points, and planes.
- You can also select the **Rotation** contextual menu, and enter an angle value in the X Axis Rotation dialog box.



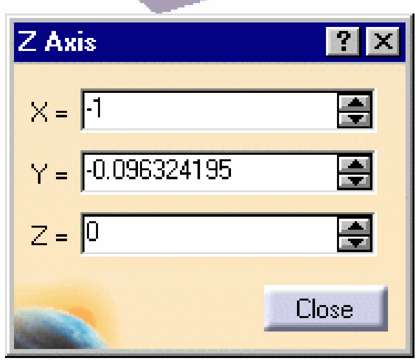
4. Click the y axis in the geometry to reverse it.

   Checking the **Reverse** button next to the **Y Axis** field reverses its direction too.



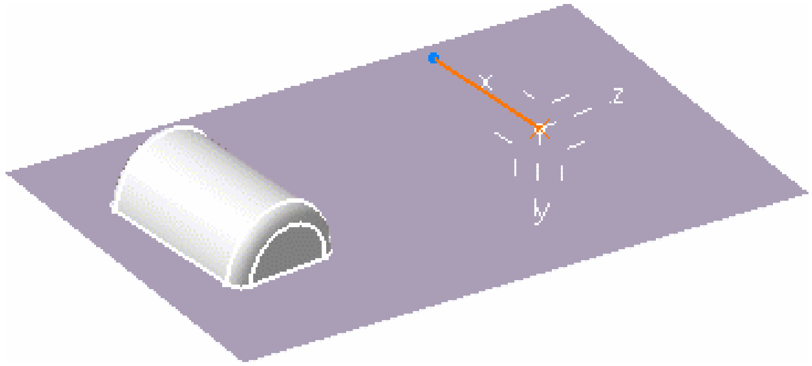
5. You can also define axes through coordinates.

   Right-click the Z Axis field and select the **Coordinates** contextual command. The Z Axis dialog box appears.



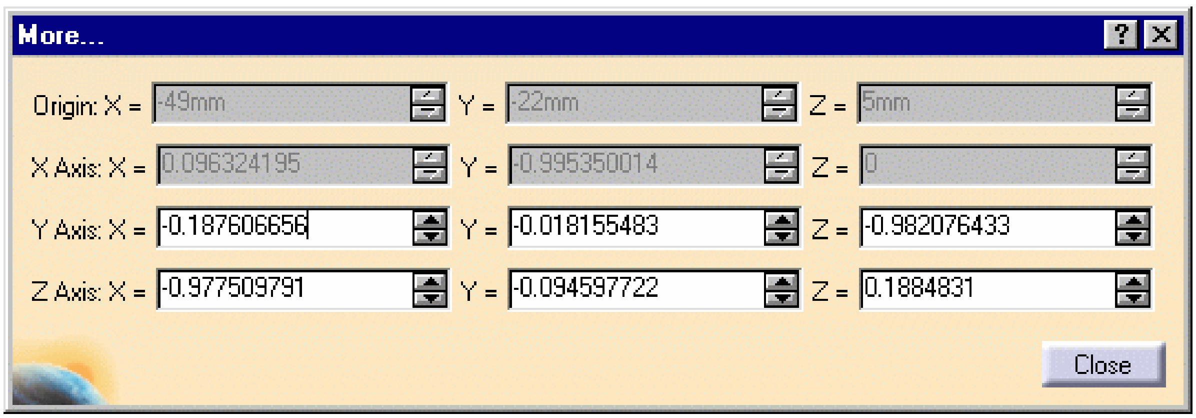
6. Key in X = -1, retain the Y and Z coordinates, and click Close.

The axis system is modified accordingly, and is now left-handed.



7. Click **More...** to display the More... dialog box.

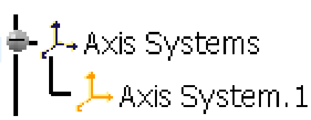
The first rows contains the coordinates of the origin point. The coordinates of X axis are displayed in the second row. The coordinates of Y and Z axis are displayed in the third and fourth row respectively.

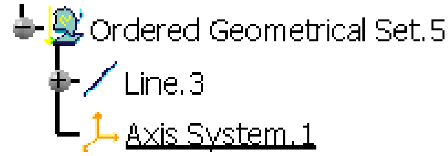


As you are defining your axis system, the application detects if its axes are orthogonal or not. Inconsistencies are revealed via the Update diagnosis dialog box.

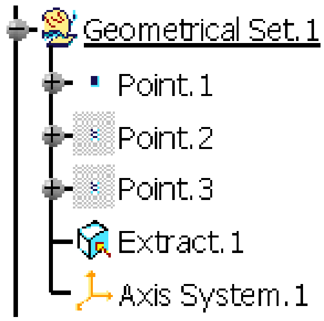
8. Uncheck the **Current** option if you do not want to set your axis as the reference. The absolute axis at the bottom right of the document then becomes the current three axis system.

9. Uncheck the **Under the axis system node** option if you do not want the axis system to be created within the Axis system node in the specification tree.





It will be created either in the current geometrical set or right after the current object in an ordered geometrical set. In this case, the axis system becomes the new current object.



10. Click OK.

The axis system is created.

When it is set as current, it is highlighted in the specification tree.

11. 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.

- You can change the location of the axis system and put it in a geometrical set.

  To do so, select it in the specification tree, right-click and select **Axis System.1 object -> Change Geometrical Set**. Choose the destination of the axis system using the drop-down list.

  Please refer to the Managing Geometrical Sets chapter to have more information.

- 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.

- You can contextually retrieve the current local axis direction.
  Refer to the Stacking Commands chapter to have further information.

- There is an associativity between the feature being created and the current local axis system. Therefore when the local axis system is updated after a modification, all features based on the axis direction are updated as well.

- 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

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.X object 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.

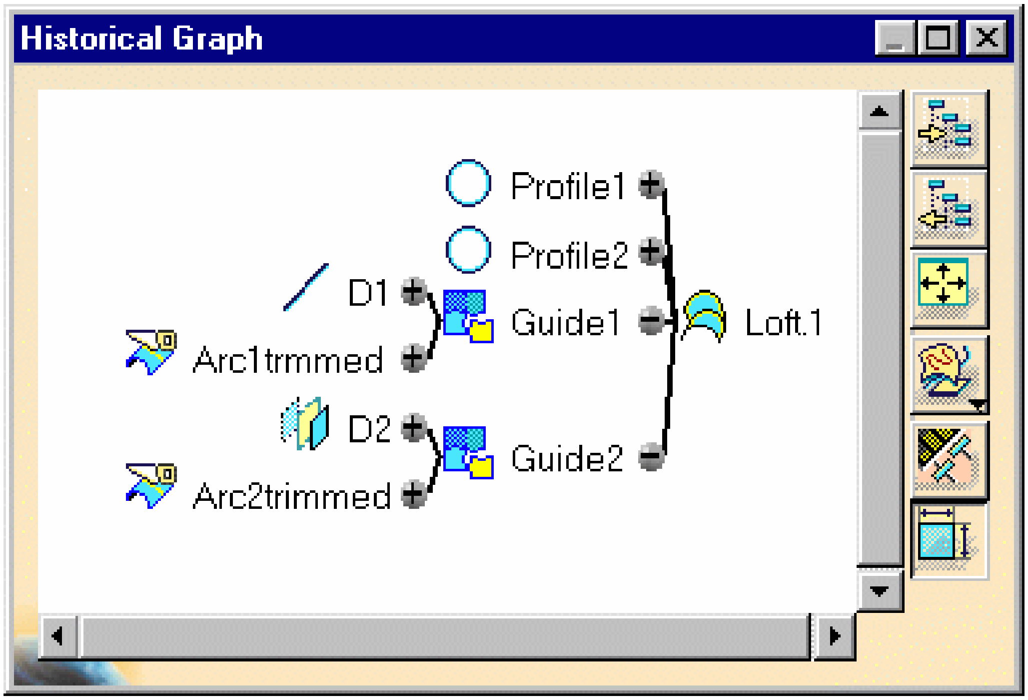
The **Under the axis system node** option is not available when editing an axis system.

Using the Historical Graph

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

This task shows how to use the Historical Graph.

1. Select the element for which you want to display the historical graph.



2. Click the **Show Historical Graph** icon .

The Historical Graph dialog box appears.

The following icon commands are available.

- Add graph
- Remove graph
- Reframe graph
- Surface or Part graph representation
- Parameters filter
- Constraints filter.

3. Just close the dialog box to exit this mode.

Working with a Support

This task shows how to create a support. It may be 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 in the support, whenever you need a reference point to create other geometric elements.

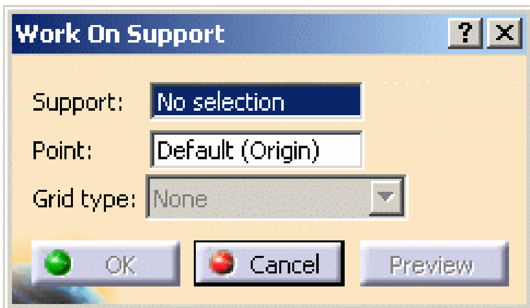
- Creating a support from a surface
- Creating a support from a plane
- Creating an infinite axis from the active work on support

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.

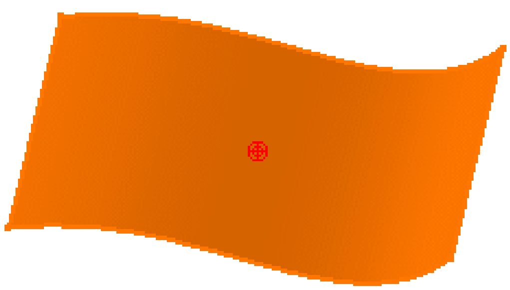


2. Select the surface to be used as support element.

   If a plane is selected, a grid is displayed to facilitate visualization.

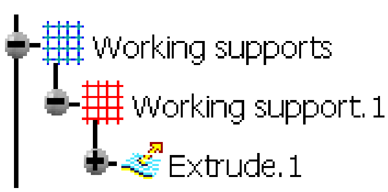
3. Optionally, select a point.

   By default the surface's midpoint is selected.



4. Click OK in the dialog box.

   The element (identified as Working support.xxx) is added to the specification tree under the Working



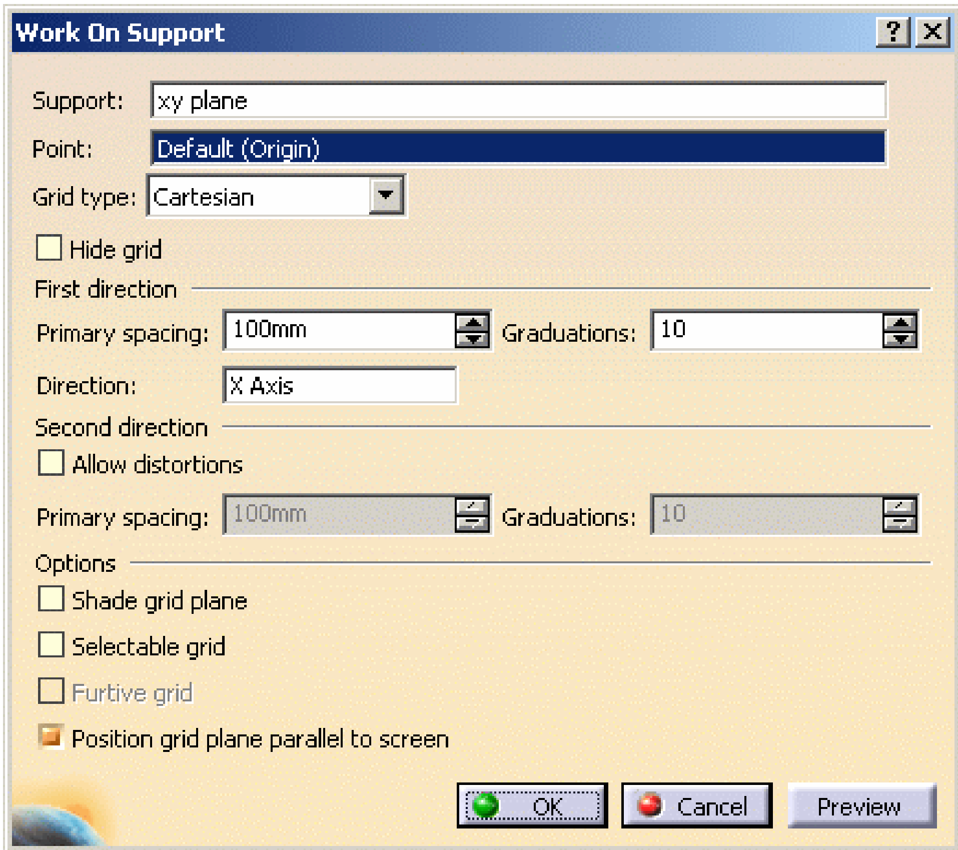
supports node.

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.

- 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.

- Check the **Selectable grid** option to enable the selection of sub-elements of the grid (lines and points) as a support for a future selection.

  Selected sub-elements are featurized.

- Check the **Furtive grid** option to see the grid only when it is parallel to the screen.

This option is only active only if the **Selectable grid** option is checked.

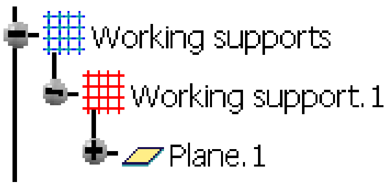
- Check the **Position grid plane parallel to screen** to reset the grid visualization parallel to the screen.

The **Primary spacing** and **Graduations** options are defined in **Tools -> Options -> Shape -> Generative Shape Design**.

Please refer to the Customizing section for further information.

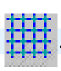
7. Click OK in the dialog box.

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

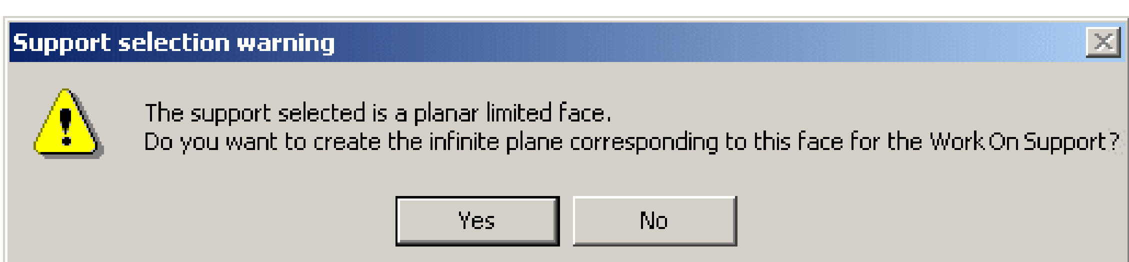


Creating an infinite plane from a limited planar surface

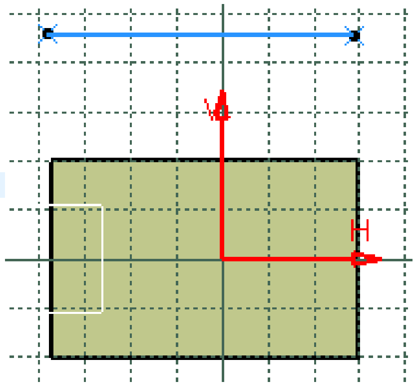
Open the WorkOnSupport3.CATPart document.

1. Click the **Work on Support** icon .
2. Select a face of Extrude.1.

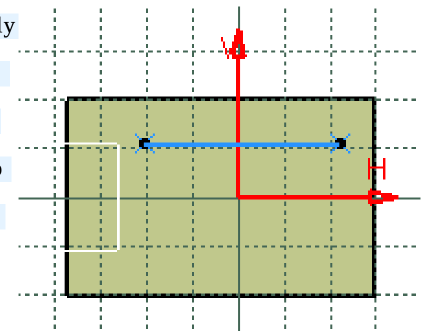
A warning message is issued asking you whether you wish to create an infinite work on support from this face.



- If you click Yes, the plane is inserted in the current geometrical set or ordered geometrical set and is used as the **Support**. You will be able to create features on this support.



- If you click No, only the face is used as the **Support**. You will only be able to create features on this limited face.



Creating an infinite axis from the active work on support

This capability is only available with the Rotate and Helix commands.
Let's take an example with the Rotate command.

Open the WorkOnSupport2.CATPart document.

1. Click the **Rotate** icon .

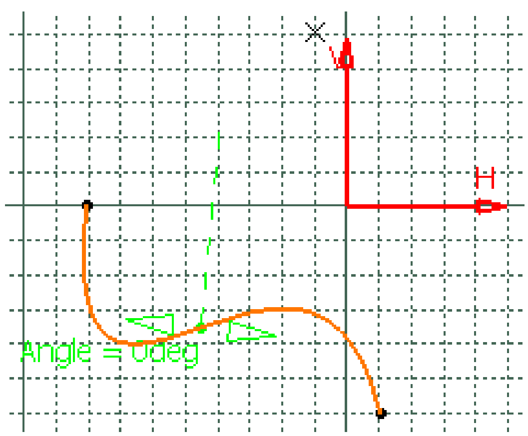
   The Rotate dialog box displays.

2. Select the Spline as the element to be rotated.

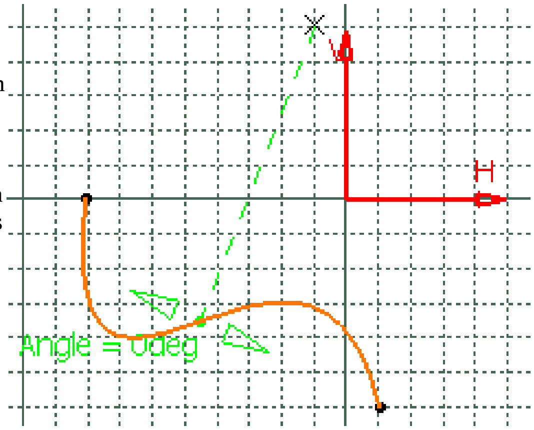
3. Select the axis.

There are two ways to create an infinite axis on the fly:

a. Click anywhere on the Work on Support.
   The point and the axis needed for the axis are created.



b. Select a point in the 3D geometry.
The axis is created through this point and is normal to the active Work on Support.



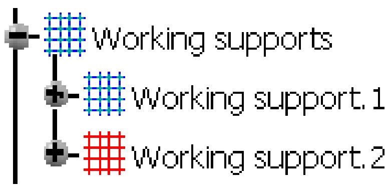
4. Click OK to create the rotated element.

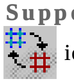
The axis is an infinite line normal to the support and passing through the featurized point. This line is aggregated to the Rotate.x feature and put in no show.

This capability is available with a Work on Support defined by a planar element (whether finite or not).

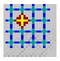
Setting a work on support as current

By default the last created working support is displayed in red in the specification tree.



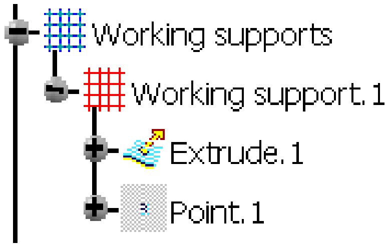
Use the **Set As Current/Set As Not Current** contextual menu on the working support features, or the **Working Supports Activity**  icon to define which is the default current support that will be automatically selected when entering a command that requires a working support.

Snapping to a point

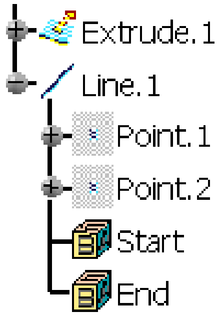
Click the **Snap to point** icon  to snap the point being created onto the nearest intersection point on the grid.

- 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.

- Points created while in the Work on Support command, regardless of the type of working support created (surface or plane), are aggregated under the Working support and put in no show.



- 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, polyline, and most commands where you need to select points as inputs. The created points using a support are aggregated under the parent command that created them and put in no show in the specification tree.



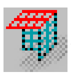
- Working supports can be edited, updated, or deleted just as any other feature.

Working with a 3D Support

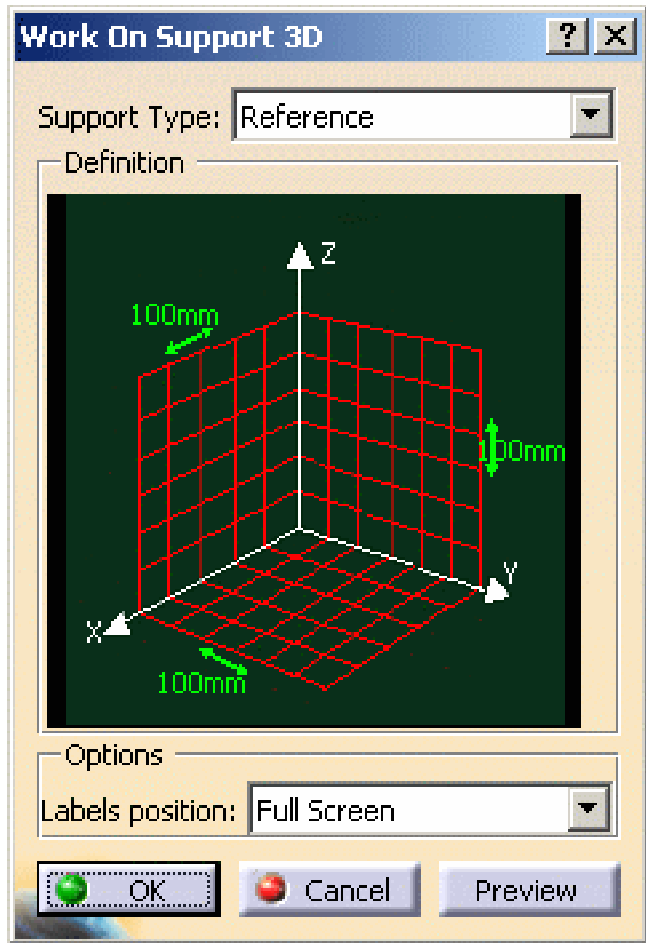
This command is only available with the Automotive BiW Template product.

This task shows how to create a 3D support. It is composed of three regular grid of lines, generally set on the three main planes of the part, that aggregates 3 selectable work on supports.
It allows you to create reference points on the fly on each support, whenever you need a reference point to create other geometric elements. You will no longer have to explicitly select the support element.
It also allows you to create sub-elements of the grid on the fly (points, edges). These features do not appear neither in the specification tree nor in the 3D geometry but are aggregated under the feature using them.

Open the WorkOnSupport1.CATPart document.

1. Click the **Work on Support 3D** icon .

   The Work on Support 3D dialog box appears.



- Each of three grid lines has one default primary spacing of 100mm for each direction. The three directions of the main axis system define the grids directions.
  You can edit the spacing values by clicking on the spacing tag to edit and modify them.
  Note that you can modify these values at creation, not at edition, and that there can only be one value per grid.
  Grids are used both as an input to create geometry as well as visual help.
- You can also modify the name of the labels of the main directions by clicking on the direction tag.

Labels' directions and primary spacing are defined in **Tools -> Options -> Shape -> Generative Shape Design**.
Please refer to the Customizing section for further information.

2. Choose the Labels position:

   - **Full screen**: labels are displayed all around the screen
   - **Bottom/Left**: labels are displayed on the bottom left of the screen
   - **None**: no label is displayed

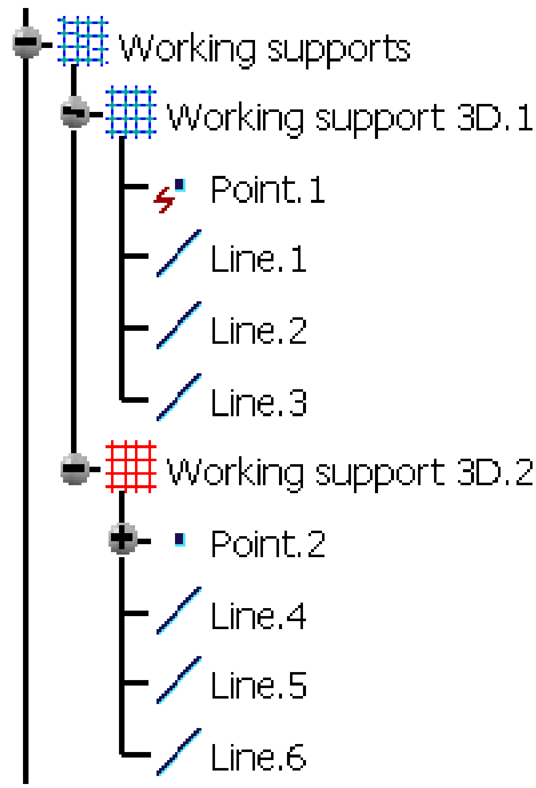
3. Define the Support Type:

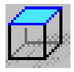
   - **Reference**: the 3D support is created according to the main axis system. There can be only one reference 3D work on support.
   - **Local**: a local axis system must be specified. There can be as many local 3D works on support as desired.

4. Click OK in the dialog box.

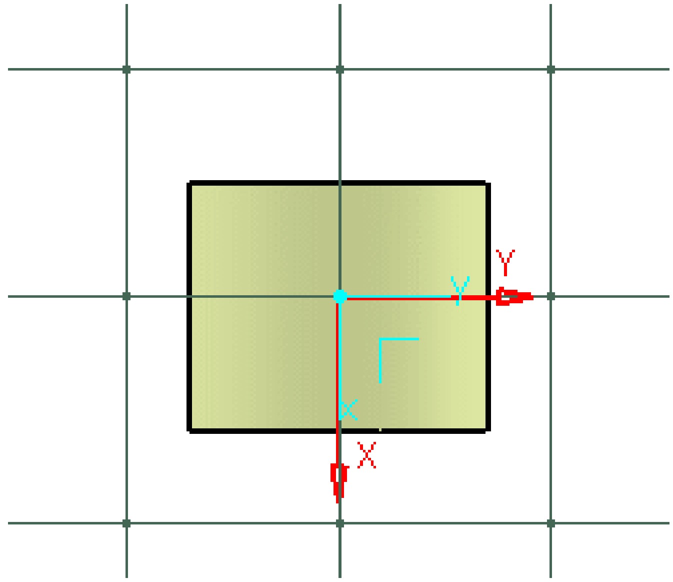
   Here is an example with a reference and a local 3D work on support.

   The elements (identified as Working support 3D.xxx) are added to the specification tree under the Working supports node.



5. Select the **Top View**  icon from the Quick View toolbar.

The active work on support is visualized and labels are displayed on each straight line.

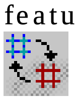


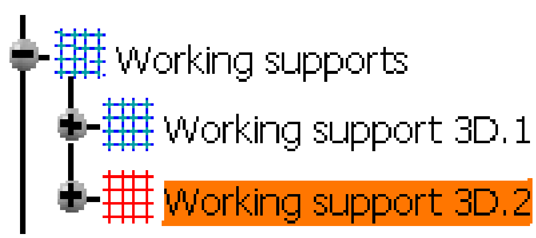
- The work on support must be parallel to one of the three planes to be visualized. As a consequence, the active 3D work on support may be seen independently in each view of the same document.
- If you move the compass, the 3D work on support is no longer parallel to the screen.

Note:

- There can only be one active 3D work on support at the same time.
- When the local axis system is modified, all related features are updated.

Setting a work on support as current

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 on the working support features, 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.
You can also set the axis system as not current to reactivate the three planes and define the reference 3D support as the current support.



Snapping to a point

Click the **Snap to point** icon to snap the point being created onto the nearest intersection point on the grid.

Switching the featurization to lines or planes

Use the **Support Featurize Line** icon to switch the featurization of the grid lines to lines.

Conversely, use the **Support Featurize Plane** icon to switch the featurization of the grid lines to planes.
Featurized lines and planes are created normal to the current grid.

- 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.
- Once you choose to work on the 3D support, you can directly click onto the support to create points. This capability is available with commands such as point, line, spline, polyline, and most commands where you need to select points as inputs.
  The created points using a support are aggregated under the parent command that created them and put in no show in the specification tree.
- Each 3D working support can be edited, updated, or deleted just as any other feature.

Creating Plane Systems

The Plane System command provides tools letting you define a number of planes in a given direction. Planes can then be used as reference planes or supports when creating other items.

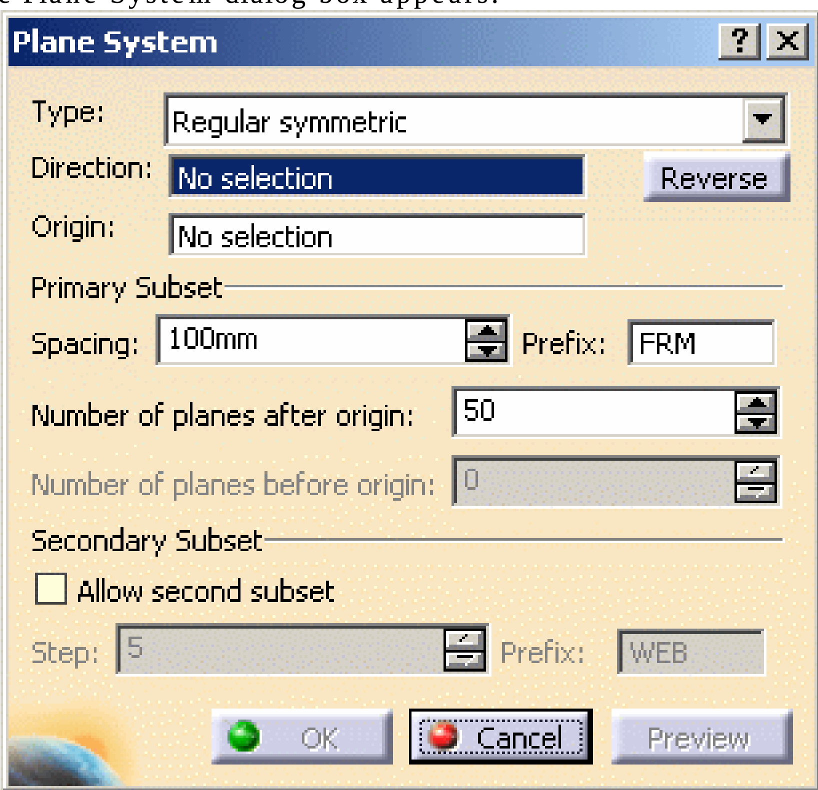
In structure applications, you can, for example, define reference planes in each ship direction to assist you place structural elements. You must define one plane system for each direction.

This task shows you how to create a regular asymmetric, a irregular asymmetric and a semi-regular plane system.

1. Click the **Plane System** icon.

- In the Generative Shape Design workbench, this icon is to be found in the Tools toolbar. You can also select **Insert -> Advanced Replication Tools -> Plane System...**
- In the Structure Preliminary Layout workbench, this icon is to be found in the Structure Grid Set toolbar. Having selected this command, you also need to select an entry in the specification tree under which you want to create a new CATPart to locate your plane system before proceeding. If you want to use an existing CATPart, then select that CATPart in the tree.
- For Structure Functional Design, switch to the Generative Shape Design workbench.

The Plane System dialog box appears.



2. Select the type of plane system you want to create:

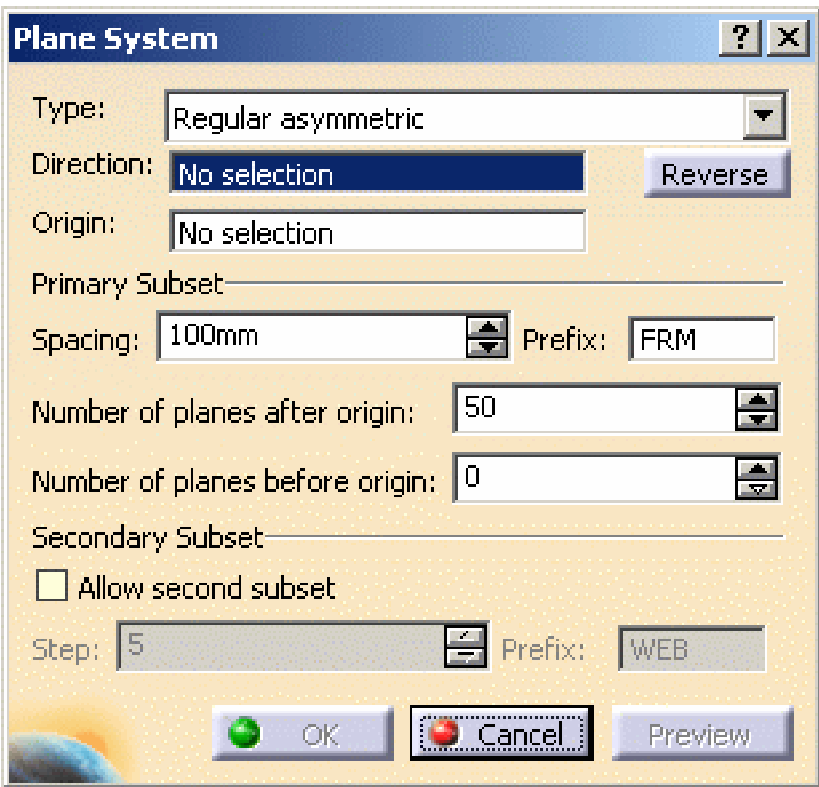
Five types are available:

- Regular symmetric
- Regular asymmetric
- Semi-regular
- Irregular symmetric
- Irregular asymmetric.

Symmetric Plane Systems

Symmetric plane systems are created in similar fashion to asymmetric plane systems. The difference being that they have the same number of planes on either side of the origin.

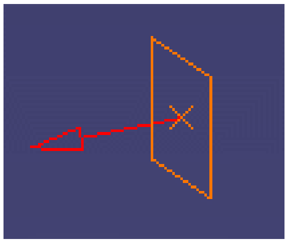
Creating a Regular Asymmetric Plane System



3. Select a plane or a line to define the direction of the plane system.

If you select a plane, the center of the plane is automatically taken as the origin of the plane system and an arrow appears showing the direction.

You can, if desired, change the origin.



4. If you selected a line, select a point to define the origin,

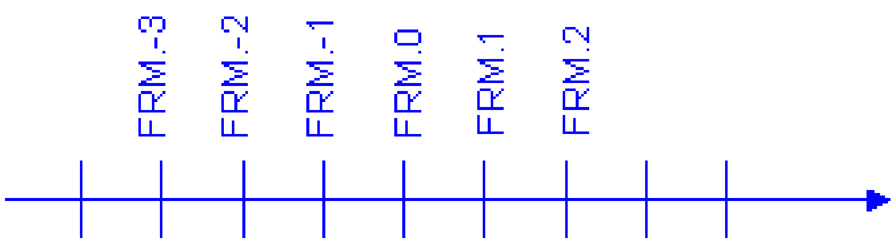
   Or,

   If you selected a plane and want to change the origin, click the Origin field and select a point.

- Use the Reverse button in the dialog box or select the arrow in the geometry area to invert the direction.
- The contextual menus in Direction and Origin fields let you create appropriate geometry directly without having to exit the current command.

5. Specify the primary subset:

- Specify the distance between two planes in the Spacing field.
- Enter a prefix identifying all planes in this set.
  Planes are identified by this prefix plus a positive or negative number that increments away from the origin. Plane numbers are positive in the direction of the plane system. The origin is identified by prefix.0.



- Specify the number of planes.
  The number you enter is the number of planes you want to create on either side of the origin. Note that the number of planes does not include the plane at the origin.

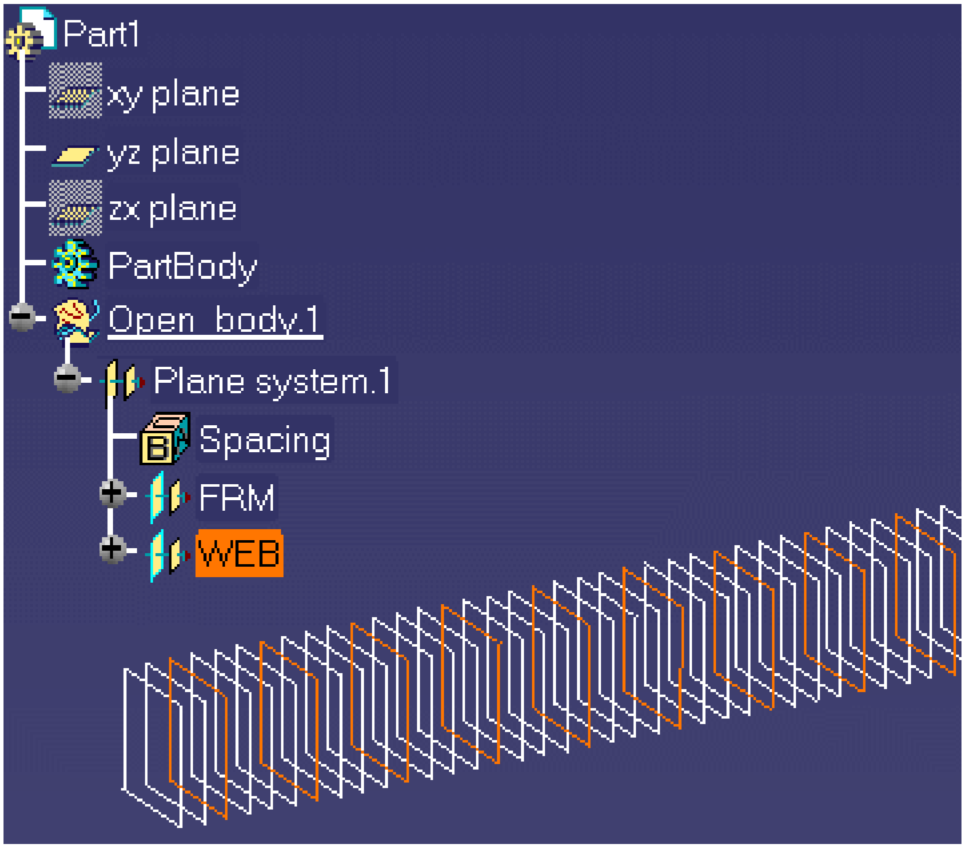
4. Optionally, check Allow secondary subset to group a number of planes in the primary subset together and create a secondary subset:

- Specify the step. For example, enter 4.
  Every fourth primary subset plane will belong to the secondary subset.
- Enter a prefix identifying all planes in this set.

Note: The plane at the origin always belongs to the primary subset.

Select the subset in the specification tree to visualize all planes in this set in the geometry area.

6. Click **OK** when done to create a plane system along the specified direction.



Creating an Irregular Asymmetric Plane System

Plane systems can be created by importing a TSV (tab-separated) file containing the definition of the plane system.

This file must be formatted as follows:

positive_or_negative_absolute_distance_from_origin<TAB>subset_prefix
where <TAB> denotes a TAB character

and should contain an entry **0<TAB>subset_prefix**. Typically,

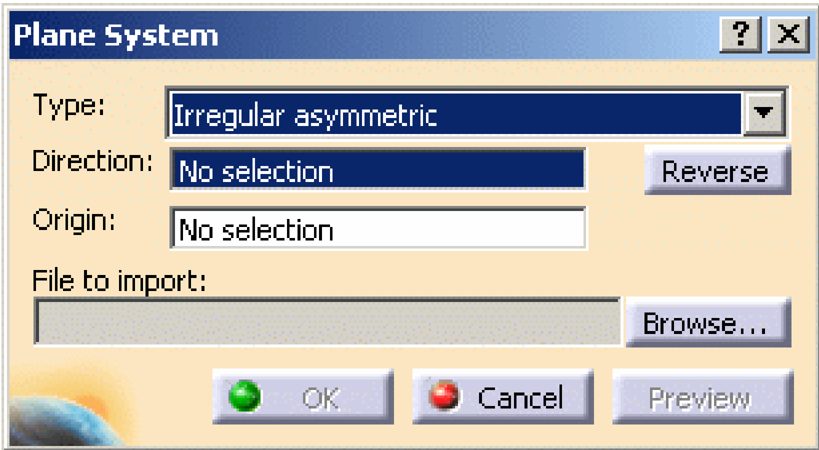
-4800 WEB
-4200 FRM
-3600 FRM
-3000 FRM
-2400 WEB
...
0 FRM

...
2300 WEB
2700 WEB

Notes:

- Do not type space characters using the space bar.
- It is not necessary to specify the positive sign '+' when entering positive distances.

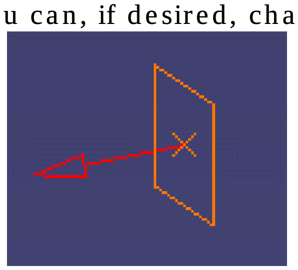
It defines a plane system in one ship direction only but can contain as many subsets as desired.



3. Select a plane or a line to define the direction of the plane system.

If you select a plane, the center of the plane is automatically taken as the origin of the plane system and an arrow appears showing the direction.

You can, if desired, change the origin.



4. If you selected a line, select a point to define the origin,

   Or,

   If you selected a plane and want to change the origin, click the Origin field and select a point.

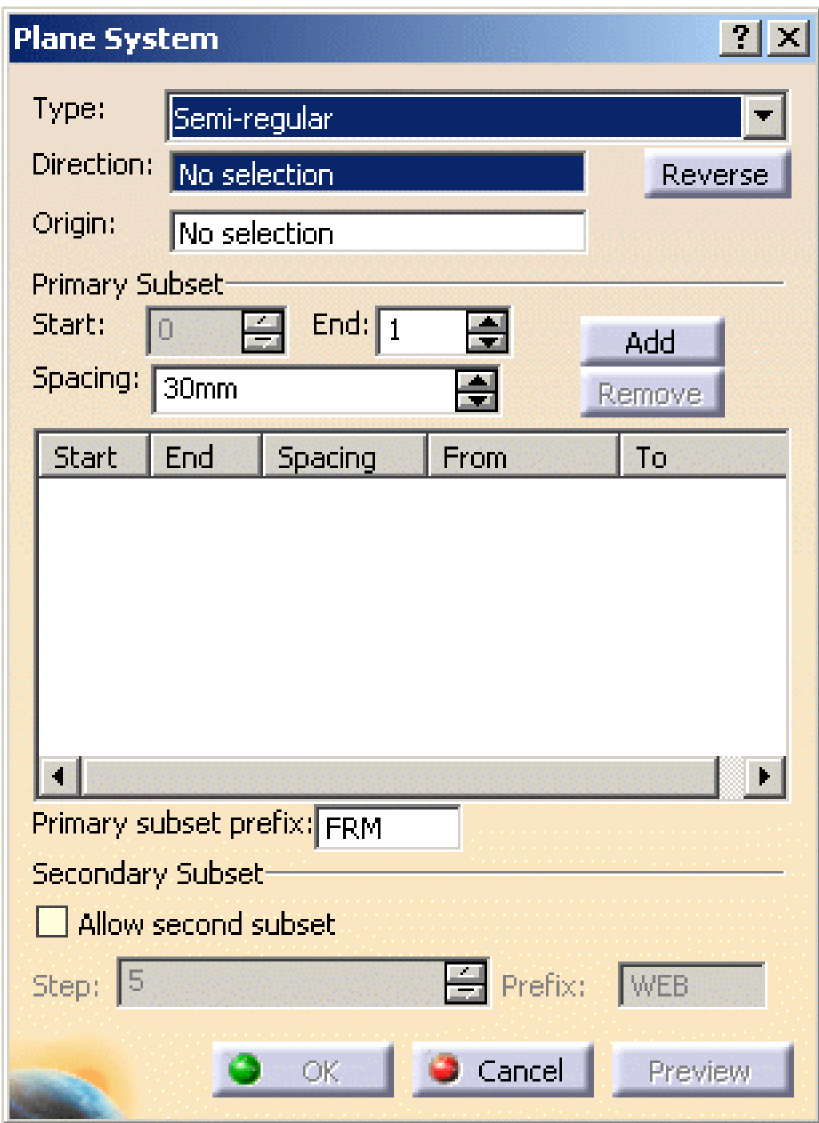
- Use the Reverse button in the dialog box or select the arrow in the geometry area to invert the direction.
- The contextual menus in Direction and Origin fields let you create appropriate geometry directly without having to exit the current command.

5. Click Browse... and navigate to the file containing the plane system definition.

6. Click **OK** when done to create a plane system along the specified direction.

Creating a Semi-Regular Plane System

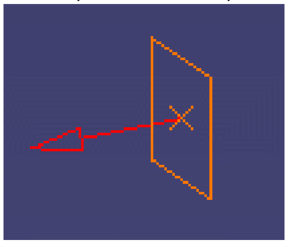
The semi-regular option lets you easily and rapidly define a plane system comprising groups of planes with different spacings.



3. Select a plane or a line to define the direction of the plane system.

If you select a plane, the center of the plane is automatically taken as the origin of the plane system and an arrow appears showing the direction.

You can, if desired, change the origin.



4. If you selected a line, select a point to define the origin,

   Or,

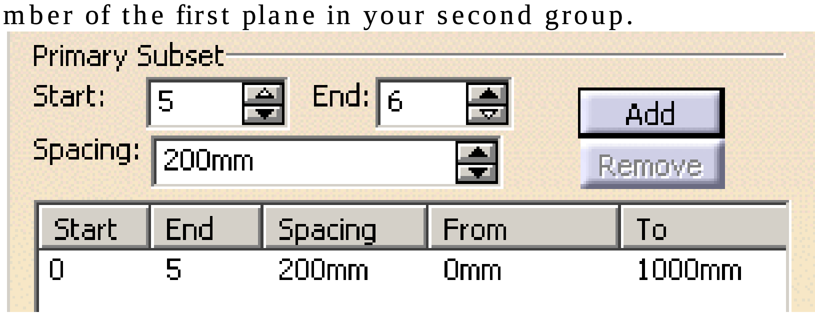
   If you selected a plane and want to change the origin, click the Origin field and select a point.

- Use the Reverse button in the dialog box or select the arrow in the geometry area to invert the direction.
- The contextual menus in Direction and Origin fields let you create appropriate geometry directly without having to exit the current command.

5. Specify the primary subset:

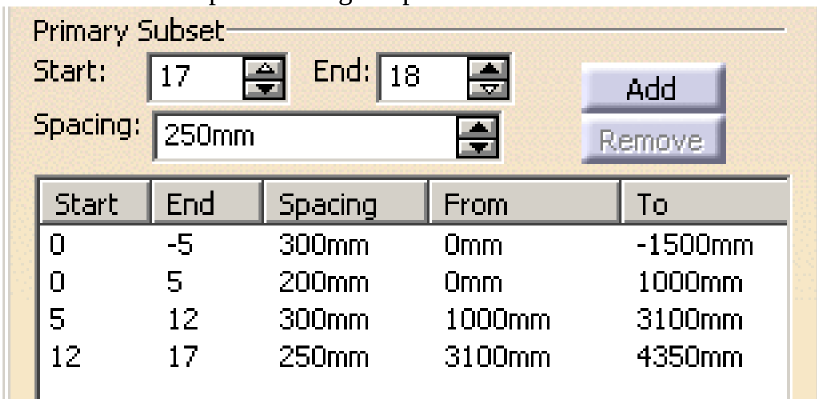
- Specify the distance between two planes in your first group in the Spacing field.
- Enter the number of the last plane having the specified spacing in the End field.
- Click Add to confirm your first group.

The first group is identified in the list view control and the Start field incremented to display the number of the first plane in your second group.



- Repeat to specify the spacing and the number of the last plane to be created with this spacing, then click Add.
- Continue until satisfied.

Note: If the current spacing is the same as the spacing of the previous group, any new planes are added to the previous group.



- Enter a prefix identifying all planes in the primary set.
  Planes are identified by this prefix plus a positive or negative number that increments away from the origin. Plane numbers are positive in the direction of the plane system. The origin is identified by prefix.0.

6. Optionally, check Allow secondary subset to group a number of planes in the primary subset together and create a secondary subset:

- Specify the step. For example, enter 4.
  Every fourth primary subset plane will belong to the secondary subset.
- Enter a prefix identifying all planes in this set.

Notes: The plane at the origin always belongs to the primary subset.
Select the subset in the specification tree to visualize all planes in this set in the geometry area.

Adding Groups to Your Plane System

- Click in the list view control to return to the Add mode.

Modifying Groups in Your Plane System

- Select the group you want to modify.
- Enter a new spacing value or modify the End value to change the number of planes in the group.
  Note: You cannot modify the Start value.
- Click Modify.

The plane system is updated. Changing the number of planes in any one group does not affect the number of planes in other groups.

Note: Click in the list view control to cancel unwanted modifications that have not been confirmed using Modify.

Removing Groups

- Select the group you want to remove.
- Click Remove.

7. Click **OK** when done to create a plane system along the specified direction.

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.

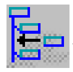
A click on the icon activates the Datum mode for the current or the next command.

The History mode (active or inactive) will remain fixed from one session to another: it is in fact a setting.

Inserting Elements

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

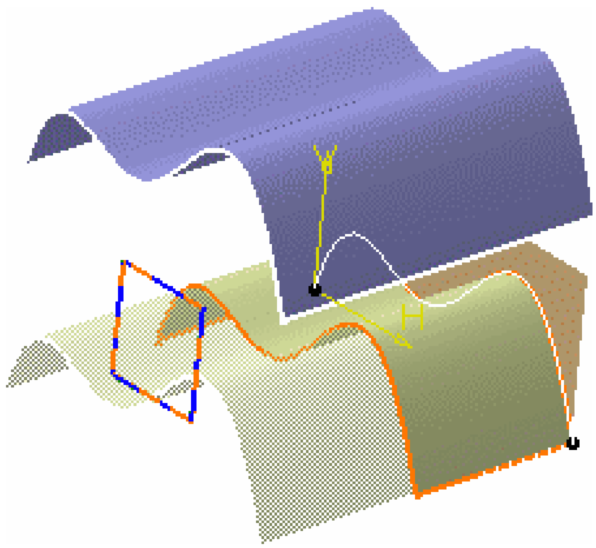
This task shows how to create a geometric element and automatically inserting it next to its main parent in the specification tree.
This may be useful to add an element that was not not initially designed, while retaining its logical positioning within the specification tree.
All children of the main present are then attached to the inserted element.
Open the Insert01.CATPart document.
It contains, amongst other geometric elements, an extruded surface that has been translated.

1. Click the **Insert Mode** icon .

   It stays active and you can select another icon.

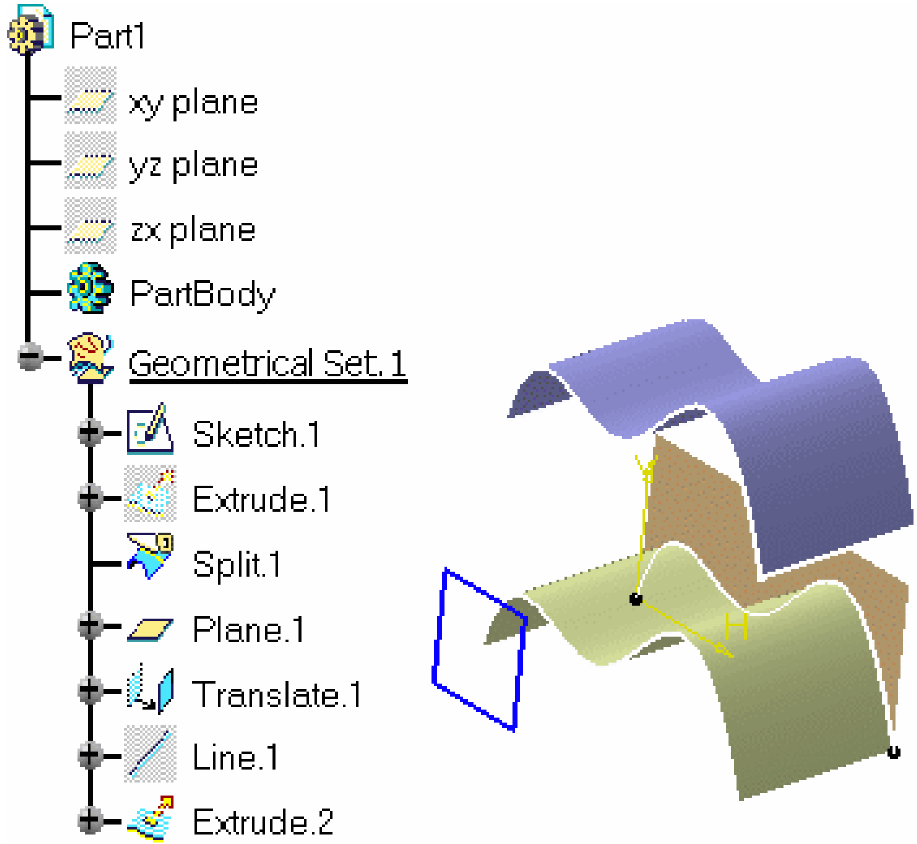
2. Click the **Split** icon .

3. Successively select the extruded surface (Extrude.1) and the plane (Plane.1) as the splitting element.

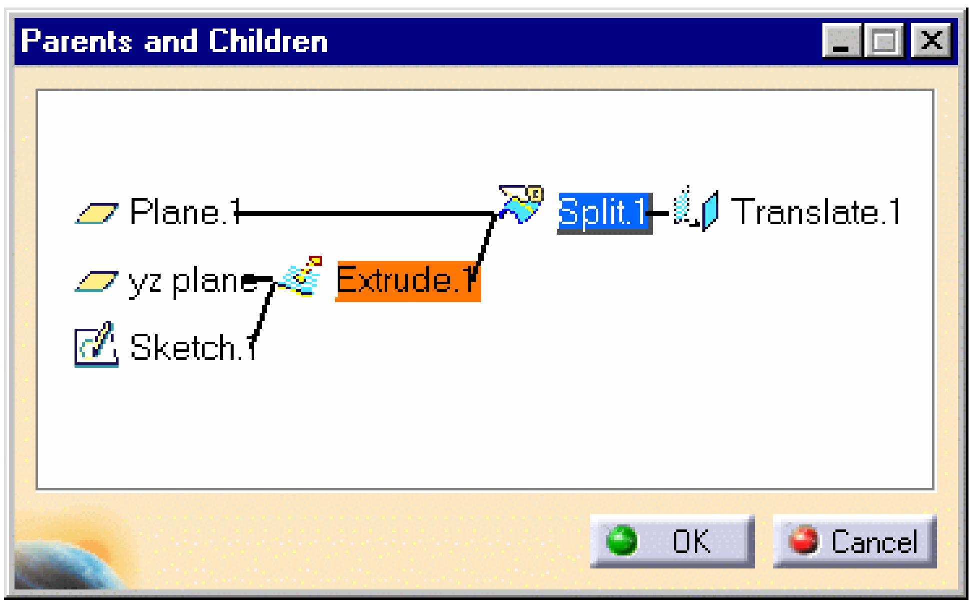


4. Click OK in the Split Definition dialog box.

The Split.1 element is created and inserted directly below the Extrude.1 element, and the translated surface is split as well, as the splitting operation takes place chronologically before the translation.



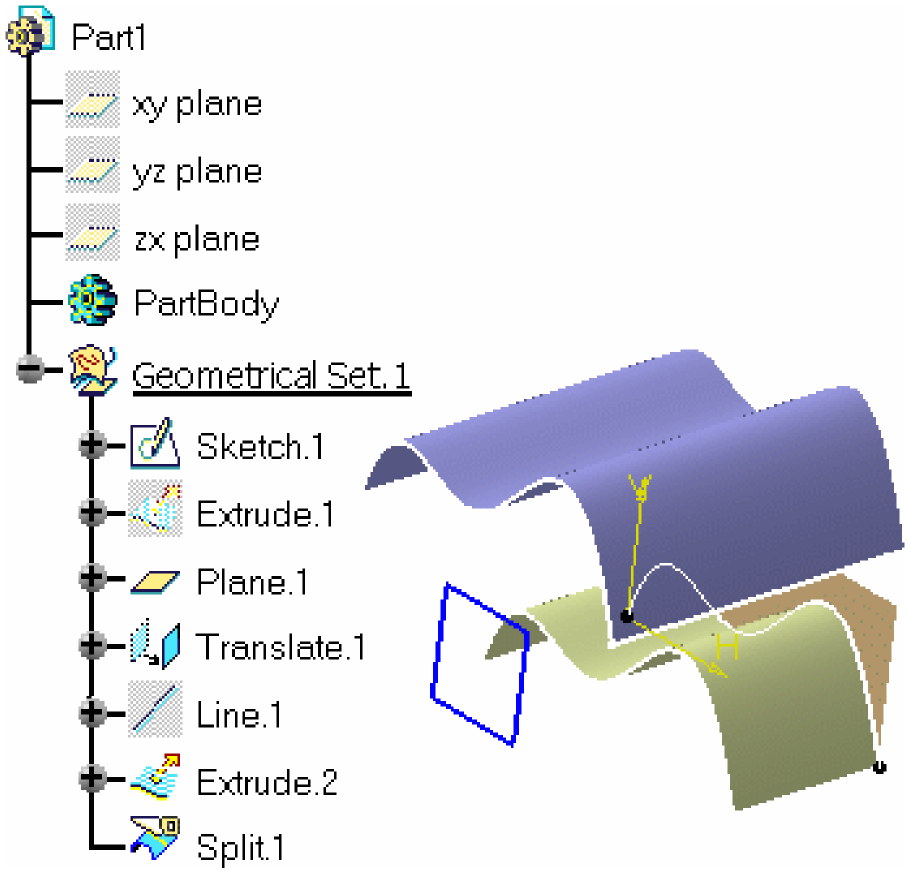
5. Right-click the Split.1 element in the specification tree and choose the **Parent/children** contextual menu.



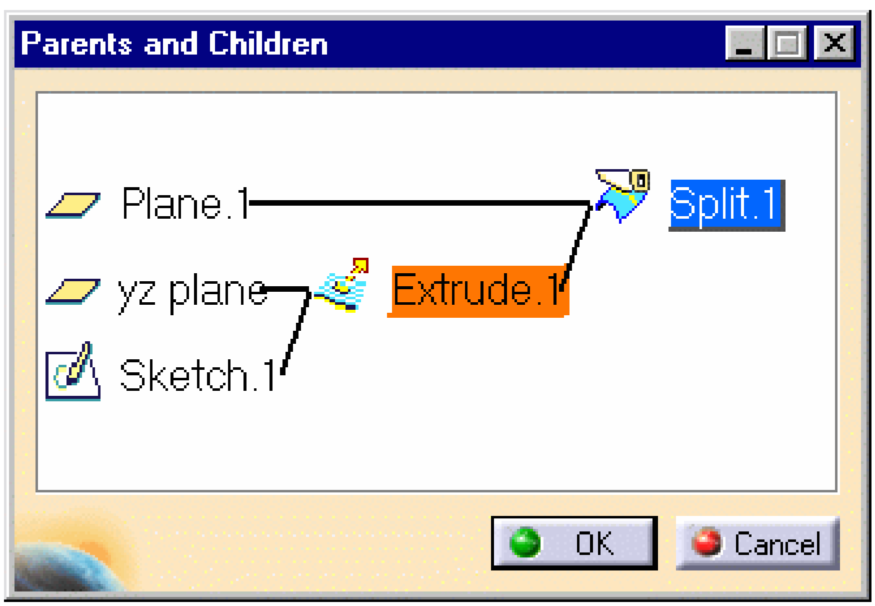
The Parents and Children dialog box is displayed.

The Extrude.1 and Plane.1 are parents of the split surface, which children is the translated surface (Translate.1).

Would the insert mode not been activated, the splitting operation would have been recorded in the specification tree, at the end of the currently active geometrical set, and the translated surface would not have been split:



Using the **Parent/children** contextual menu, on the split surface, you notice that the Split.1 element does not have any children.



This insertion capability is available when creating:

- shape fillets
- split elements
- trimmed elements
- extrapolated surfaces or curves
- joined elements
- inverted elements

Keeping the Initial Element

This task shows you how to retain an element on which you are performing an operation. When this command is active, as soon as you perform an action in which you create or modify geometry, you are in fact working on a copy of the initial element.

The Keep and No Keep modes can be activated via the **Keep Mode** and **No Keep Mode** icons in the Tools toolbar.

Keep Mode

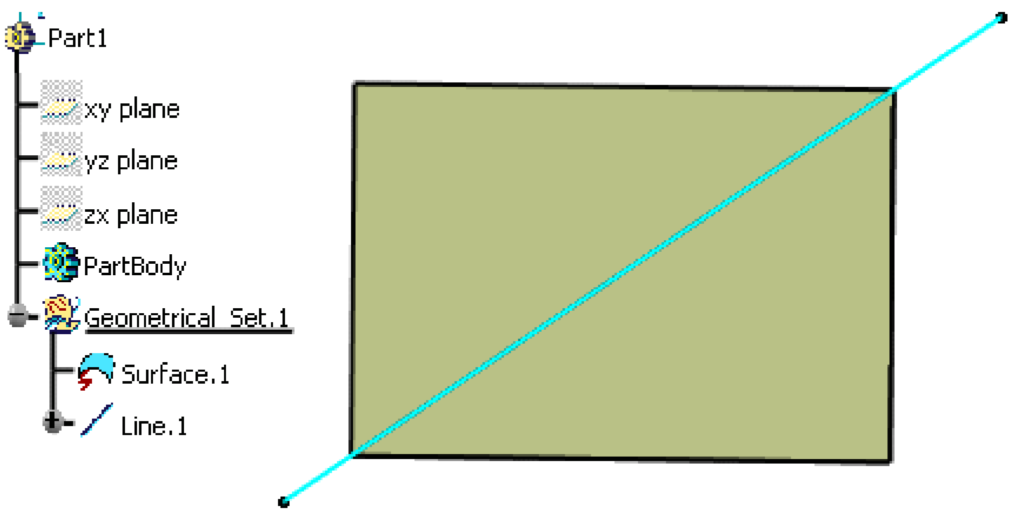
The implementation of this mode allows modification features to have the same behavior as creation features. This mode is identical for both geometrical set and ordered geometrical set environments, whatever type of the input and output (= result) elements are, that is to say whether they are datum or not.

The input element:

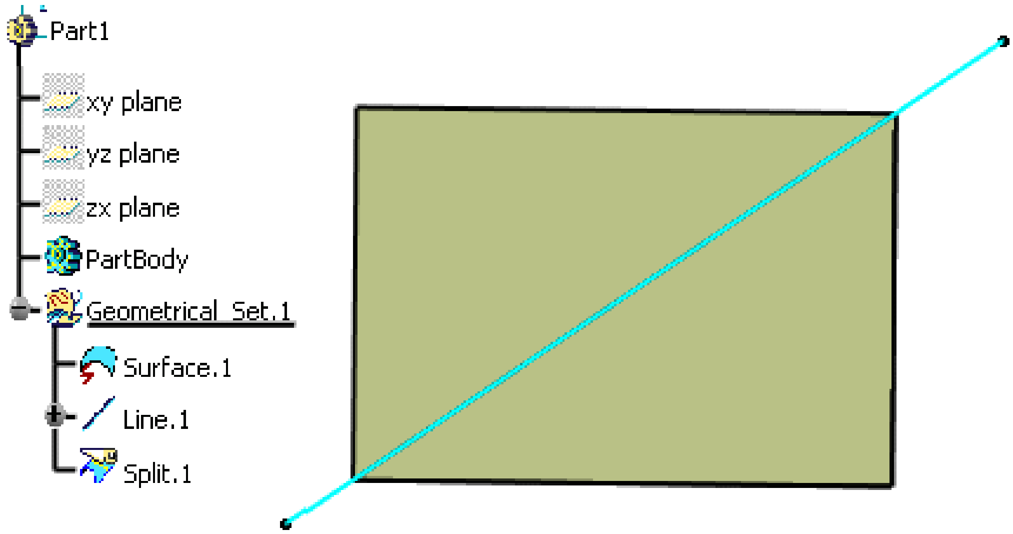
- remains in the show area
- can be detected and selected in the 3D geometry
- can be detected in the specification tree

Let's take an example with the Split command:

A surface and a line are created. The surface is to be intersected with the line.



1. Check that the **Keep Mode** mode is activated.
2. Click the **Split** icon .
3. Split the surface by the line.



The whole surface remains in the show area.

Double-clicking the **Keep Mode** icon lets you work in a global mode: as a consequence, all created features will be in Keep mode.

No Keep Mode

The No Keep mode is only available with the modification commands. It has no impact on the creation commands.
Here are the list of commands impacted by the No Keep mode.

Command	Conditions
3D Curve Offset	
All transformations	
Blend	With Trim support
Bump	
Combine	
Connect Curve	With Trim mode
Corner	With Trim Support
Curve Smooth	
Develop	
Diabolo	No GSD plane as input
Extrapolate	With Assemble result
Fillet	With Trim Support
Healing	
Inverse	
Join	
Near	
Offset	

Parallel Curve	
Project	
Shape Morphing	
Split	No GSD plane as input
Sweep	Tangent sweep with Trim Support
Trim	
Variable Offset	
Wrap Curve	
Wrap Surface	

The implementation of this mode depends on the type of the input and output (= result) elements, that is to say whether they are datum or not.
This mode transforms contextual creation features into modification features.

Datum Input and Datum Result

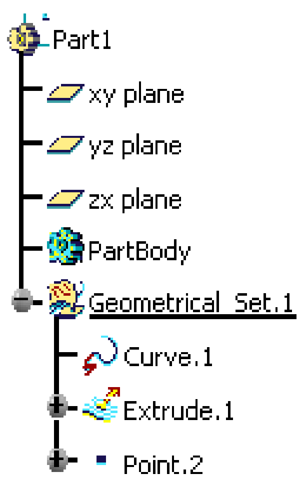
This mode is identical, whatever the environment (geometrical set or ordered geometrical set).
The input element:

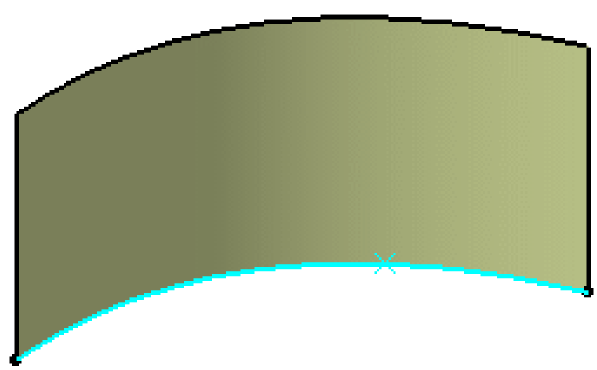
- is deleted
- is replaced by the created feature (if their dimensions are strictly identical)
- its child features are impacted

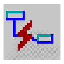
=> **Behavior 1** (see table below)

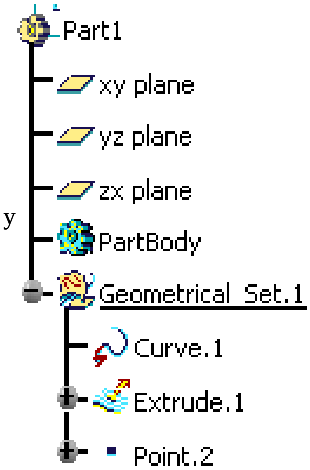
Let's take an example with the Split command.

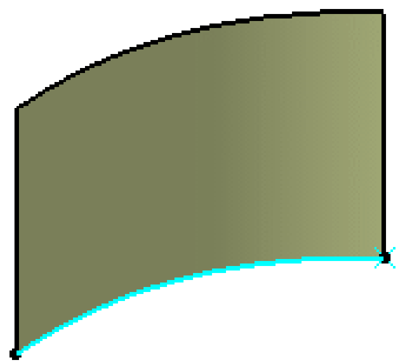
A datum curve, a point on the curve, and a surface based on this curve are created.





1. Check that the **No Keep Mode**  and the **Create Datum**  icons are activated.
2. Click the **Split** icon .
3. Split the curve by the point.





The input curve is replaced by the resulting split curve and the surface is impacted.

Feature Input and Datum Result

a. Geometrical set environment

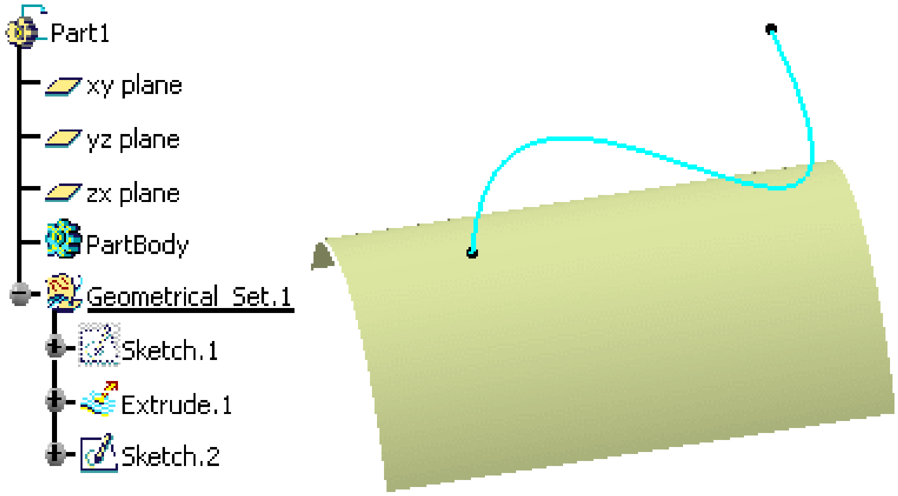
The input element:

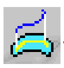
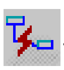
- is put in the no show area
- cannot be detected and selected in the 3D geometry
- can be detected and selected in the specification tree
- its child features are not impacted

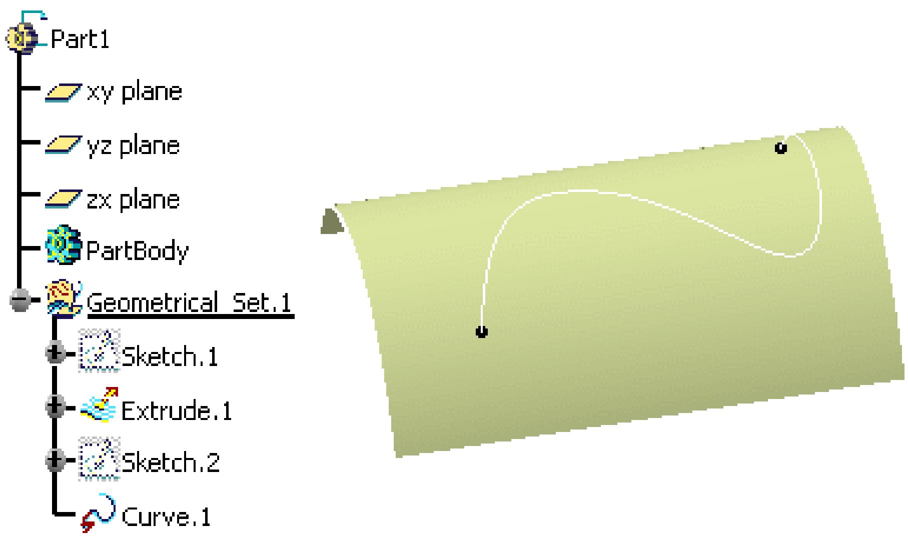
=> **Behavior 2** (see table below)

Let's take an example with the Project command.

A sketch and a surface are created. The sketch is to be projected onto the surface.



1. Check that the **No Keep Mode** icon  is activated.
2. Click the **Project** icon .
3. Click the **Create Datum** icon .
4. Project the element (Sketch.2) onto the surface (Extrude.1)



The input sketch is put in no show and a datum curve is created.

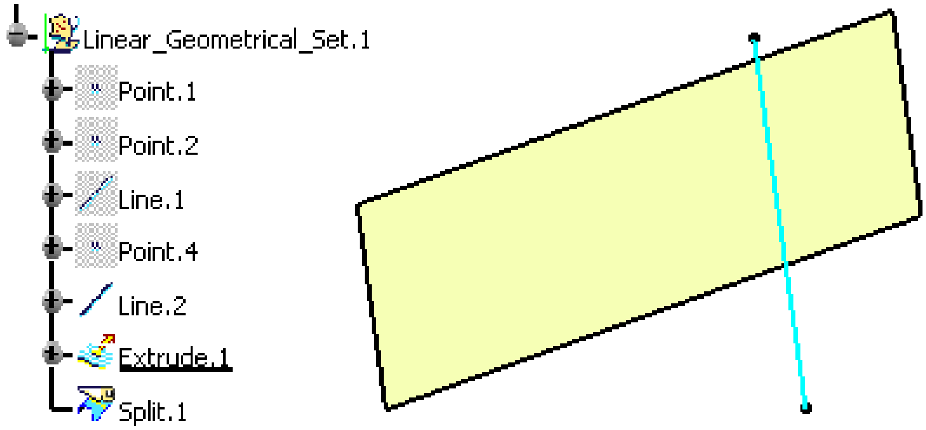
b. Ordered geometrical set environment

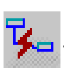
The input element:

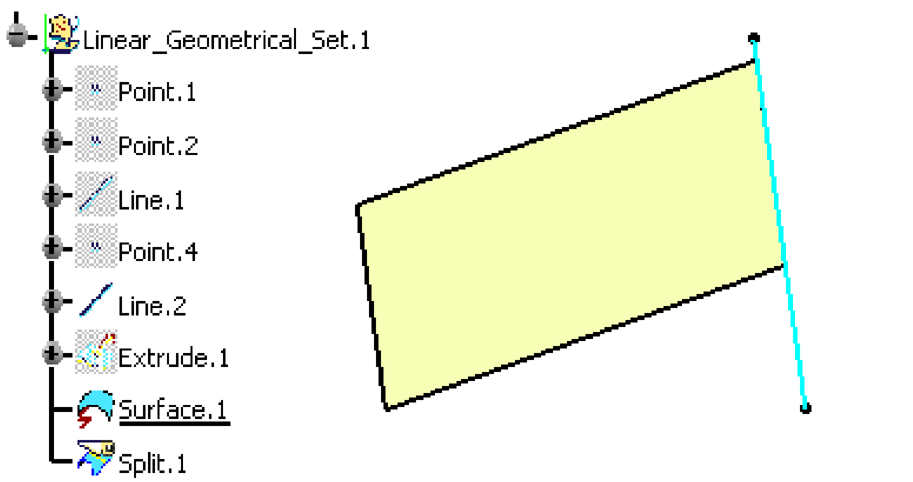
- is put in the no show area
- cannot be detected and selected in the 3D geometry
- can be detected and selected in the specification tree
- its child features are impacted if the created feature is inserted before them

Let's take an example with the Split command.

An extruded surface is created and a line intersects it (Line.2). The extruded surface has a child feature (Split.1) and is defined as the current object.



1. Check that the **No Keep Mode** icon  is activated.
2. Click the **Split** icon .
3. Click the **Create Datum** icon .
4. Split the extruded surface with the line.



The extruded surface is put in no show and its child feature is impacted: its input is now the new surface (Surface.1).

Therefore, when the Datum mode is associated to the No Keep mode and the result can replace the input, the behavior is the one described above.

Datum or Feature Input and Feature Result

a. Geometrical set environment

The behavior is the same as above (Feature Input and Datum Result).

=> **Behavior 2** (see table below)

b. Ordered geometrical set environment

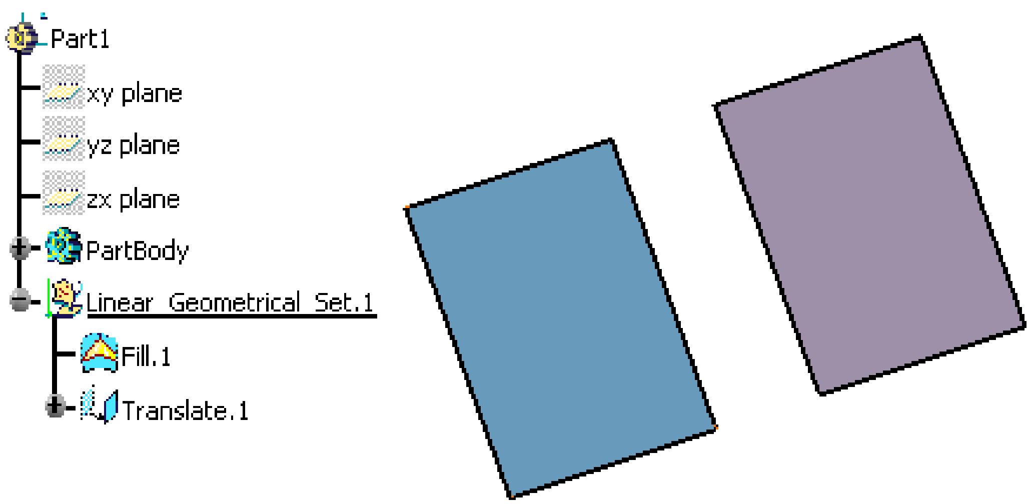
The input element:

- is in the ghost area
- cannot be detected and selected in the 3D geometry
- can be detected and selected in the specification tree
- its child features are impacted if the new feature is the created inserted before them

=> **Behavior 3** (see table below)

Let's take an example with the Offset command.

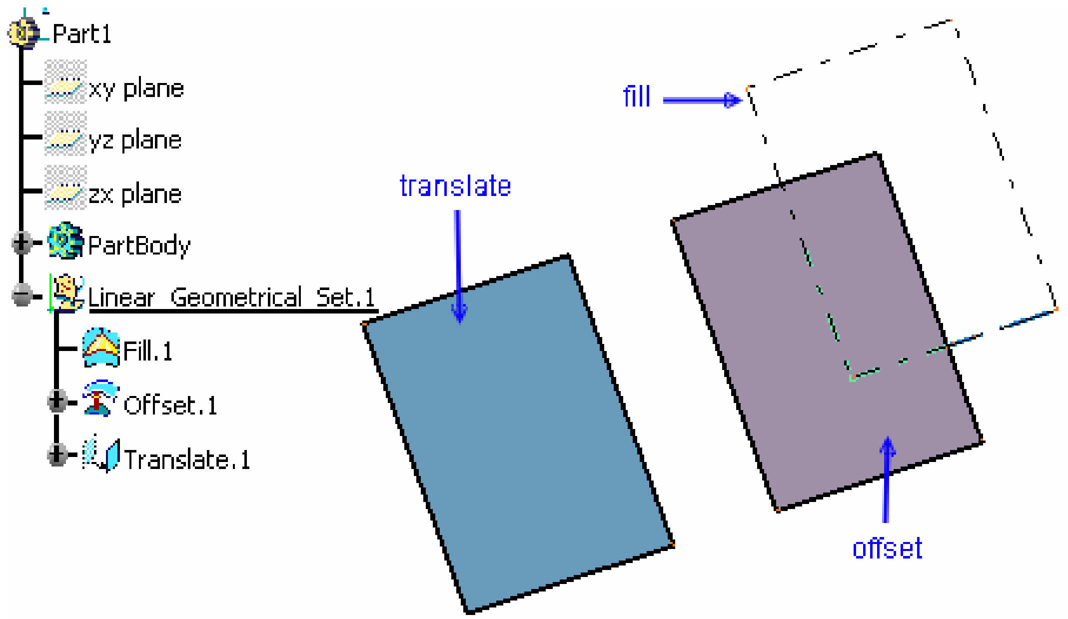
A fill and a translate of this fill are created. The translate is thus a child of the fill.
The fill is defined as the current object.



1. Check that the **No Keep Mode** icon is activated.

2. Click the **Offset** icon and offset Fill.1.

The offset surface is created before the translate. The fill is absorbed and the translate is impacted.



Double-clicking the **No Keep Mode** icon lets you work in a global mode: as a consequence, all created features will be in No Keep mode.

To conclude with the No Keep mode, here is a table summarizing the different behaviors:

		Datum Result	Feature Result
Datum Input	**Geometrical Set**	Behavior 1	Behavior 2
	Ordered Geometrical Set	Behavior 1	Behavior 3
Feature Input	**Geometrical Set**	Behavior 2	Behavior 2
	Ordered Geometrical Set	Behavior 2	Behavior 3

- The default option is Keep mode for creation features and, and No Keep mode for modification features.
- Features created using the contextual menu are always set in Keep mode.
- If a sub-element is selected as an input of a command in No Keep mode, it is not put in the no show area.
- When editing a feature, you cannot change its mode.

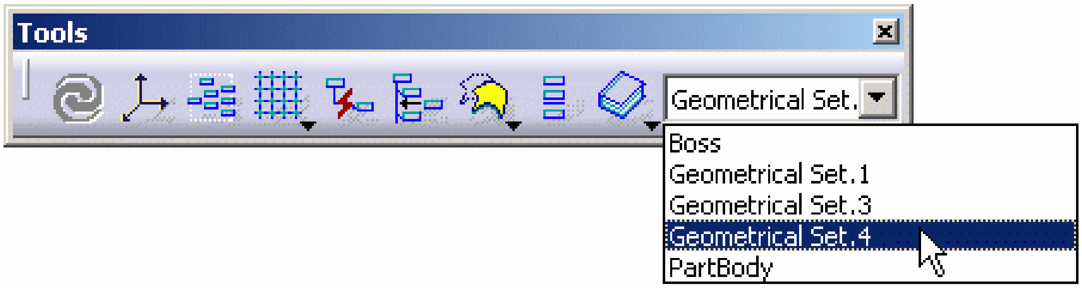
Selecting Bodies

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

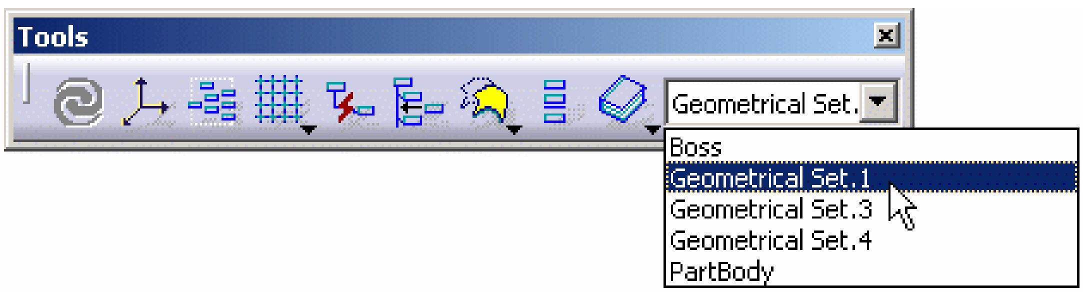
This task shows how to rapidly select a specific Body, whether a Geometrical Set, an Ordered Geometrical Set or PartBody, using the Body Selector. This is especially useful when the specification tree is hidden or too large to be easily manipulated, in the case of a large document for example.

Open the PowerCopyStart1.CATPart document.

1. From the Tools toolbar, click the arrow on the drop-down list to display the list of Bodies present in the document.



2. Choose the body you want to work in, from the list.



The selected body is displayed in the Body Selector's field, and underlined in the specification identifying it as the current body.

- All Bodies are displayed in the list alphabetically, whether they are in Show or No Show mode.
- This command is equivalent to selecting the Body in the specification tree using the icon, right-clicking it and choosing the **Define In Work Object** command.
  To rename your Body, you need to follow five steps:

1. Select the object from the specification tree
2. Choose the **Properties** contextual menu
3. Click the **Feature Properties** tab in the Properties dialog box
4. Key in a new **Name**
5. Click **OK** in the Properties dialog box.

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.

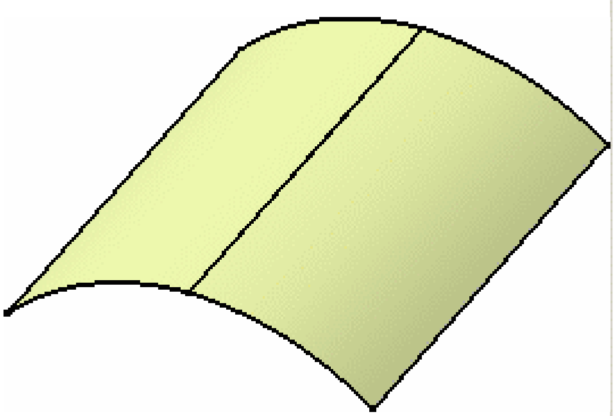
- **Distance**: the values are expressed in millimeters
- **Tangency**: the values are expressed in degrees
- **Curvature**: the values are expressed in percentage.

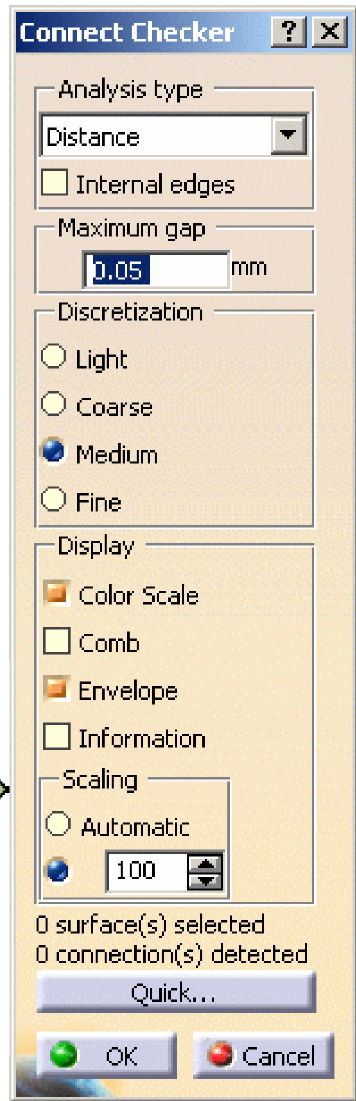
Open the ConnectChecker1.CATPart document.

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.

The **Auto Min Max** button enables to automatically update the minimum and maximum values (and consequently all values between) each time they are modified.





Check the **Internal edges** option if you want to analyze the internal connections.

By default, the check box is unchecked.

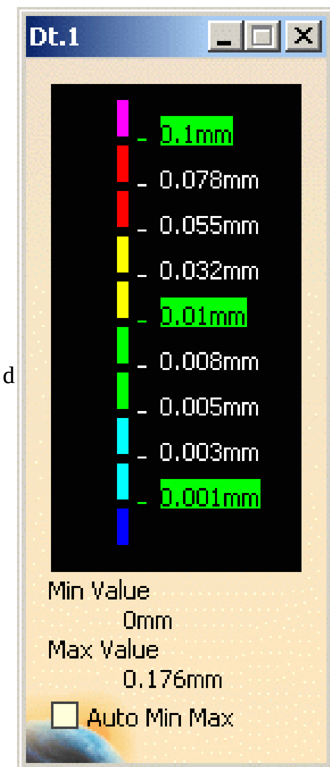
Two cases are available:

- Surfaces are isolated.
  Only geometrical connections are checked, that is all pairs of neighboring surface edges within the tolerance given by the Maximum gap.
  Depending on the Maximum gap value, interference connections may be detected, for instance when surfaces have a size smaller the Maximum gap. In this case, you must decrease the Maximum gap value or join the surfaces to be analyzed (see next point)
- Surfaces are joined (using the Join command for instance) and the Internal edges option is checked.
  Topological connections are checked first, that is all edges shared by two topological surfaces. Then, the corresponding pairs of surface edges are checked to detect any geometrical connections within the tolerance given by the Maximum gap.

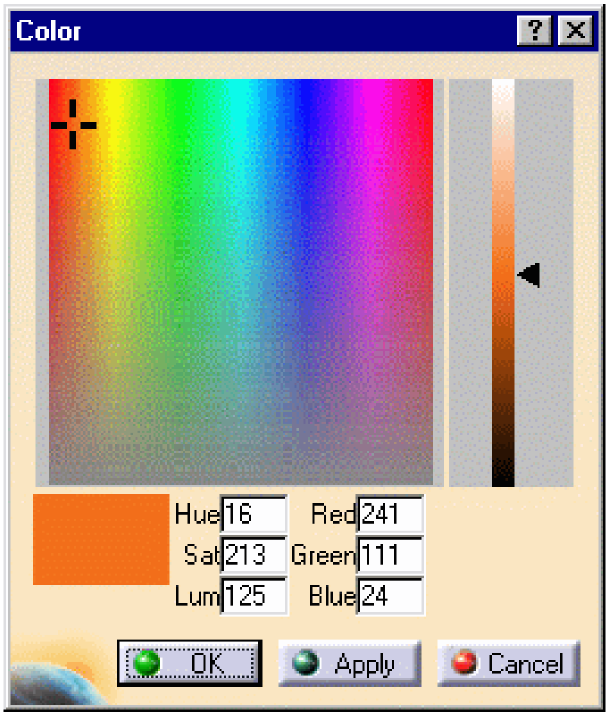
3. Choose the analysis type to be performed: **Distance**, **Tangency** or **Curvature**.

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.

In the color scale, the **Auto Min Max** button enables to automatically update the minimum and maximum values (and consequently all values between) each time they are modified.



- You can right-click on a color in the color scale to display the contextual menu:



- Edit: it allows you to modify the values in the color range to highlight specific areas of the selected surface. The Color dialog box is displayed allowing the user to modify the color range.

- **Unfreeze**: it allows you to perform a linear interpolation between non defined colors. The unfreezed values are no longer highlighted in green.

- **No Color**: it can be used to simplify the analysis, because it limits the number of displayed colors in the color scale. In this case, the selected color is hidden, and the section of the analysis on which that color was applied takes on the neighboring color.

- You can also right-click on the value to display the contextual menu:

- **Edit**: it allows you to modify the edition values. The Value Edition dialog box is displayed: enter a new value (negative values are allowed) to redefine the color scale, or use the slider to position the distance value within the allowed range, and click OK. The value is then frozen, and displayed in a green rectangle.

- **Use Max/Use Min** : it allows you to evenly distribute the color/value interpolation between the current limit values, on the top/bottom values respectively, rather than keeping it within default values that may not correspond to the scale of the geometry being analyzed. Therefore, these limit values are set at a given time, and when the geometry is modified after setting them, these limit values are not dynamically updated.

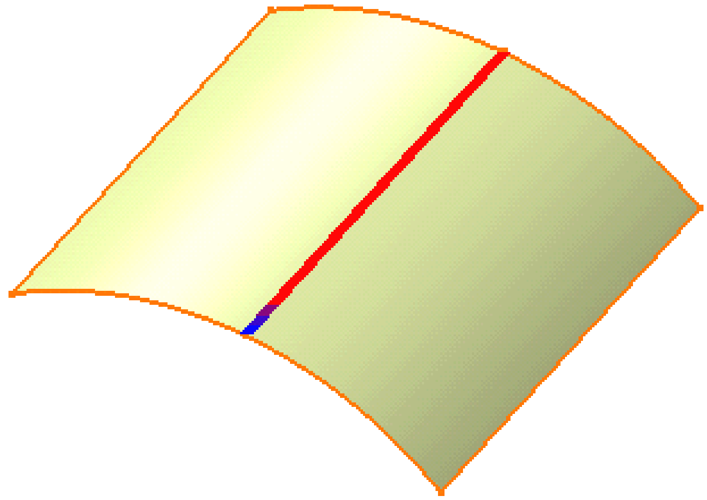
The **Use Max** contextual item is only possible if the maximum value is higher or equal to the medium value. If not, you first need to unfreeze the medium value.

Only the linear interpolation is allowed, meaning that between two set (or frozen) colors/values, the distribution is done progressively and evenly.

The color scale settings (colors and values) are saved when exiting the command, meaning the same values will be set next time you edit a given draft analysis capability.
However, new settings are available with each new draft analysis.

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.



There may be a tangency discontinuity while a curvature continuity exists. This may appear for instance in the case of two non tangent planar surfaces.

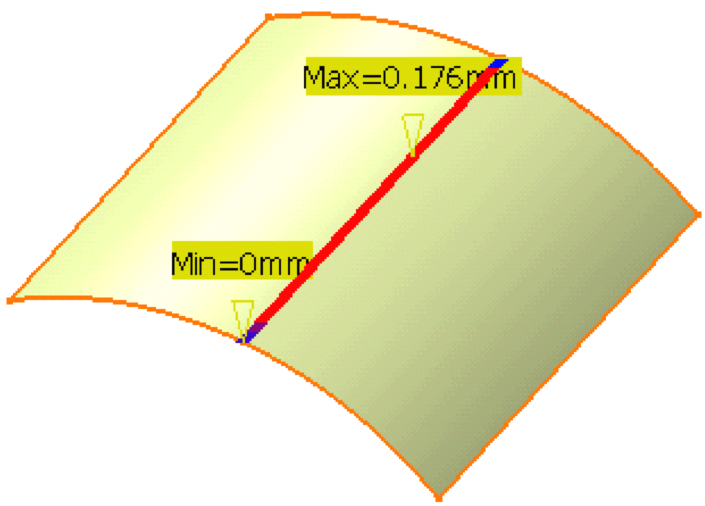
From the Connect Checker dialog box, you can choose a number of visualization and computation options:

- 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 displayed in the 3D geometry

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.

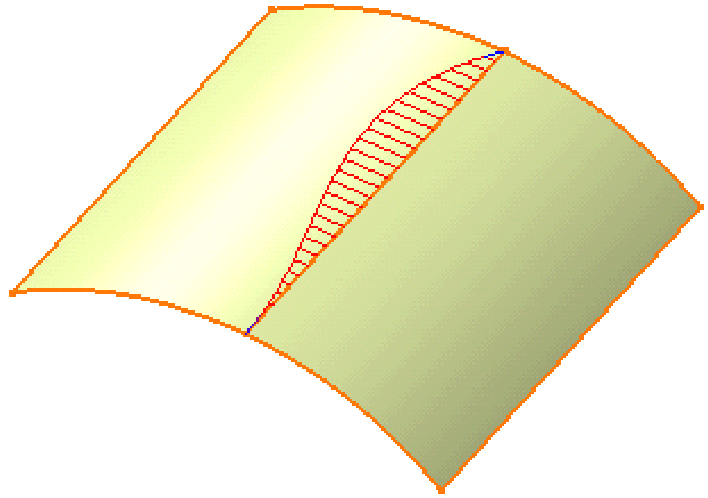
6. Check the **Information** button:

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).
The number of spikes corresponds to the number of points used for the computation:

- **Light**: 5 spikes are displayed.



This mode enables to obtain consistent results with the visualization of sharp edges.
An edge is considered as sharp if its tangency deviation is higher than 0.5 degree. To only detect tangency deviations on sharp edges, specify a deviation of 0.5 degree minimum.

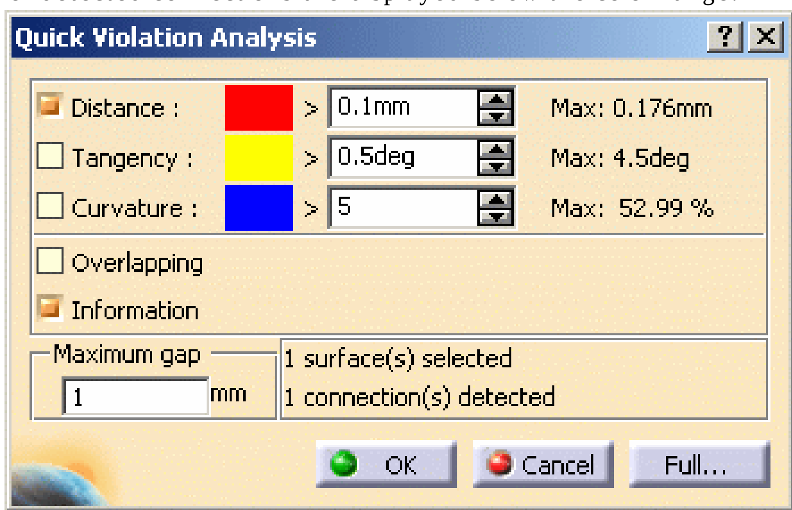
To visualize sharp edges, make sure the **View -> Render Style -> Shading with Edges and Hidden Edges** option is checked.

- **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 2 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.



The comb is no longer displayed.
The Connect Checker dialog box changes to this dialog box.

The **Maximum gap** and information are retained from the full analysis.
The maximum deviation value is also displayed on the geometry.

You can use the check button to select one or several analyses (up to three). As a consequence, the colorful area displaying the deviation tolerance between the surfaces shows the continuity whose value is the lowest.

In the case you select several types of continuity, the **Information** button is greyed out.

- 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.

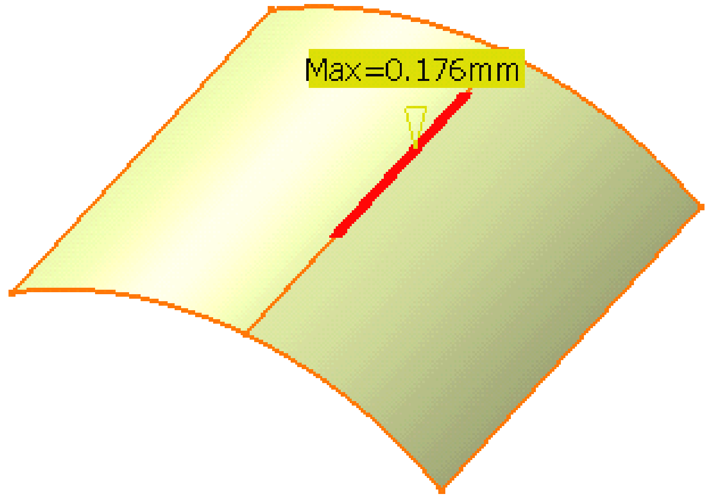
- You can check the **Information** button to display the minimum and maximum values in the 3D geometry, or uncheck it to hide the values.

In P1 mode, only the quick analysis is available.

8. Use the spinners to define the deviation 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.

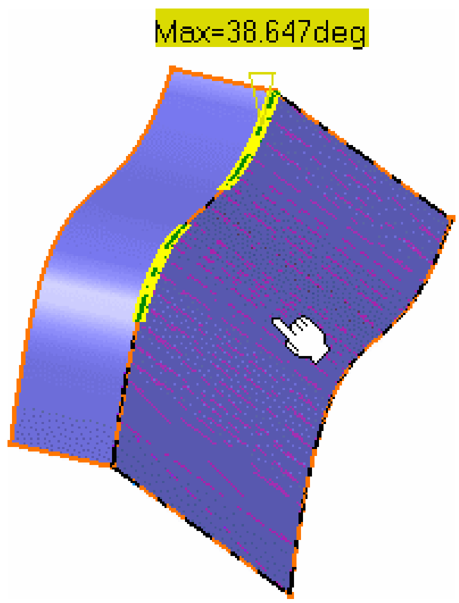
If you do not wish to create the analysis, simply click **Cancel**.

- 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:

  (|C2 - C1|) / ((|C1 + C2|) / 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:



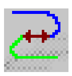
- You can create an analysis on an entire geometrical set simply by selecting it in the specification tree.

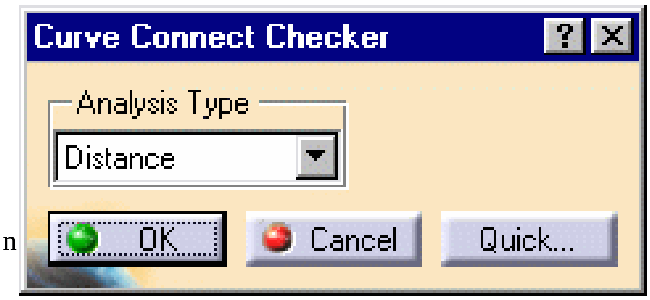
Checking Connections Between Curves

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.

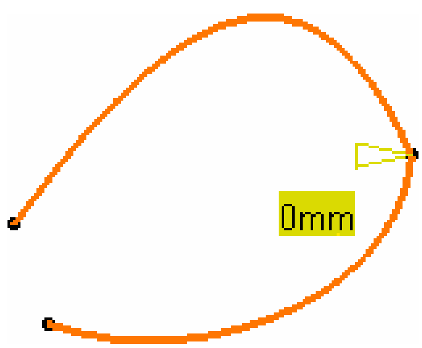
- **Distance**: the values are expressed in millimeters
- **Tangency**: the values are expressed in degrees
- **Curvature**: the values are expressed in percentage
- **Overlapping**: the system detects overlapping curves

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.

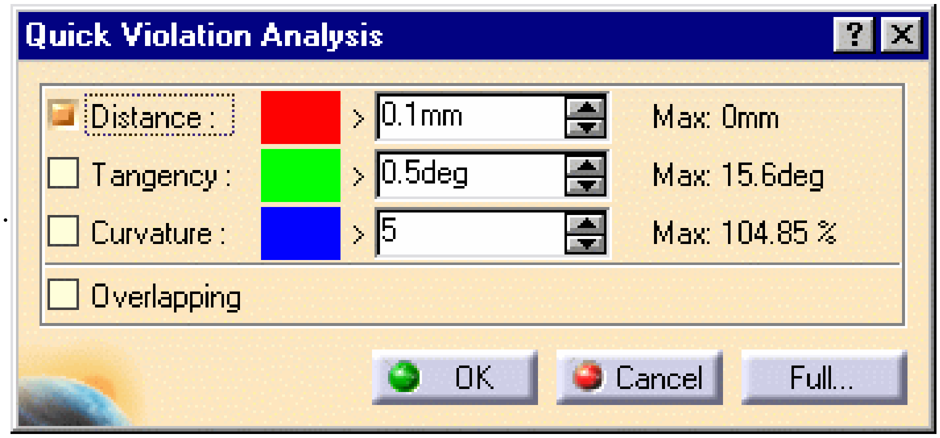


You can choose the type of analysis to be performed using the combo: distance, tangency or curvature.

In P1 mode, only this mode is available (no quick mode available).

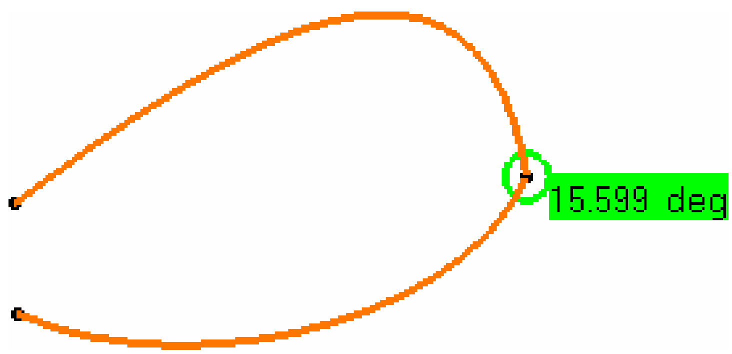
P2 This step is P2 only for Wireframe and Surface.

3. Press the Quick button.



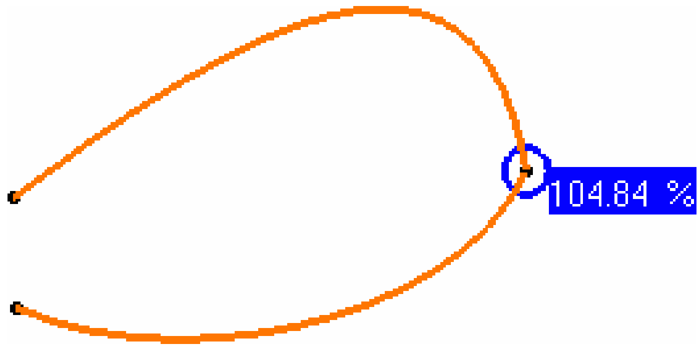
The dialog box changes along with the text on the geometry..
With our example, the text in the geometry disappears because the distance between the two curves is smaller than the set Distance value.

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.

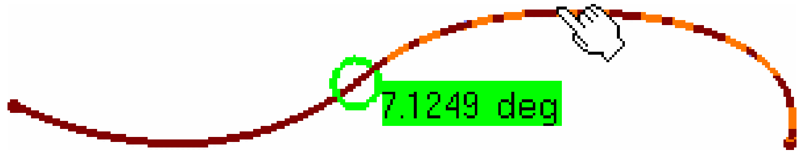
The analysis (identified as Curve Connection Analysis.x) is added to the specification tree.

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**.

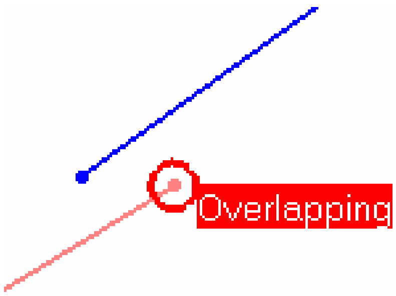
- Double-click the Curve Connection Analysis from the specification tree to edit it.

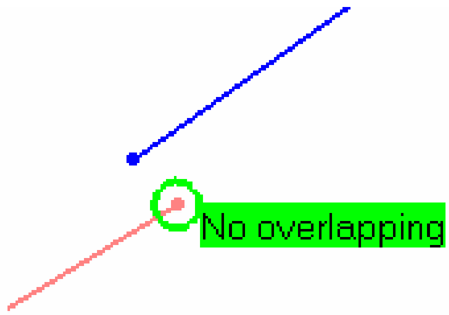
- You can analyze internal edges of a element, such as a Join for example, by selecting only one of the initial elements:



- 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.

Performing a Draft Analysis

- This command is not available with the Generative Shape Design 1 product.
- Used in Part Design workbench, this command requires the P2 configuration mode.

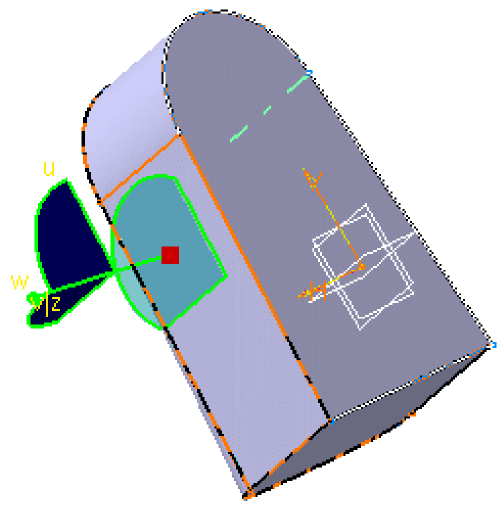
This task shows how to analyze the draft angle on a surface.
The Draft Analysis command enables you to detect if the part you drafted will be easily removed.

This type of analysis is performed based on color ranges identifying zones on the analyzed element where the deviation from the draft direction at any point, corresponds to specified values.
These values are expressed in the unit as specified in **Tools -> Options -> General -> Parameters -> Unit** tab.
You can modify them by clicking on their corresponding arrow or by entering a value directly in the field.
Open the DraftAnalysis1.CATPart document.

- The visualization mode should be set to **Shading With Edges** in the **View -> Render Style** command
- The discretization option should be set to a maximum: in **Tools -> Options -> Display -> Performances**, set the 3D **Accuracy -> Fixed** option to 0.01.
- Check the **Material** option in the **View -> Render Style -> Customize View** command to be able to see the analysis results on the selected element. Otherwise a warning is issued.
- Uncheck the **Highlight faces and edges** option in **Tools -> Options -> Display -> Navigation** to disable the highlight of the geometry selection.

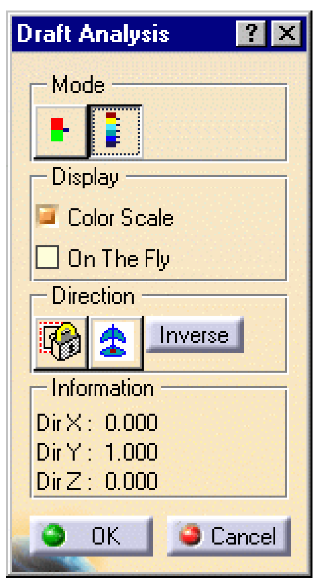
1. Select a surface.

   It is highlighted.



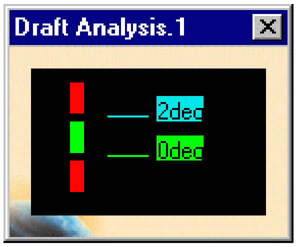
2. Click the **Draft Analysis** icon in the Shape Analysis toolbar.

The Draft Analysis dialog box is displayed. It gives information on the display (color scale), the draft direction and the direction values.
The Draft Analysis.1 dialog box showing the color scale and identifying the maximum and minimum values for the analysis is displayed too.

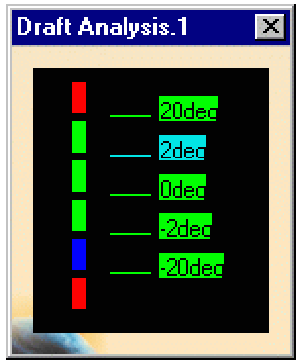


Mode option

The mode option lets you choose between a quick and a full analysis mode. These two modes are completely independent.
The default mode is the quick mode. It simplifies the analysis in that it displays only three color ranges.



Quick mode

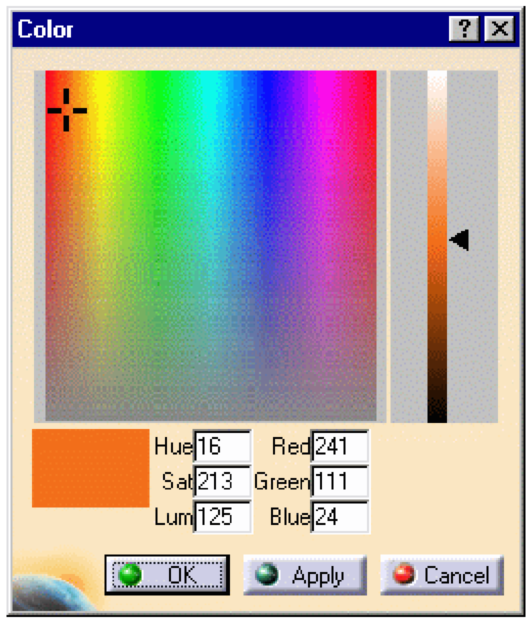


Full mode

P1 In P1 mode, only the quick mode is available.

- You can right-click on a color in the color scale to display the contextual menu:

- Edit: it allows you to modify the values in the color range to highlight specific areas of the selected surface. The Color dialog box is displayed allowing the user to modify the color range.

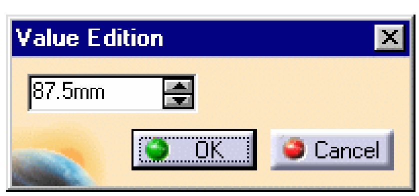


- **Unfreeze**: it allows you to perform a linear interpolation between non defined colors. The unfreezed values are no longer highlighted in green.

- **No Color**: it can be used to simplify the analysis, because it limits the number of displayed colors in the color scale. In this case, the selected color is hidden, and the section of the analysis on which that color was applied takes on the neighboring color.

- You can also right-click on the value to display the contextual menu:

- **Edit**: it allows you to modify the edition values. The Value Edition dialog box is displayed: enter a new value (negative values are allowed) to redefine the color scale, or use the slider to position the distance value within the allowed range, and click OK.
The value is then frozen, and displayed in a green rectangle.



- **Use Max/Use Min** : it allows you to evenly distribute the color/value interpolation between the current limit values, on the top/bottom values respectively, rather than keeping it within default values that may not correspond to the scale of the geometry being analyzed. Therefore, these limit values are set at a given time, and when the geometry is modified after setting them, these limit values are not dynamically updated.

The Use Max contextual item is only possible if the maximum value is higher or equal to the medium value. If not, you first need to unfreeze the medium value.

Only the linear interpolation is allowed, meaning that between two set (or frozen) colors/values, the distribution is done progressively and evenly.

The color scale settings (colors and values) are saved when exiting the command, meaning the same values will be set next time you edit a given draft analysis capability.
However, new settings are available with each new draft analysis.

Display option

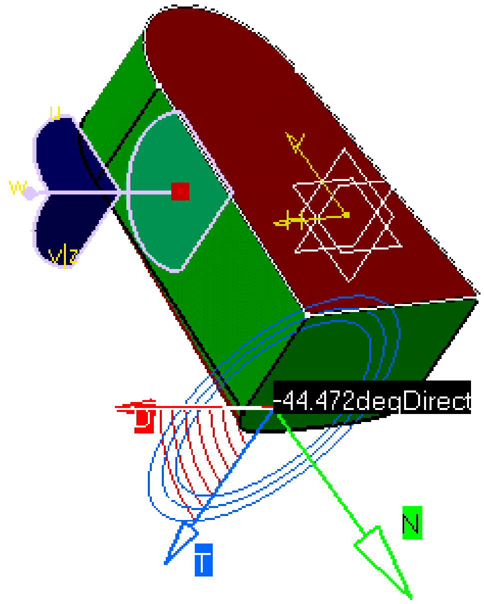
- Uncheck the **Color Scale** checkbox to remove the Draft Analysis.1 dialog box.

This dialog box only appears in edition mode.

- Activate the **On the fly** checkbox and move the pointer over the surface.

  This option enables you to perform a local analysis.

  Arrows are displayed under the pointer, identifying the normal to the surface at the pointer location (green arrow), the draft direction (red arrow), and the tangent (blue arrow). As you move the pointer over the surface, the display is dynamically updated.



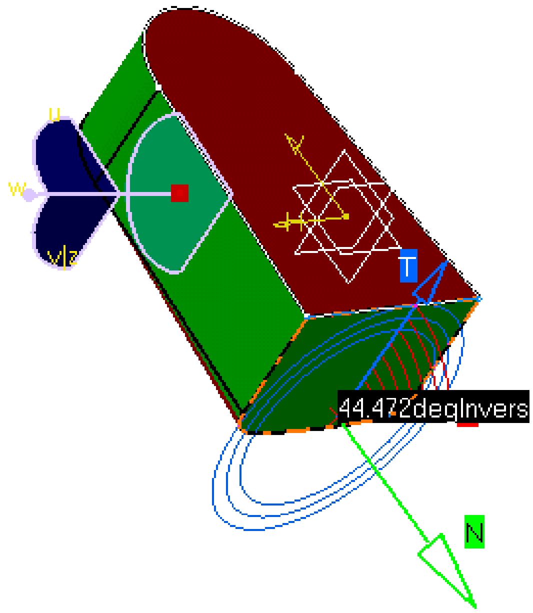
Furthermore, circles are displayed indicating the plane tangent to the surface at this point.
The displayed value indicates the angle between the draft direction and the tangent to the surface at the current point.
It is expressed in the units set in using the **Tools -> Options -> General -> Parameters -> Units** tab.

The **On the fly** analysis can only be performed on the elements of the current part.

Note that you can activate the **On the fly** option even when not visualizing the materials. It gives you the tangent plane and the deviation value.

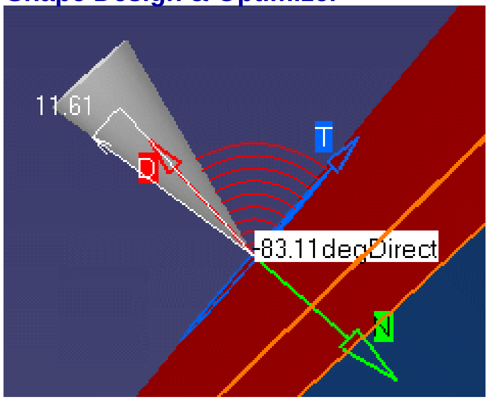
- Click the **Inverse** button to automatically reverse the draft direction:

  When several elements are selected for analysis, the draft direction is inverted for each element when the button is clicked.

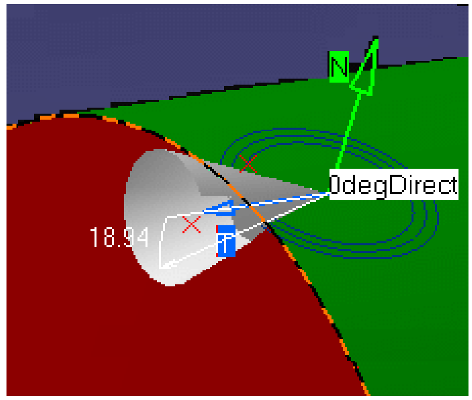


In case of an obviously inconsistent result, do not forget to invert locally the normal direction via the **Inverse** button.

The manipulator on the draft direction allows you to materialize the cone showing the angle around the direction.



Direction in the cone

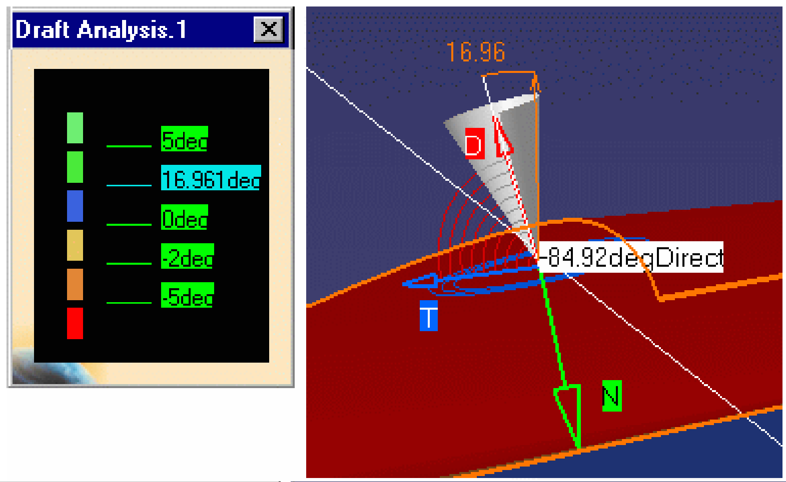


Direction out of the cone

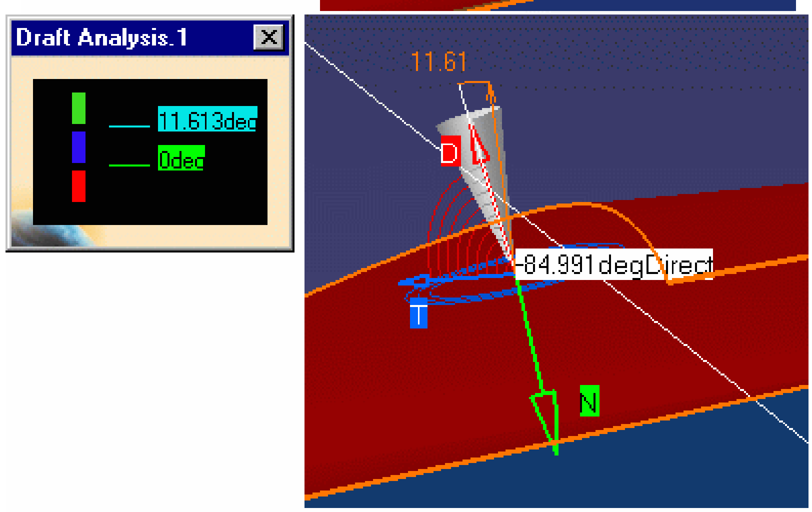
- Right-click the cone angle to display the Angle Tuner dialog box.
  When you modify the angle using the up and down arrows, the value is automatically updated in the color scale and in the geometry.

Please note that you cannot modify the angle below the minimum value or beyond the maximum value.

Full mode

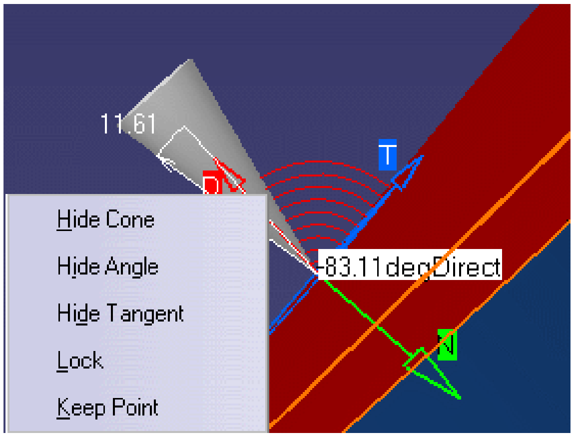


Quick mode



- Right-click the Direction vector to display the contextual menu.
  It allows you to:

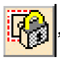
  - hide / show the cone
  - hide / show the angle
  - hide / show the tangent
  - lock / unlock the analysis position
  - keep the point at this location
    A Point.xxx appears in the specification tree.

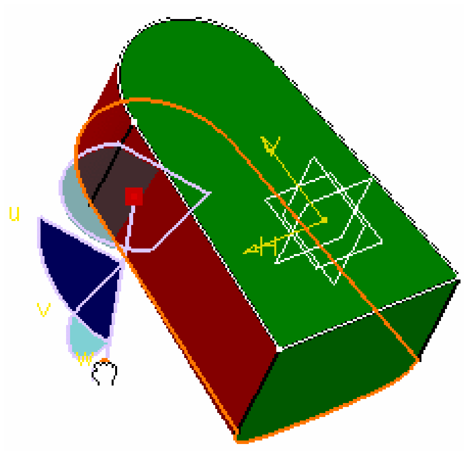


In P1 mode, the Keep Point option is not available.

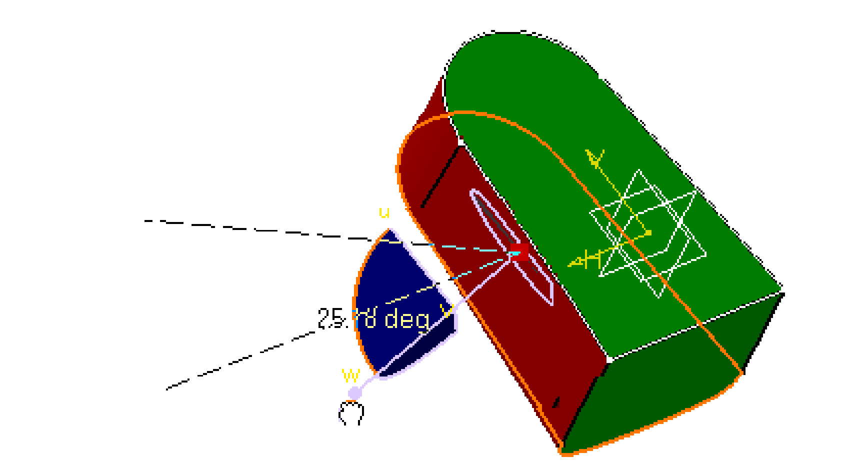
Direction

By default the analysis is locked, meaning it is done according to a specified direction: the compass w axis.
In P1 mode, the default analysis direction is the general document axis-system's z axis.

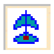
- Click the **Locked direction** icon , and select a direction (a line, a plane or planar face which normal is used), or use the compass manipulators, when available.

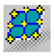


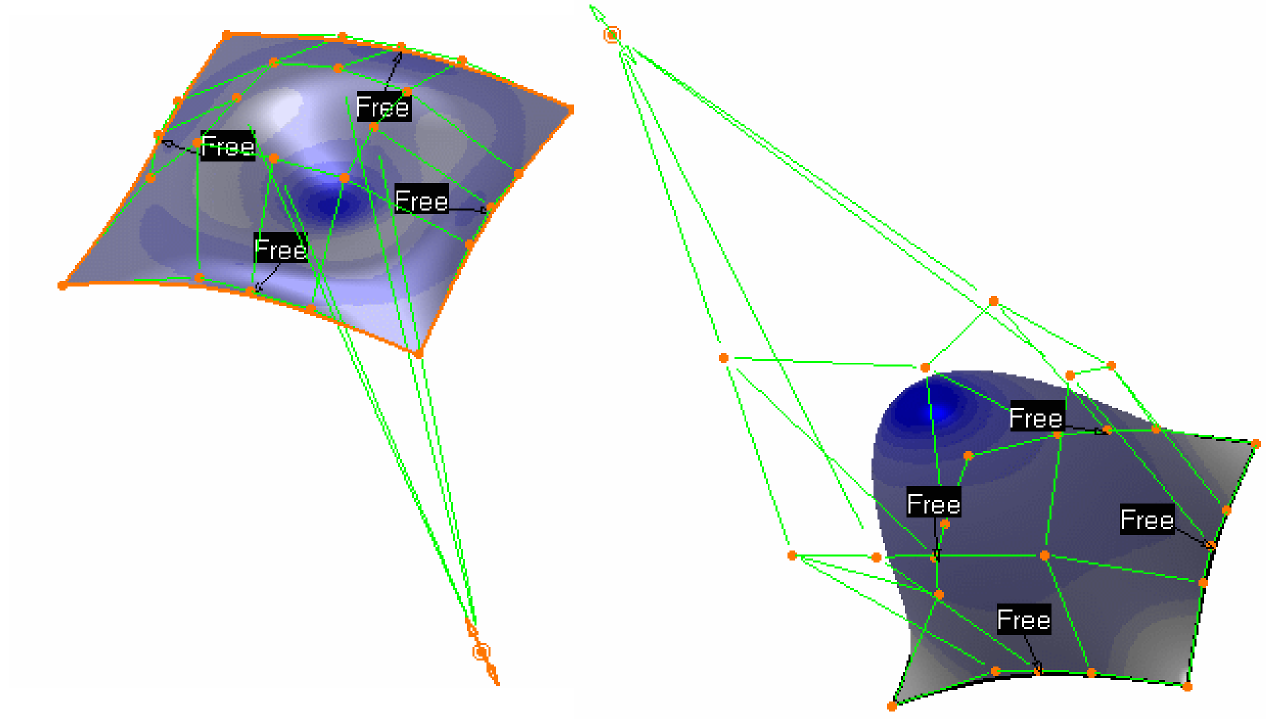
Using the compass manipulators



Selecting a specific direction

- Click the **Compass** icon  to define the new current draft direction.
  The compass lets you define the pulling direction that will be used from removing the part.

You can display the control points by clicking the **Control Points**  icon, yet the draft analysis is still visible, then allowing you to check the impact of any modification to the surface on the draft analysis.



3. Once you have finished analyzing the surface, click **OK** in the Draft Analysis dialog box.

The analysis (identified as Draft Analysis.x) is added to the specification tree.

This capability is not available in P1 mode.

- Note that settings are saved when exiting the command, and redisplayed when you select the **Draft Analysis** icon again.
- Be careful, when selecting the direction, not to deselect the analyzed element.
- A draft analysis can be performed just as well on a set of surfaces.
- If an element belongs to an analysis, it cannot be selected simultaneously for another analysis, you need to remove the current analysis by deselecting the element to be able to use it again.
- In some cases, even though the rendering style is properly set, it may happen that the analysis results are not visible. Check that the geometry is up-to-date, or perform an update on the involved geometric element(s).
- The analysis results depend of the current object. May you want to change the scope of analysis, use the Define in Work object contextual command.

Performing a Surfacic Curvature Analysis

- This command is not available with the Generative Shape Design 1 product.
- Used in Part Design workbench, this command requires the P2 configuration mode.

This task shows how to analyze the mapping curvature of a surface.

Open the SurfacicAnalysis1.CATPart document.

- The discretization option should be set to a maximum: in **Tools -> Options -> Display -> Performances**, set the 3D **Accuracy -> Fixed** option to 0.01.
- Check the **Material** option in the **View -> Render Style -> Customize View** command to be able to see the analysis results on the selected element.

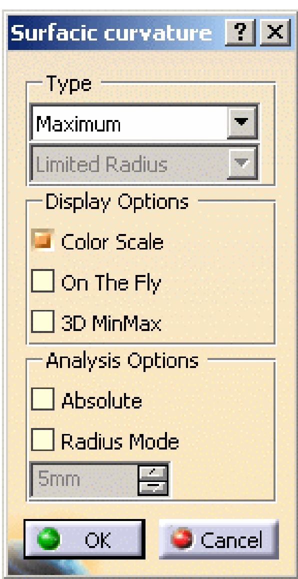
You can now perform an analysis on the fly even if the **Material** option is not checked. No warning message is issued as long as no element is selected.

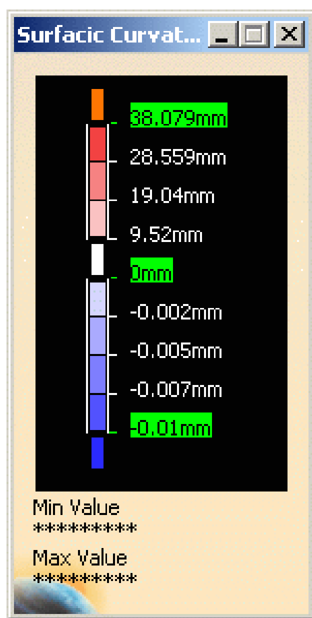
- Uncheck the **Highlight faces and edges** option in **Tools -> Options -> General -> Display -> Navigation** to disable the highlight of the geometry selection.

1. Select a surface.
2. Click the **Surfacic Curvature Analysis** icon in the Shape Analysis toolbar.

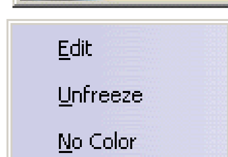
The Surfacic curvature dialog box is displayed, and the analysis is visible on the selected element. It gives information on the display (color scale), the draft direction and the direction values.

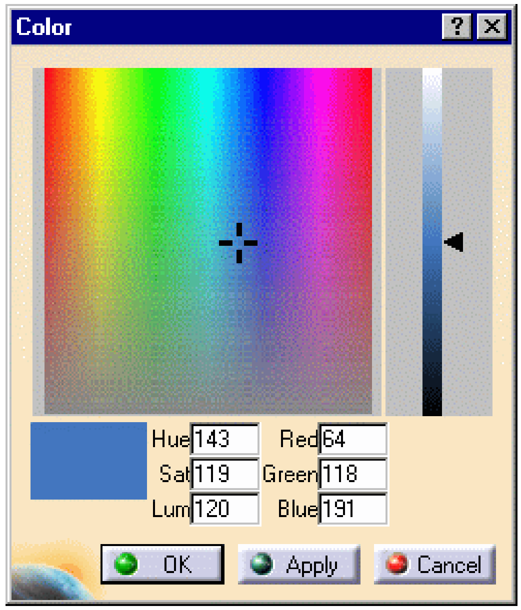
The Surfacic Curvature.1 dialog box showing the color scale and identifying the maximum and minimum values for the analysis is displayed too.





- You can right-click on a color in the color scale to display the contextual menu:



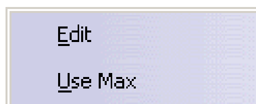


- **Edit**: it allows you to modify the values in the color range to highlight specific areas of the selected surface. The Color dialog box is displayed allowing the user to modify the color range.

- **Unfreeze**: it allows you to perform a linear interpolation between non defined colors. The unfreezed values are no longer highlighted in green.

- **No Color**: it can be used to simplify the analysis, because it limits the number of displayed colors in the color scale. In this case, the selected color is hidden, and the section of the analysis on which that color was applied takes on the neighboring color.

- You can also right-click on the value to display the contextual menu:



- **Edit**: it allows you to modify the edition values. The Value Edition dialog box is displayed: enter a new value (negative values are allowed) to redefine the color scale, or use the slider to position the distance value within the allowed range, and click OK.
The value is then frozen, and displayed in a green rectangle.

- **Use Max/Use Min**: it allows you to evenly distribute the color/value interpolation between the current limit values, on the top/bottom values respectively, rather than keeping it within default values that may not correspond to the scale of the geometry being analyzed. Therefore, these limit values are set at a given time, and when the geometry is modified after setting them, these limit values are not dynamically updated.

The Use Max contextual item is only possible if the maximum value is higher or equal to the medium value. If not, you first need to unfreeze the medium value.

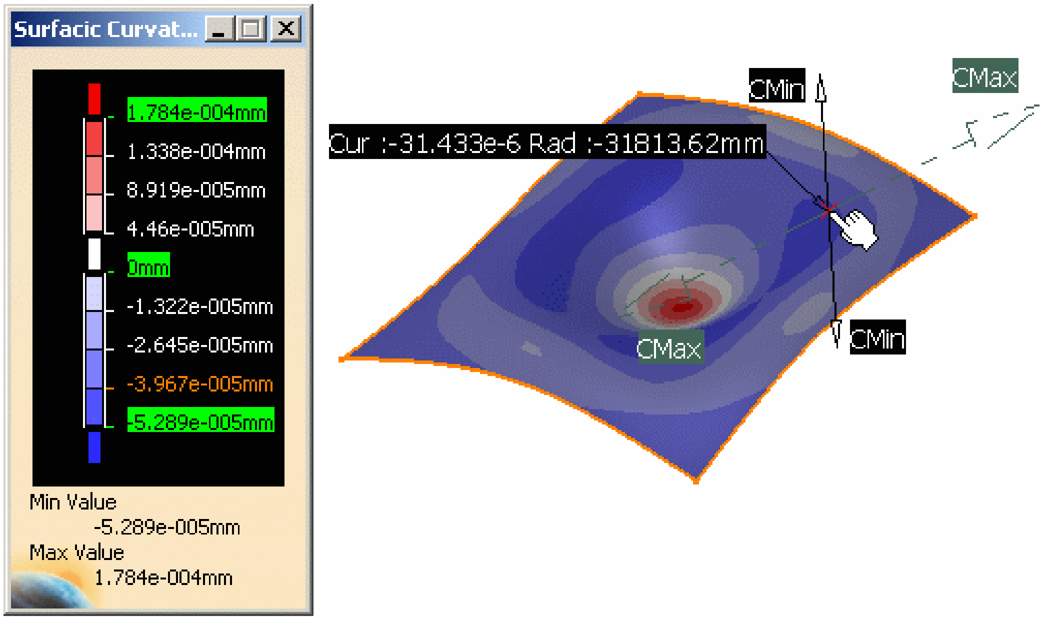
- **Interpolation**: by default the interpolation is linear.

Type

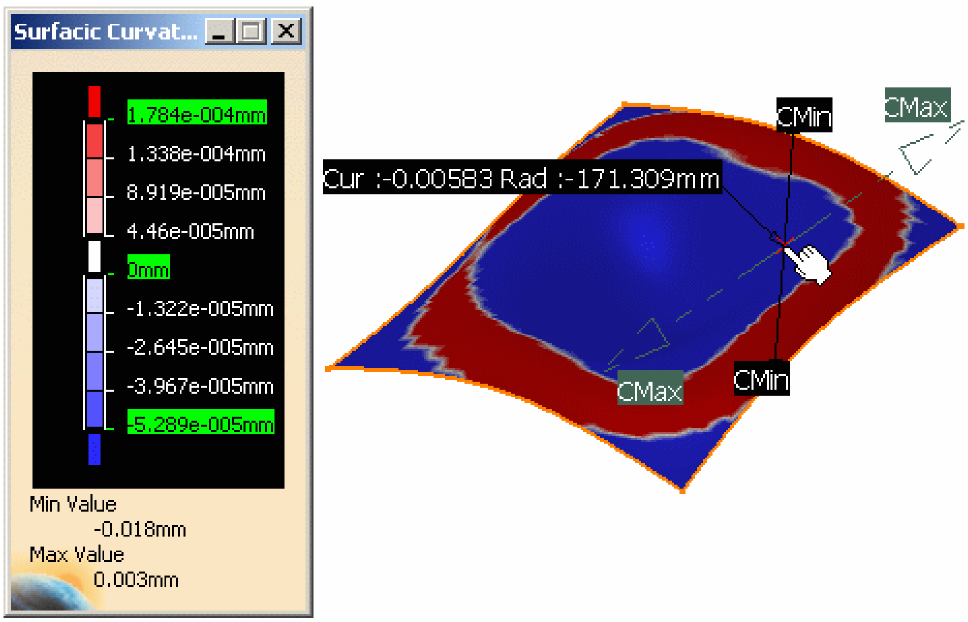
3. Select the type analysis:

In the following examples, we defined minimum and maximum values and used the on the fly option (except for Limited and Inflection Area type)

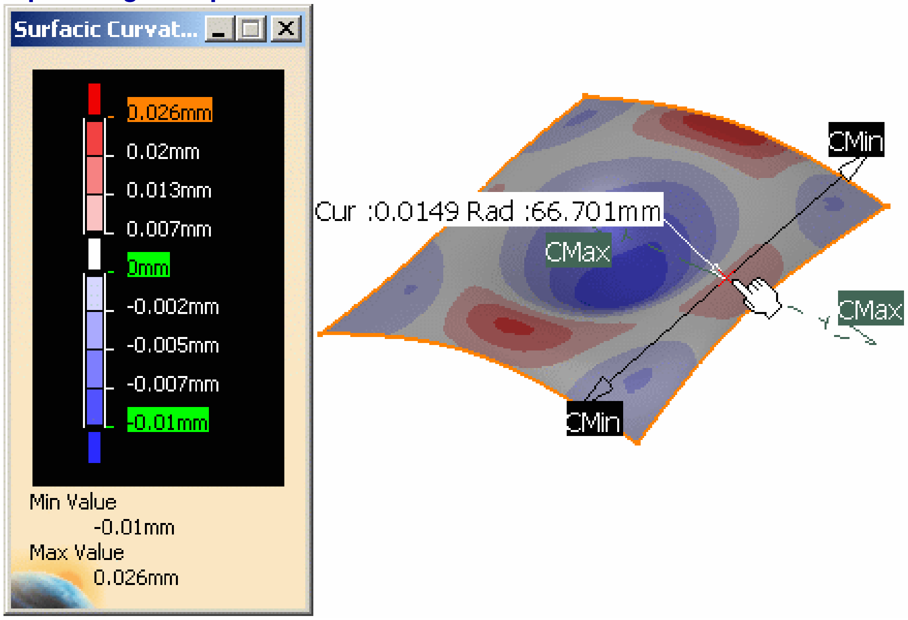
- **Gaussian**



- **Minimum**: to display the minimum curvature value

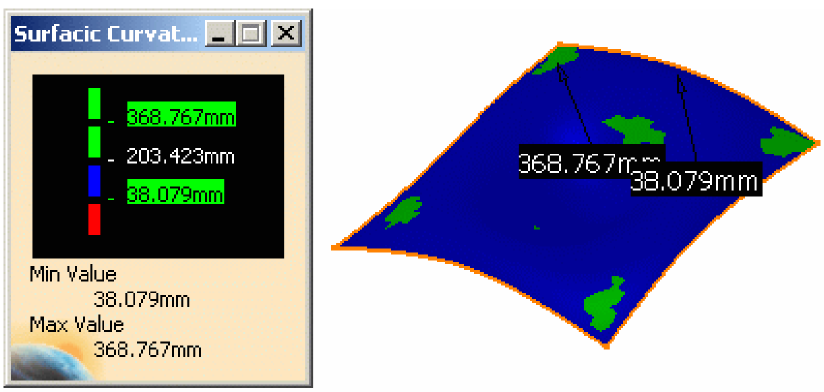


- **Maximum**: to display the maximum curvature value



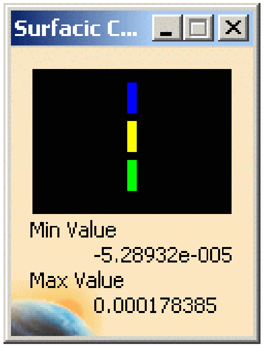
- **Limited**: the quick mode is displayed and the **Limited Radius** type is selected.

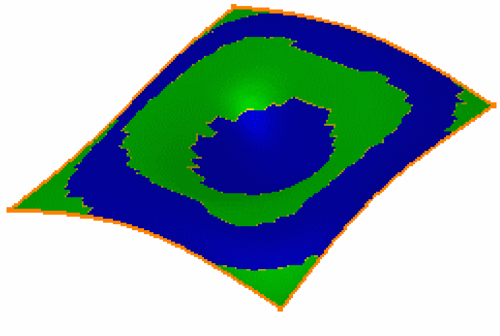
  In the Surfacic curvature dialog box, you are able to modify the radius value using the up and down arrows. The value is automatically updated in the color scale.



- **Inflection Area**: enables to identify the curvature orientation:
  - In green: the areas where the minimum and maximum curvatures present the same orientation
  - In blue: the areas where the minimum and maximum curvatures present opposite orientation

See also Creating Inflection Lines. Note that these inflection lines are always created within the green area, i.e. when the curvature orientation is changing.





Display options

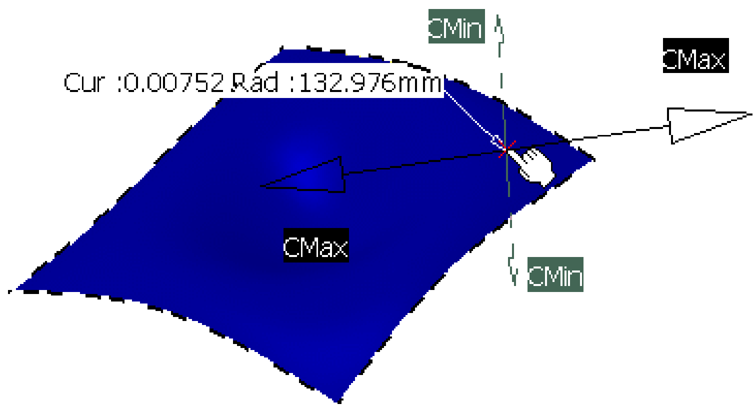
- Uncheck the **Color Scale** checkbox to remove the Surfacic Curvature Analysis.1 dialog box.

- Activate the **On the fly** checkbox and move the pointer over the surface.

  This option enables to perform a local analysis.

  The curvature and radius values are displayed under the pointer, as well as the minimum and maximum curvature values and the minimum and maximum curvature directions. As you move the pointer over the surface, the display is dynamically updated.

  The values are expressed in the units set in using the **Tools -> Options -> General -> Parameters -> Units** tab.



The displayed values may vary from the information displayed as the **Use Max / Use Min** values, as it is the precise value at a given point (where the pointer is) and does not depend on the set discretization.

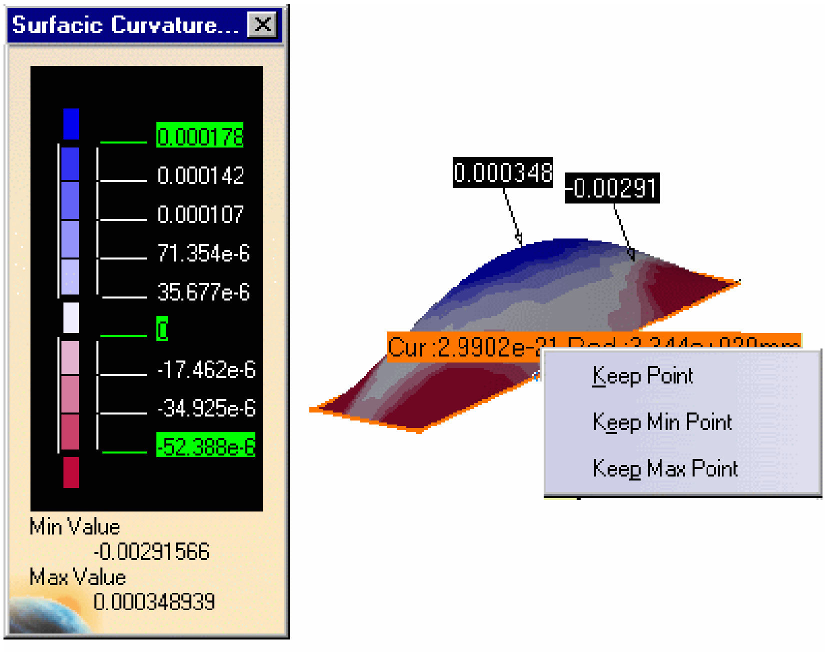
- The **On the fly** analysis can only be performed on the elements of the current part. It is not available with the **Inflection Area** analysis type.
- You cannot snap on point when performing a local on the fly analysis.

Right-click a point to display the contextual menu. It allows you to :

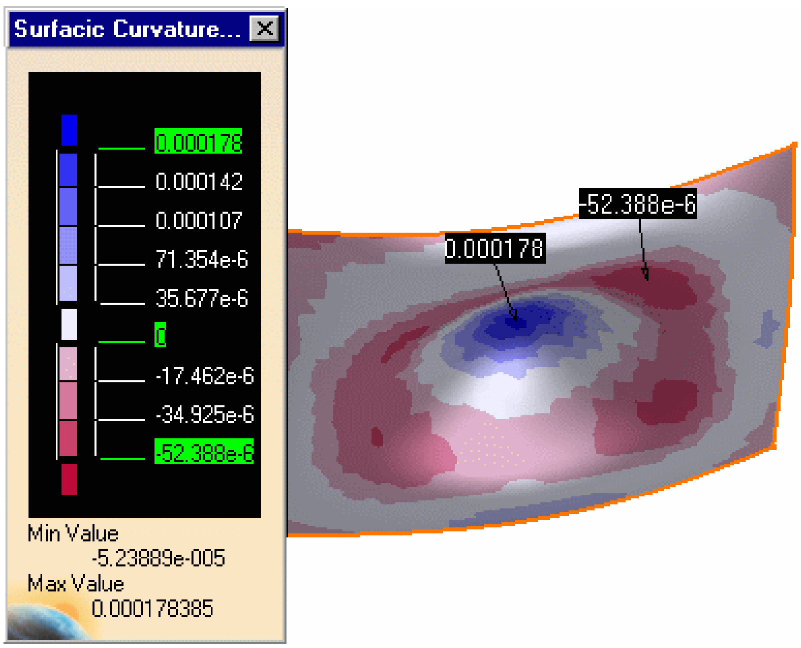
- keep the point at this location (under the pointer)
- keep the point corresponding to the minimum value
- keep the point corresponding to the maximum value

A Point.xxx appears in the specification tree.

In P1 mode, this contextual menu is not available.

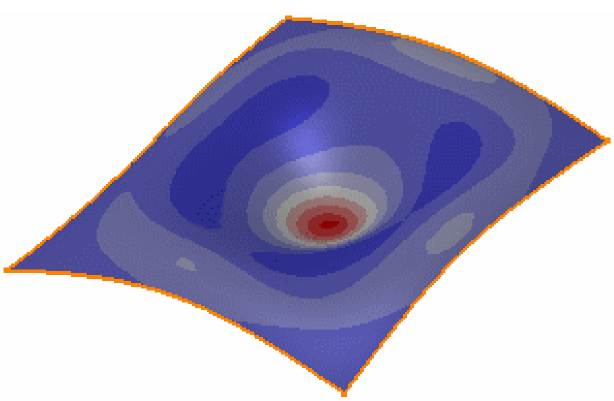


- Activate the **3D MinMax** checkbox to locate the minimum and maximum values for the selected analysis type.

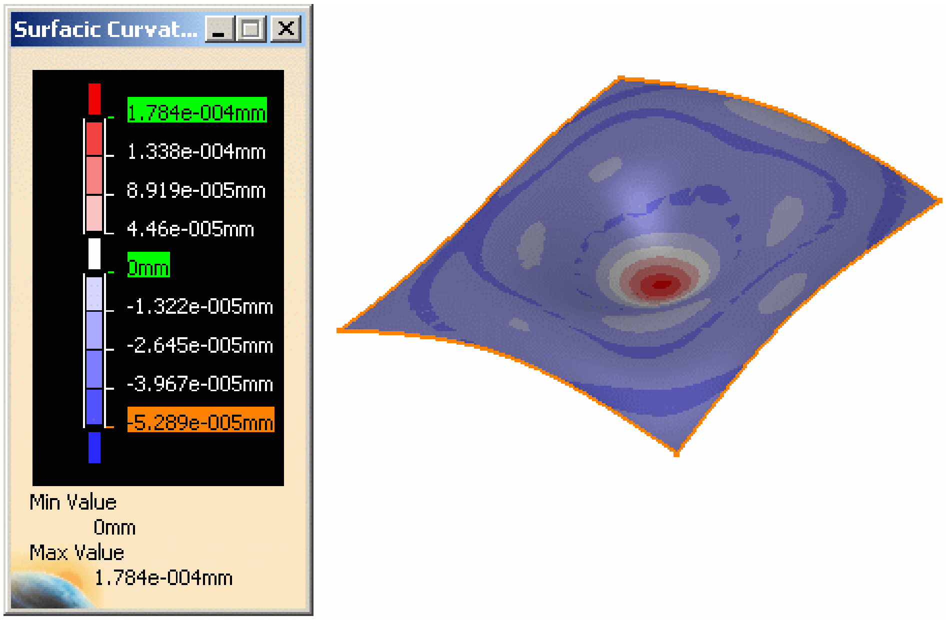


Analysis options

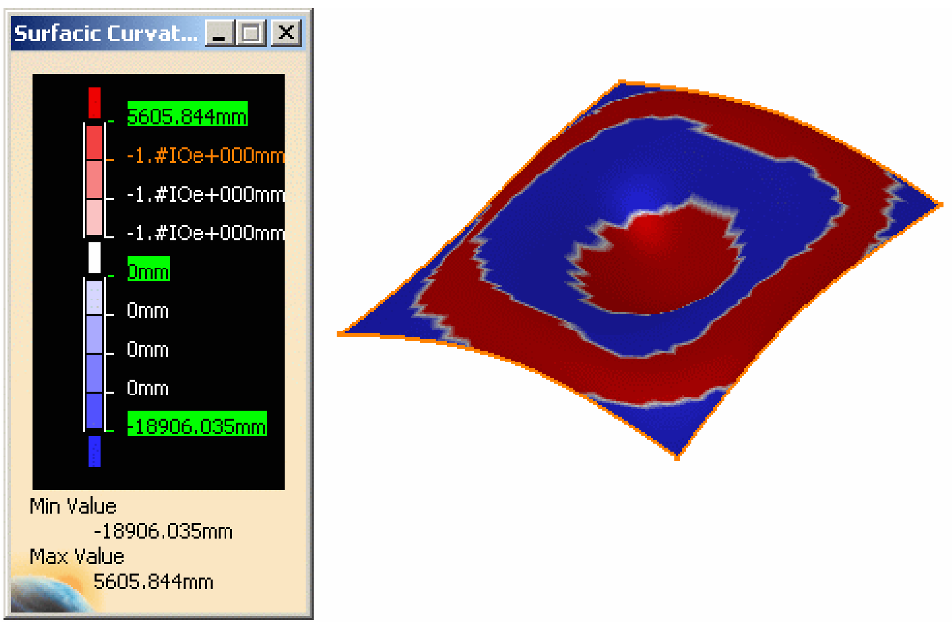
- Uncheck both **Positive only** and **Radius Mode** analysis options to get analysis values as curvature values



- Check the **Positive only** analysis option to get analysis values as positive values.



- Check the **Radius Mode** analysis option to get analysis values as radius values.



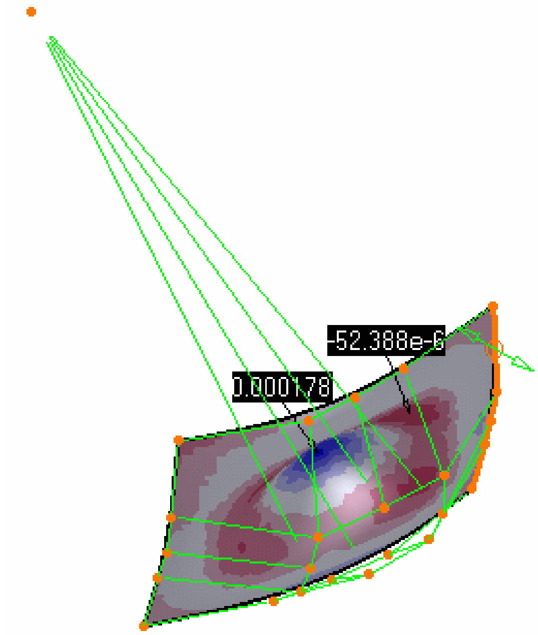
These options are not available with the **Limited** and **Inflection** Area analysis types.

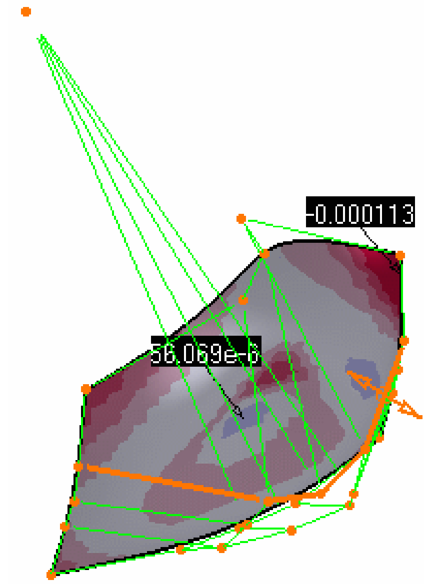
4. Once you have finished analyzing the surface, click **OK** in the Surfacic Curvature Analysis dialog box.

The analysis (identified as Surfacic Curvature Analysis.x) is added to the specification tree.

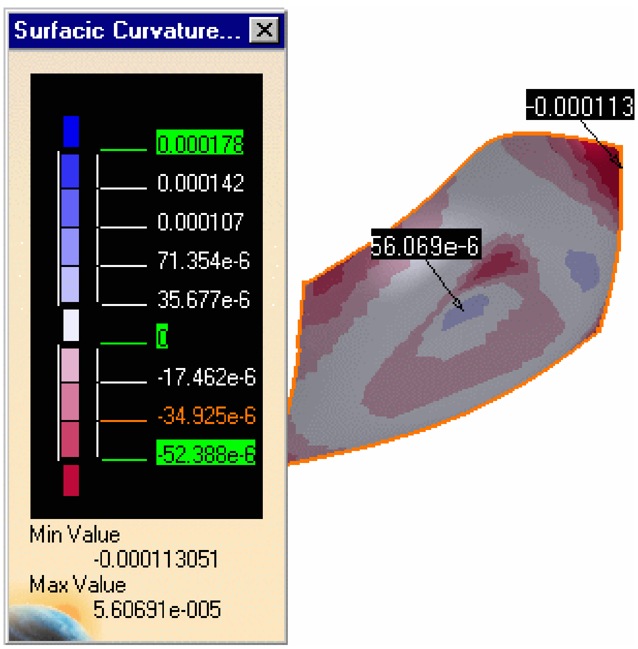
This capability is not available in P1 mode.

- You can display the control points by clicking the icon, still viewing the surfacic curvature analysis. This allows you to check the impact of any modification on the surface.
Here are examples using the 3D MinMax capability.

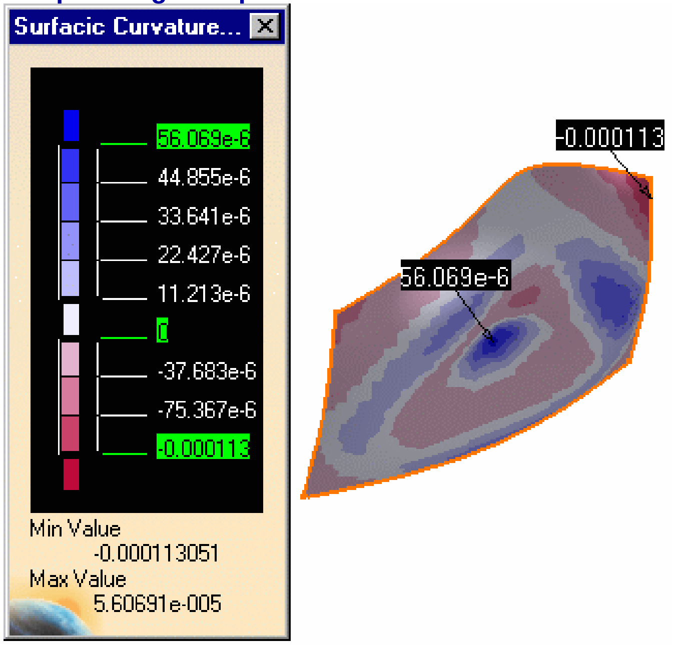




If you double-click the Surfacic Curvature Analysis.xxx in the specification tree, the Minimum and Maximum values are updated in the Surfacic Curvature.1 analysis but not in the color scale.



To update the values in the color scale, right-click the minimum value and the maximum value and select respectively Use Min / Use max from the contextual menu.



- Surfacic curvature analyses can be performed on a set of surfaces.
- If an element belongs to an analysis, it cannot be selected simultaneously for another analysis, you need to remove the current analysis by deselecting the element to be able to use it again.
- In some cases, even though the rendering style is properly set, it may happen that the analysis results are not visible. Check that the geometry is up-to-date, or perform an update on the involved geometric element(s).
- The analysis results depend of the current object. May you want to change the scope of analysis, use the Define in Work object contextual command.

- You can customize the values expressed in the color scale and in the 3D geometry.
  To do so, select the **Tools -> Options -> General -> Parameters and Measures -> Unit** command, then define or redefine the default units.
  For further information, refer to the Customizing Units chapter in the *CATIA Infrastructure User's Guide* documentation.

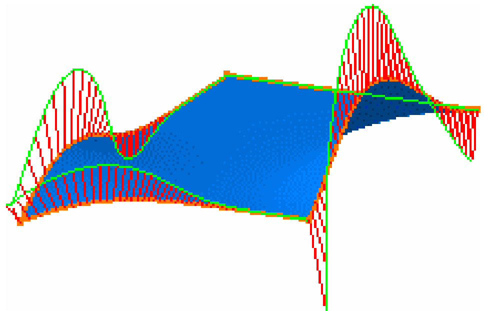
Performing a Curvature Analysis

This task shows how to analyze the curvature of curves, or surface boundaries.

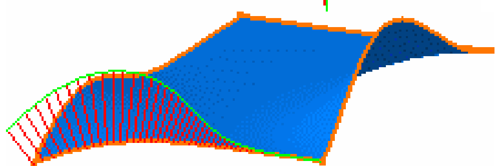
Open the Analysis1.CATPart document.

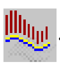
When analyzing surface boundaries:

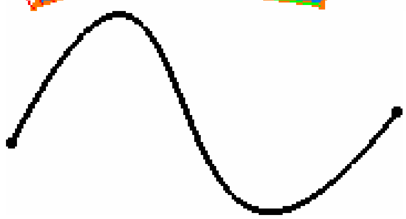
- if you select the surface, the analysis is performed on all its boundaries



- if you select a specific boundary, the analysis is performed only on this boundary.

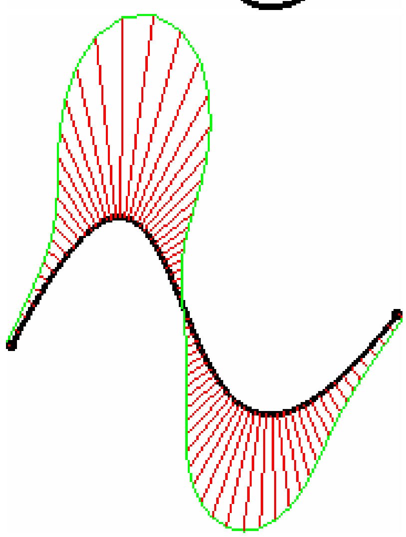


1. Click the **Porcupine Curvature Analysis** icon .



2. Select the curve.

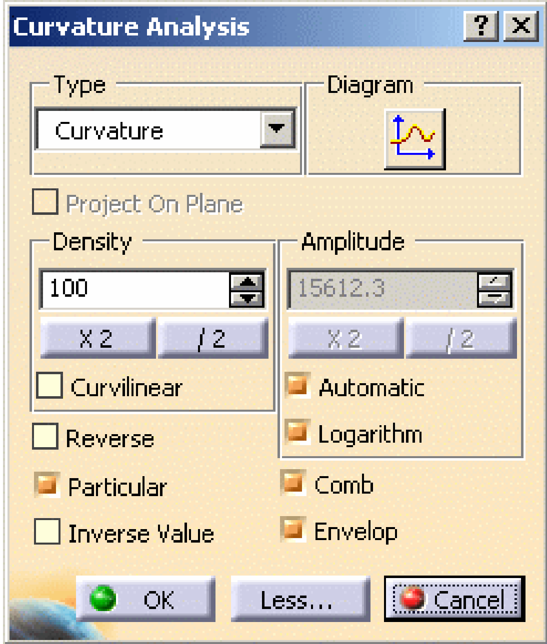
Automatically the curvature comb is displayed on the selected curve:



3. Define the analysis parameters in the Curvature Analysis dialog box.

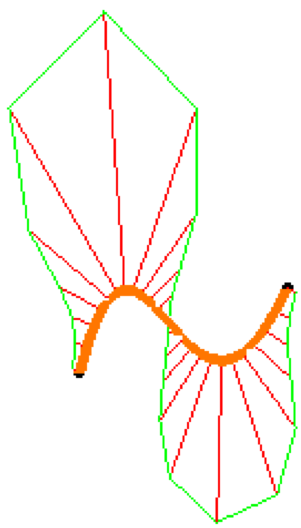
- Use the **Project on Plane** checkbox to analyze the projected curve in the selected plane referenced by the compass.

- If you uncheck the **Project On Plane** option, the analysis is performed according to the curve orientation. This is the default option.



4. Use the spinners to adjust the number of strikes and modify the density.

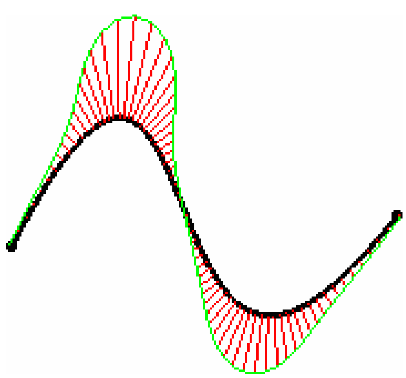
5. You can also decide to halve the number of spikes in the comb clicking as many times as wished the **/ 2** button.



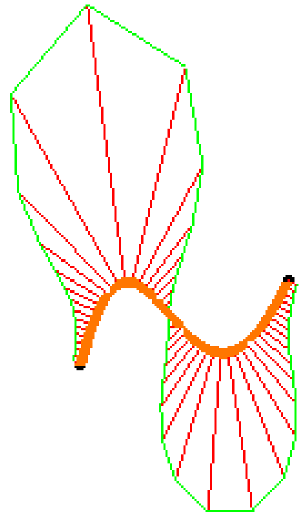
This option is particularly useful when the geometry is too dense to be read but the resulting curve may not be smooth enough for your analysis needs.

You could just as well double the number of spikes using the **X2** button.

6. Similarly, click the **/ 2** button to fine-tune the amplitude (size) of the spikes, and re-compute the analysis curve accordingly.

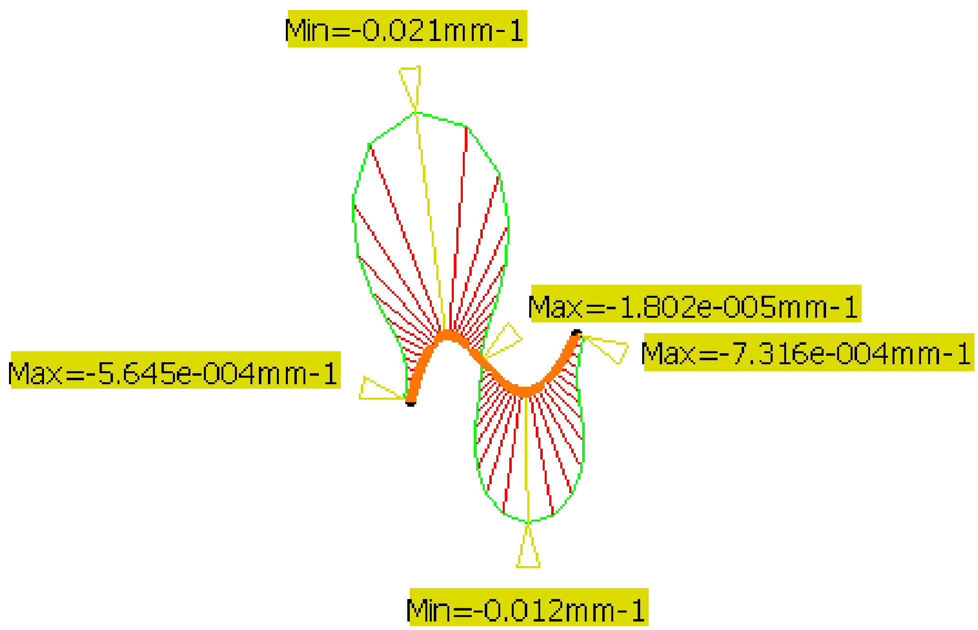


7. Click **Curvilinear** to switch from the Parametric discretization mode to the Curvilinear analysis. You will get something like this:



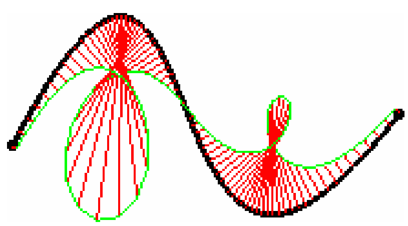
8. Check the **Automatic** option optimizes the spikes length so that even when zooming in or out, the spikes are always visible.

9. Check the **Logarithm** option to display the logarithmic values in the 3D geometry.

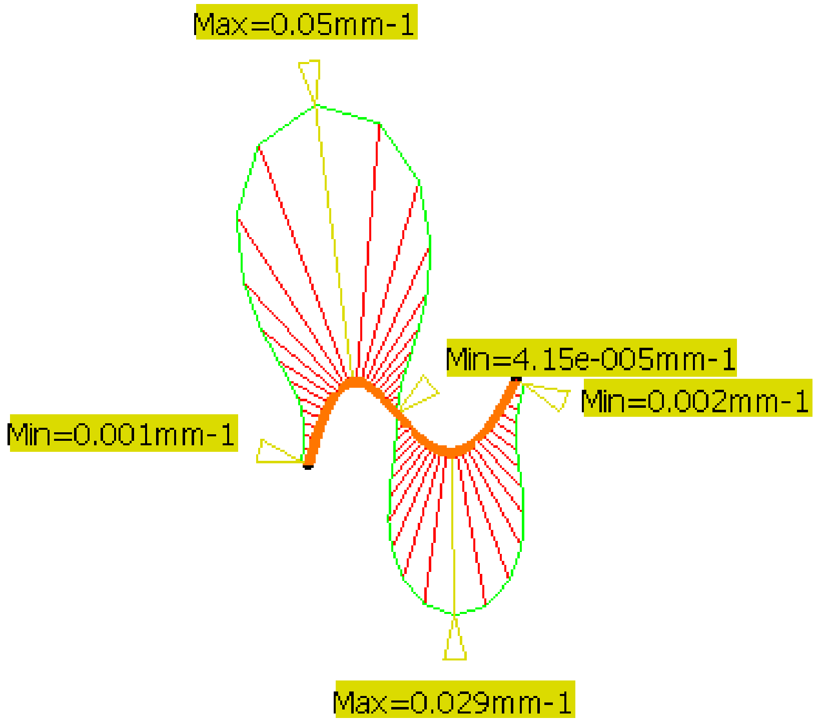


Displaying these values does not modify the analysis.

10. Click **Reverse**, you will get something like this:



That is the analysis opposite to what was initially displayed. This is useful when from the current viewpoint, you do not know how the curve is oriented.

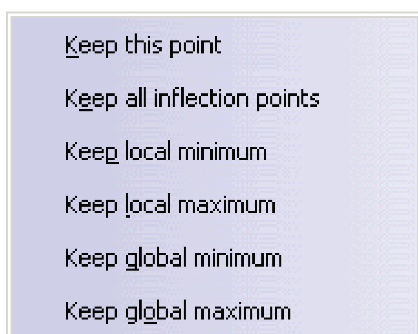


11. Use the **Particular** checkbox to display at anytime the minimum and the maximum points.

Inflection points are displayed only if the **Project on Plane** and Particular checkboxes are checked.

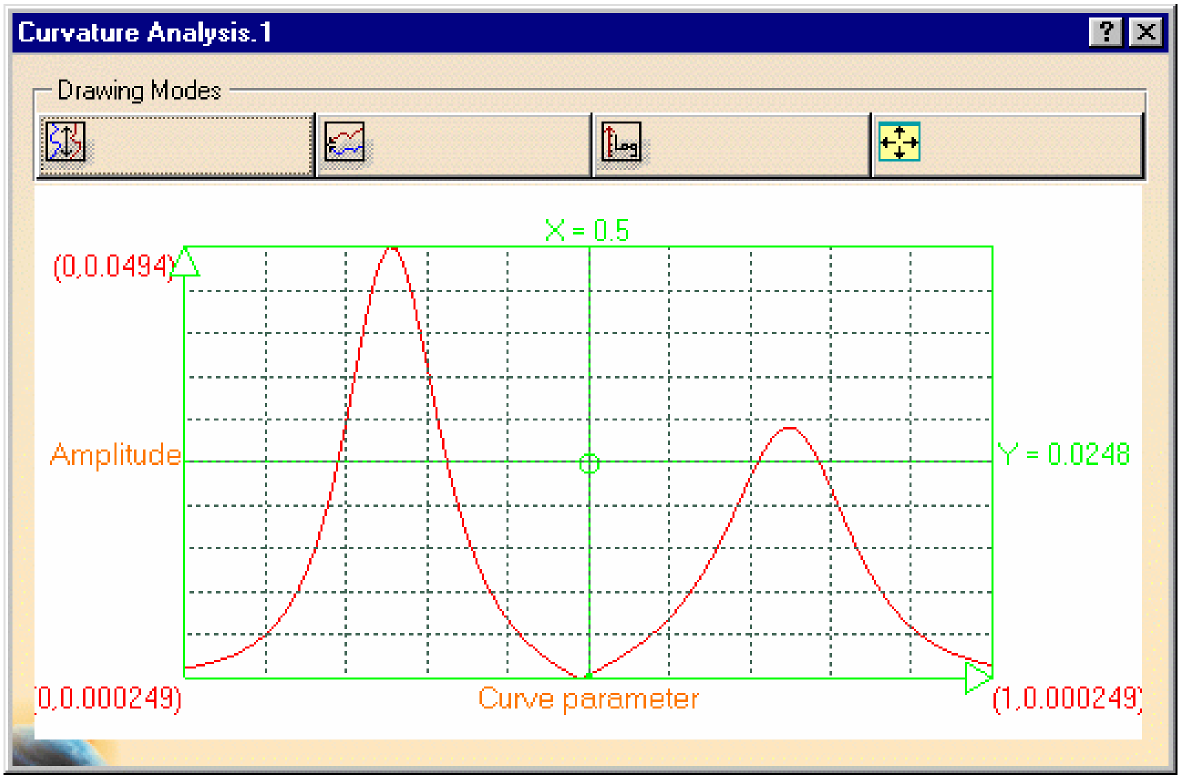
12. The **Inverse Value** checkbox displays the inverse value in **Radius**, if **Curvature** option is selected, or in **Curvature**, if **Radius** option is selected.

- You can right-click on any of the spikes and select **Keep this Point** to keep the current point at this location.
  A Point.xxx appears in the specification tree.
  If you check the **Particular** option, you have more options:
  - **Keep all inflection points**
  - **Keep local minimum** (corresponds to the absolute minimum under the running point)
  - **Keep local maximum** (corresponds to the absolute maximum under running point)
  - **Keep global minimum** (in case there are two curves, the point will be found on one or other of the curves)
  - **Keep global maximum** (in case there are two curves, the point will be found on one or other of the curves)



This option is only available in P2 mode in FreeStyle Shaper, Optimizer, and Profiler.

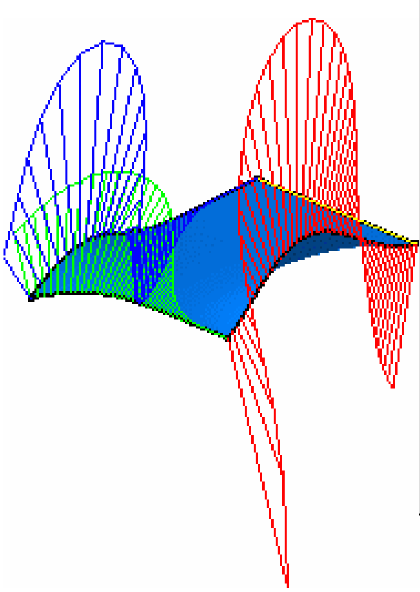
13. Finally, click the icon to display the curvature graph:

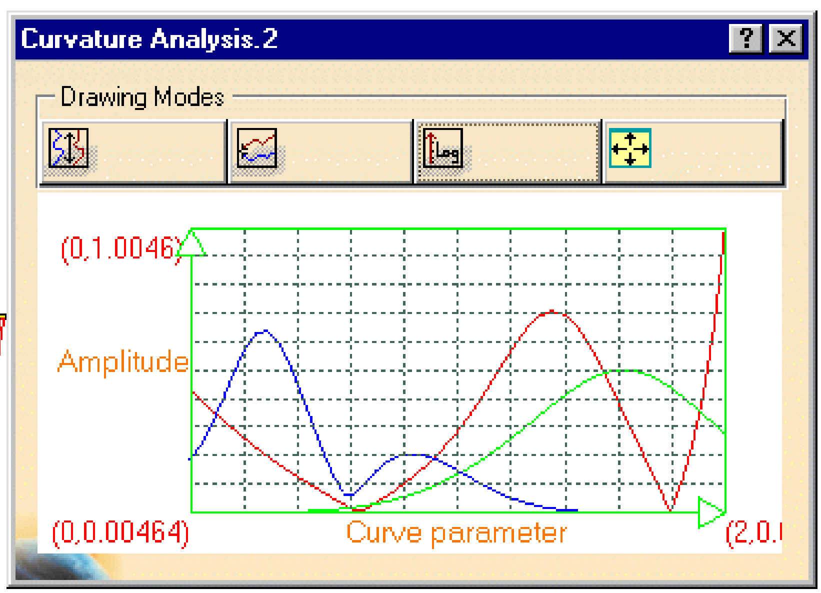


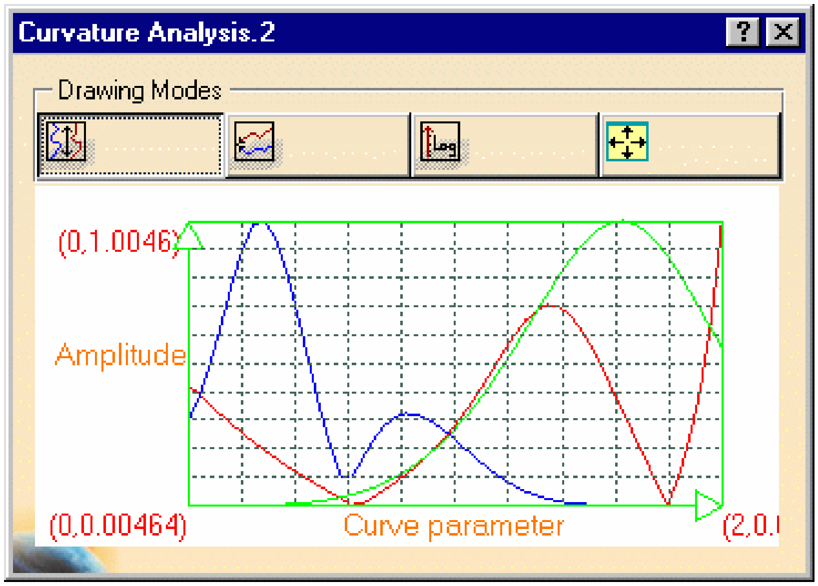
The curvature profile and amplitude of the analyzed curve is represented in this diagram.

When analyzing a surface or several curves, i.e. when there are several curvature analyses on elements that are not necessarily of the same size for example, you can use different options to view the analyses.

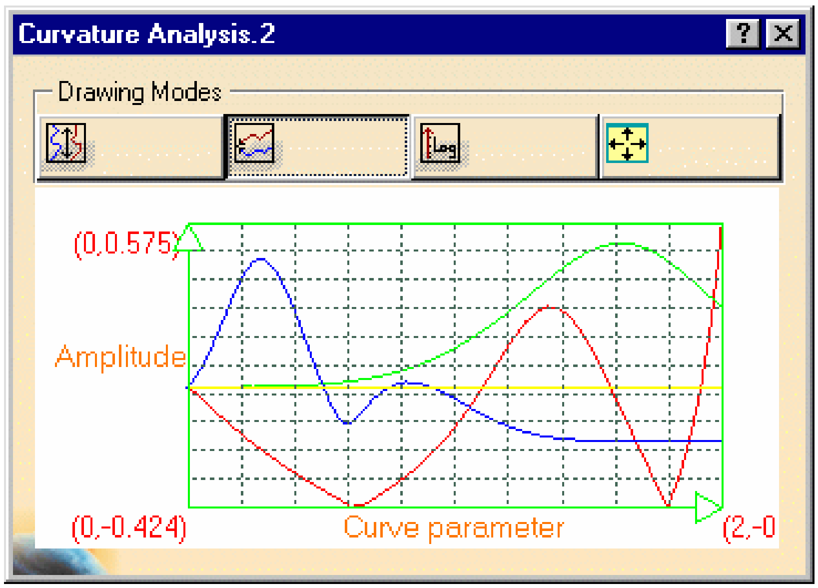
For example, when analyzing a surface, by default you obtain this diagram, where the curves color match the ones on the geometry.



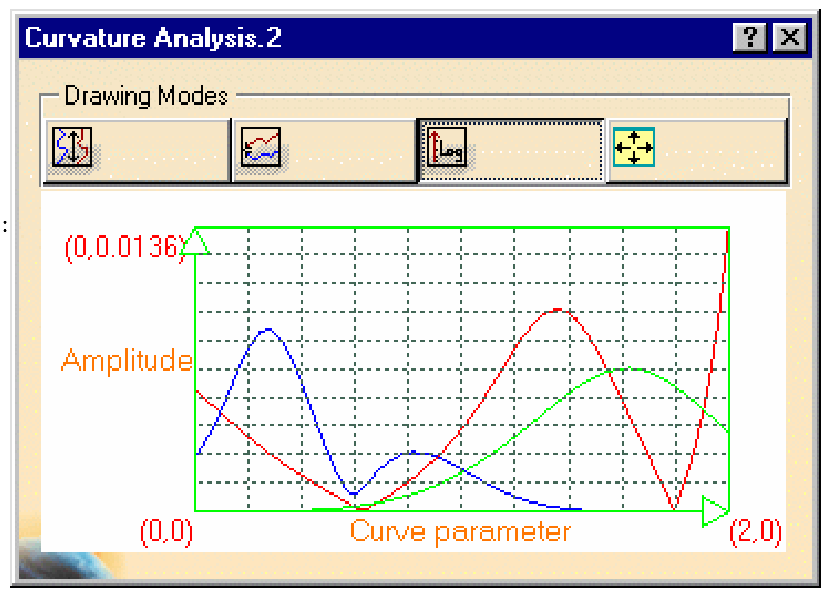




- **Same vertical length** : all curves are displayed according to the same vertical length, regardless of the scale



- **Same origin** : all curves are displayed according to a common origin point on the **Amplitude** scale



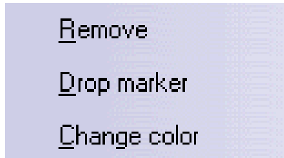
- **Vertical logarithm scale** : all curves are displayed according to a logarithm scale for the **Amplitude**, and a linear scale for the **Curve parameter**.

Depending on the chosen option, values displayed in the diagram are updated.

The last icon  is used to reframe the diagram within the window, as you may move and zoom it within the window.

14. Right-click a curve and choose one of the following options from the contextual menu:

- **Remove**: removes the curve
- **Drop marker**: adds Points.xxx in the specification tree
- **Change color**: displays the Color selector dialog box that enables you to change the color of the curve.



15. Slide the pointer over the diagram to display the amplitude at a given point of the curve.
    You can slide the pointer over the diagram and the 3D analysis.
    Click the x in the top right corner to close the diagram.
16. Click OK in the Curvature Analysis dialog box once you are satisfied with the performed analysis.

The analysis (identified as Curvature Analysis.x) is added to the specification tree.

In case of clipping, you may want to temporarily modify the Depth Effects' **Far** and **Near Limits**. See Setting Depth Effects in *CATIA Infrastructure User Guide*.

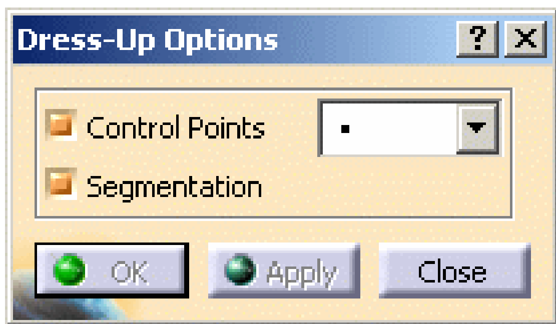
Applying a Dress-Up

This task shows you how to display or hide permanent control points and curve/surface segments on elements for analyses purposes.

Open the DressUp1.CATPart document.

1. Select **Apply Dress-Up** from the **Insert -> Analysis** menu bar.

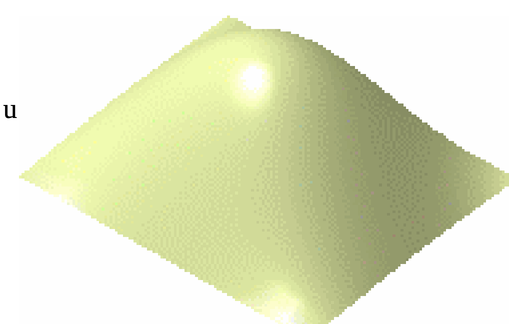
The Dress-Up Options dialog box is displayed.



2. Set the type of visualization you want to apply to geometric elements.
   You can choose:

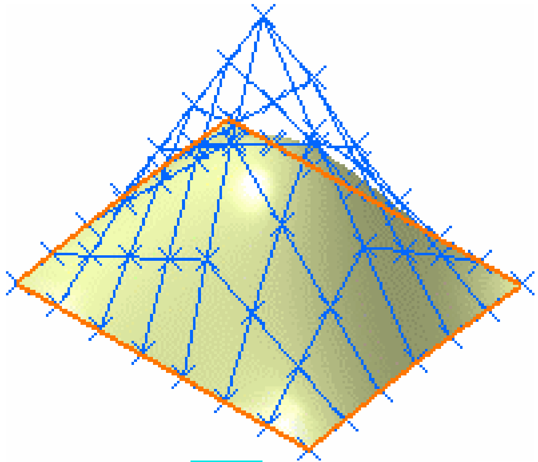
- to display, or not, control points
- the control points type
- to display, or not, segments.

3. Select the element on which you wish to display the control points.

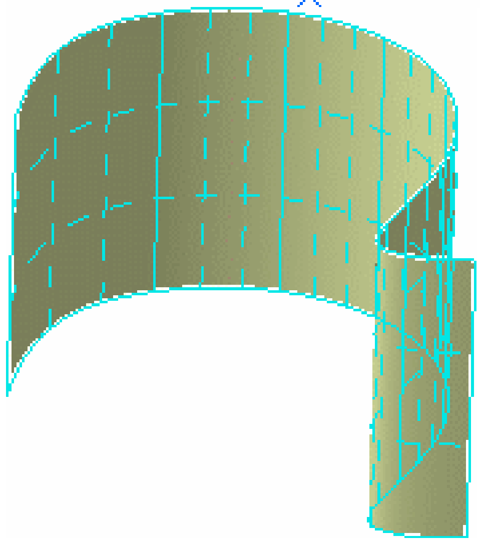


4. Click **Apply**.

The control points and mesh lines are displayed on the selected element.

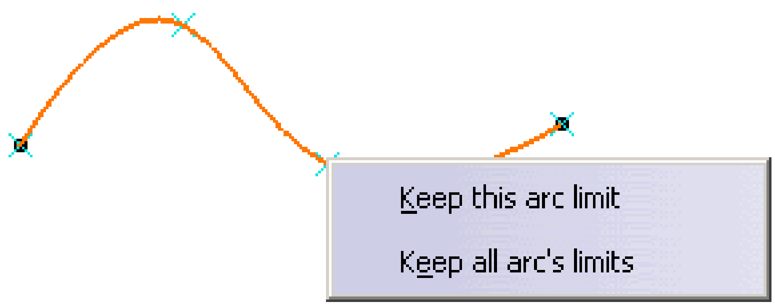


5. Activate the **Segmentation** option, uncheck the **Control Points**, then click **Apply**.

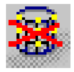


A contextual menu is available when selecting an arc limit of a curve. It enables you to either keep the arc limit you right-clicked or all the arc's limits.

- **Keep this arc limit**: a 3D point appears in datum mode in the specification tree
- **Keep all arc's limits**: all the 3D points used to create the 3D curve appear in datum mode in the specification tre

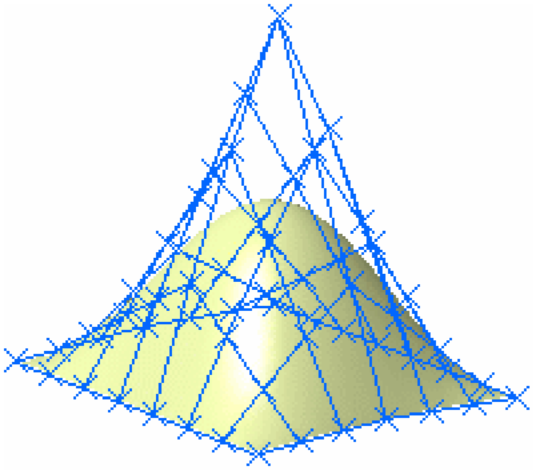


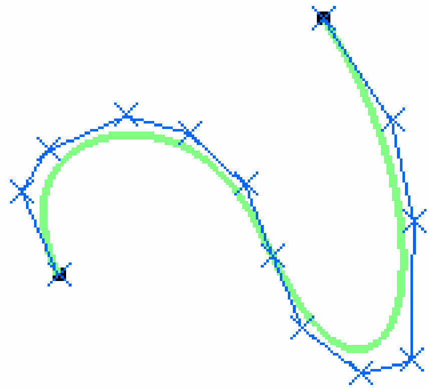
6. Click OK.

The visualization options are as defined by the user and remain on the selected elements till you select the **Remove Visualization Options** icon  from the **Insert -> Analysis** menu bar, or till you modify them using the Dress-Up Options dialog box again.

The visualization options are applied globally to the document, meaning that you can apply different options to several elements, but if you save, close then open the document again, the **options defined last** will be applied to all the elements on which visualization options have been set.

- Multi-selection applies with these display capabilities:





Displaying Geometric Information On Elements

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

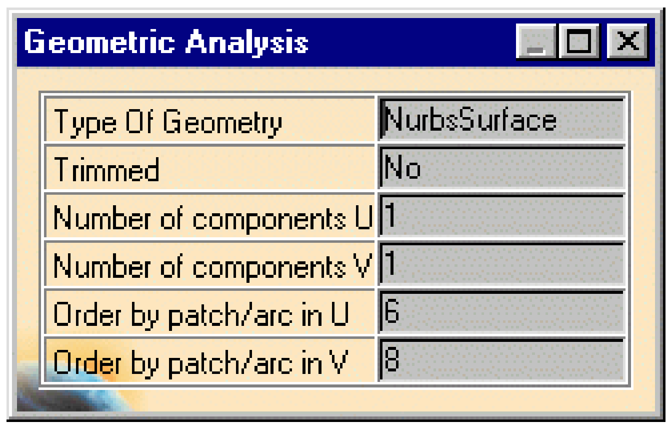
This task shows you how to display or hide geometric information on geometrical elements, such as any curve, or surface, either as a stand-alone element, or taking part in the composition of another element (intersection curve, cylinder axis, face of a pad, and so forth)

Open the GeometricInformation1.CATPart document, or any .CATPart document containing geometrical elements.

1. Click the **Geometric Information** icon .

2. Select the element for which you want to display information either in the geometric area or in the specification tree.

The Geometric Analysis dialog box is displayed.

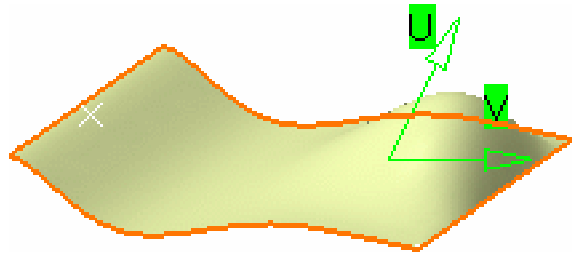


Information, such as:

- the element type (Nurbs surface or curve, Pline, planes, etc.)
- whether the element has been trimmed, or not
- the number of segments (components) in both U & V direction (where applicable)
- the degree of the element in both U & V direction (where applicable)

is displayed in the dialog box.

Moreover, a vector representing the element's orientation (U for a curve, and U & V for a surface) is displayed on the geometrical element itself.



- Uncheck the **Geometric Information** icon to exit the command, or simply click another icon.
- You cannot select an element from the specification tree as the selected element might be too complex (i.e. be composed of more than one cell) and the system cannot determine which element is to be analyzed.
- The geometry type is categorized as follows:

Displayed Type	What is it ?
NupbsCurve	Non Rational NURBS Curve
NupbsSurface	Non Rational NURBS Surface
NurbsCurve	Rational NURBS Curve
NurbsSurface	Rational NURBS Surface
PNupbs	Parametric non rational curve on a surface
PNurbs	Rational parametric curve
PSpline	Parametric curve on a surface
PLine	Isoparametric curve on a surface
Line	Line or line segment
Plane	Plane or planar face
Cylinder	Cylinder
Helix	Helix
FilletSurface	Procedural Fillet surface
SweepSurface	Procedural Sweep surface
Tabulated Cylinder	Procedural Extrude surface.
IntCurve	Intersection curve, that is resulting from the intersection of two surfaces
MergedCurve	The aggregate of two curves with different limits or parameterizations.

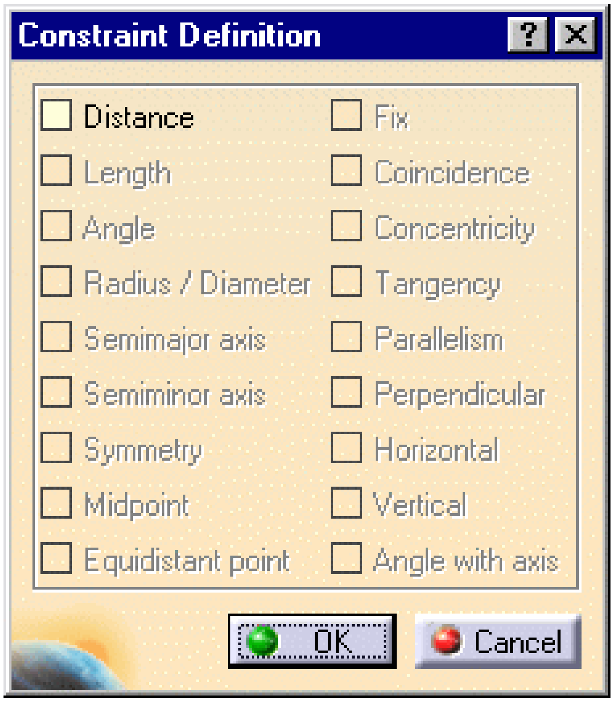
Creating Constraints

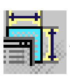
This command is only available with the Generative Shape Design 2 product.

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:



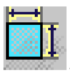
1. Multi-select two or three elements to be constrained.
2. Click the **Constraint defined in dialog box** icon .

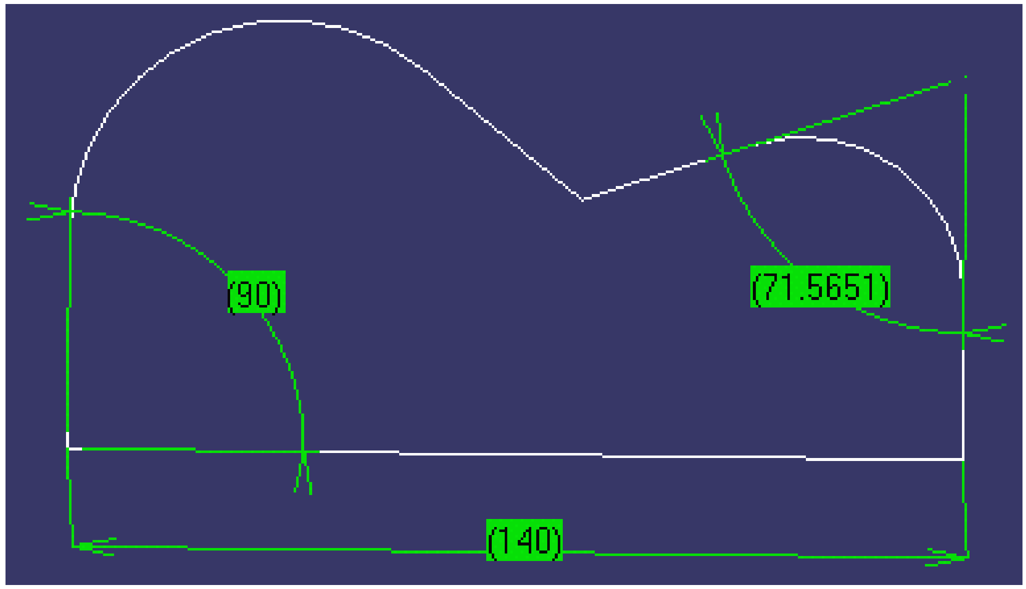
The Constraint Definition dialog box appears indicating the types of constraint you can set between the selected elements.

3. Select one of the available options to specify that the corresponding constraint is to 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.

Creating a Text with Leader

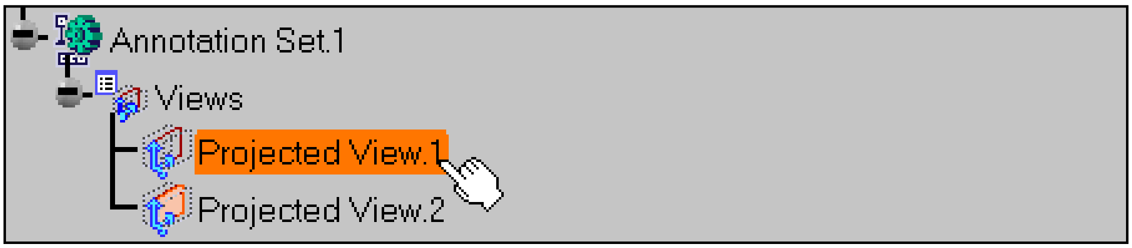
This task shows you how to create an annotation text with leader

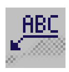
A text is assigned an unlimited width text frame.
You can set graphic properties (anchor point, text size and justification) either before or after you create the free text.
You can change any text to another kind at any time.

Open the Common_Tolerancing_Annotations_01 CATPart document.

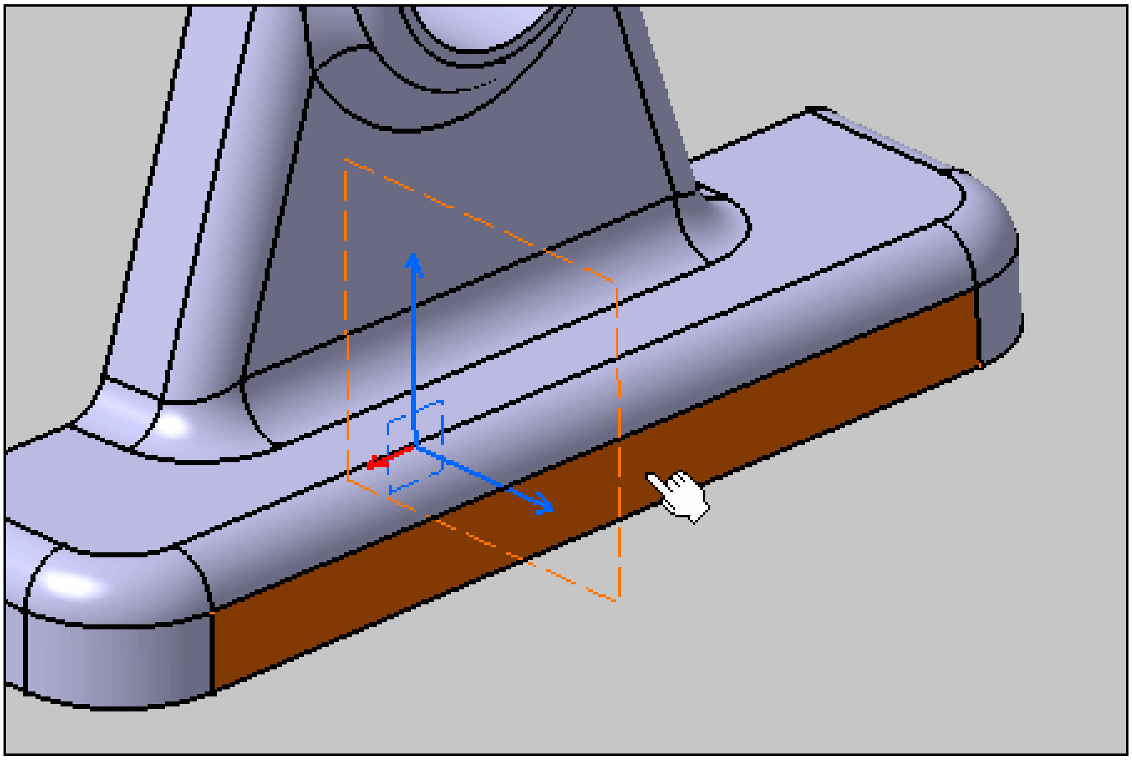
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

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



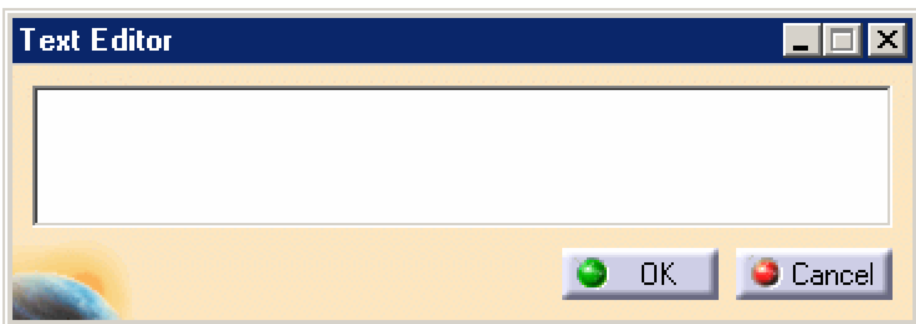
2. Click the **Text with Leader** icon: 

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

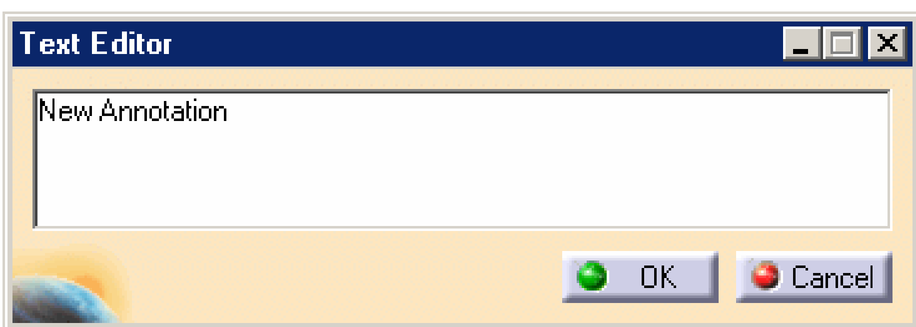


If the active view is not valid, a message appears informing you that you cannot use the active view.
Therefore, the application is going to display the annotation in an annotation plane normal to the selected face.
For more information, see View/Annotation Planes.

The **Text Editor** dialog box appears.

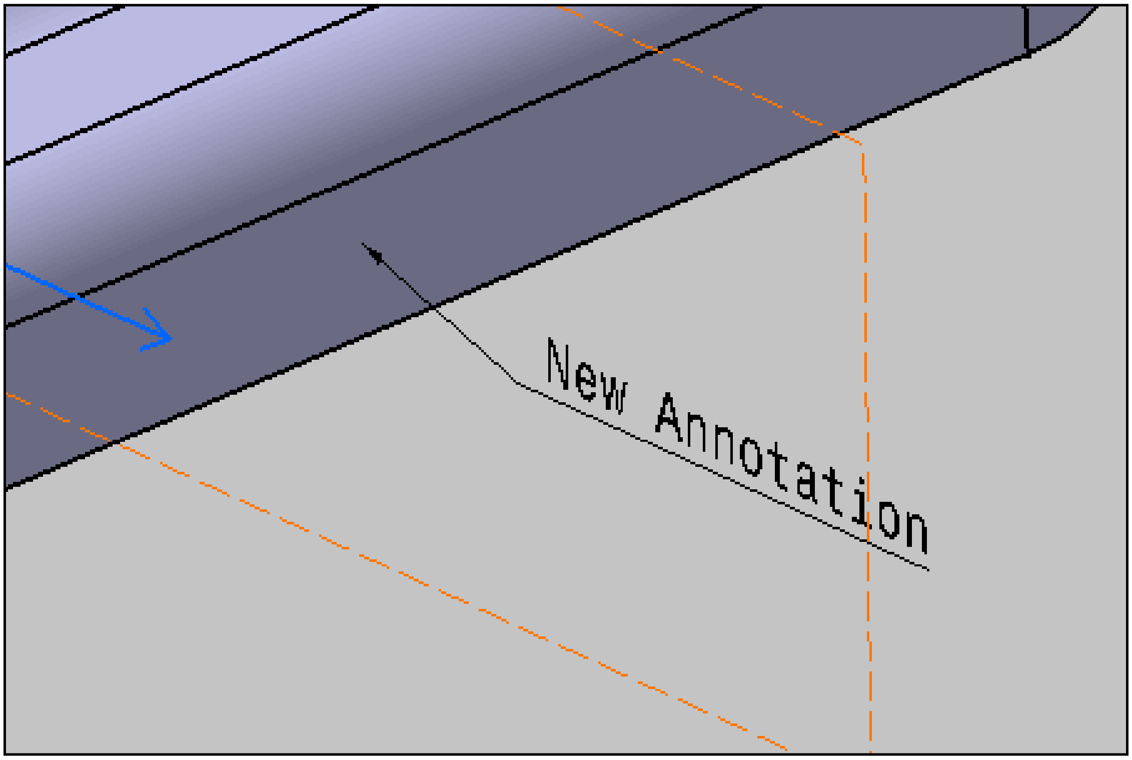


4. Enter your text, for example "New Annotation" in the dialog box.



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.

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

Creating a Flag Note with Leader

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

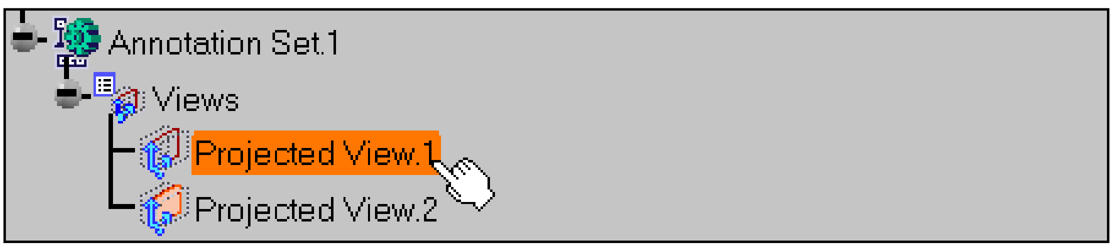
A flag note allows you to add links to your document and then use them to jump to a variety of locations, for example to a marketing presentation, a text document or a HTML page on the intranet. You can add links to models, products and parts as well as to any constituent elements.

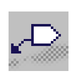
A flag note is assigned an unlimited width text frame.
You can set graphic properties (anchor point, text size and justification) either before or after you create the free text.
You can change any flag note to another kind at any time.

Open the Common_Tolerancing_Annotations_01 CATPart document.

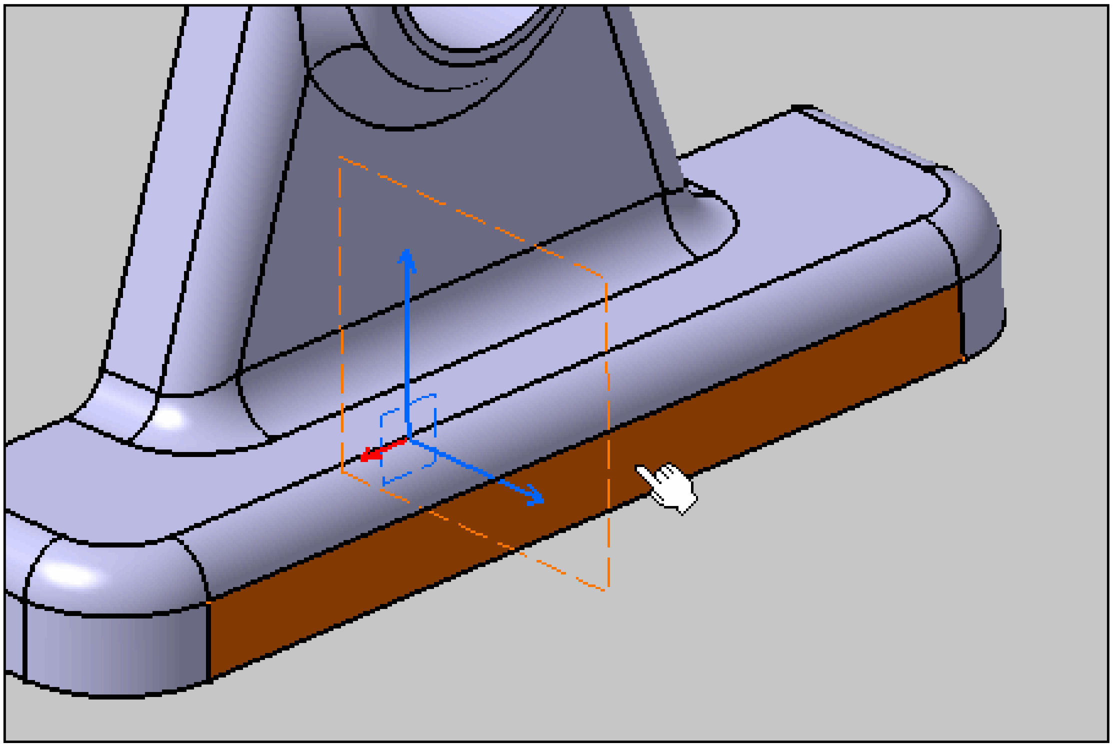
- Improve the highlight of the related geometry, see Highlighting of the Related Geometry for 3D Annotation.

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

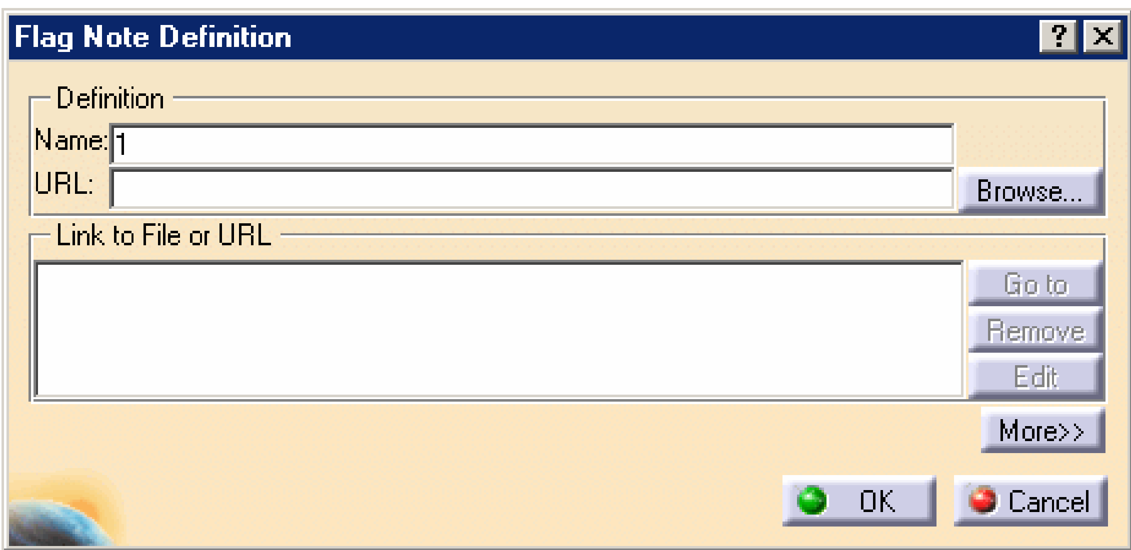


2. Click the **Flag Note with Leader** icon: 

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



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 **Manage Hyperlink** 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.

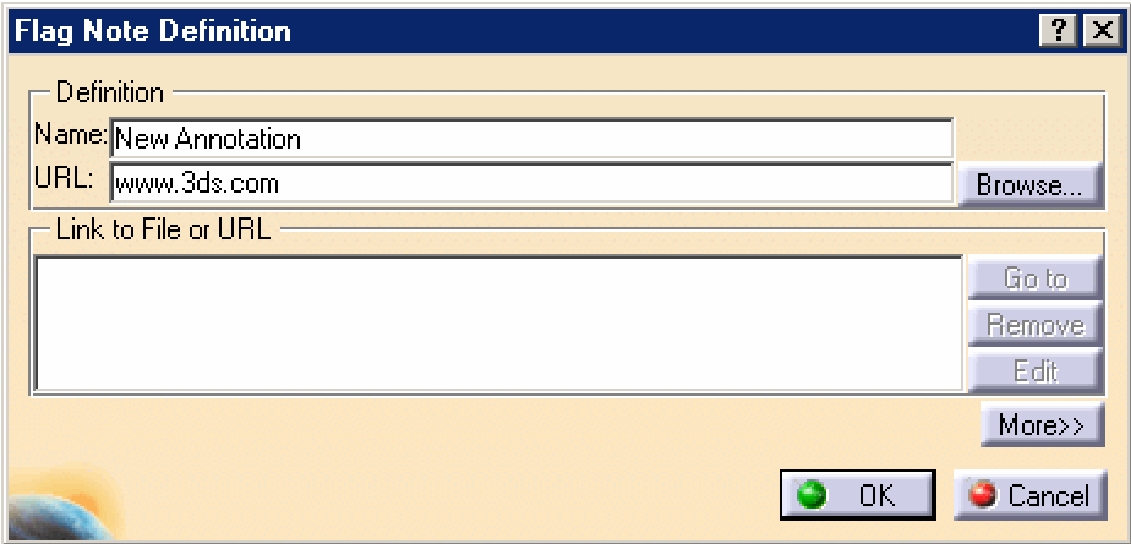
In the Link to File or URL list you can see the list of links.

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.

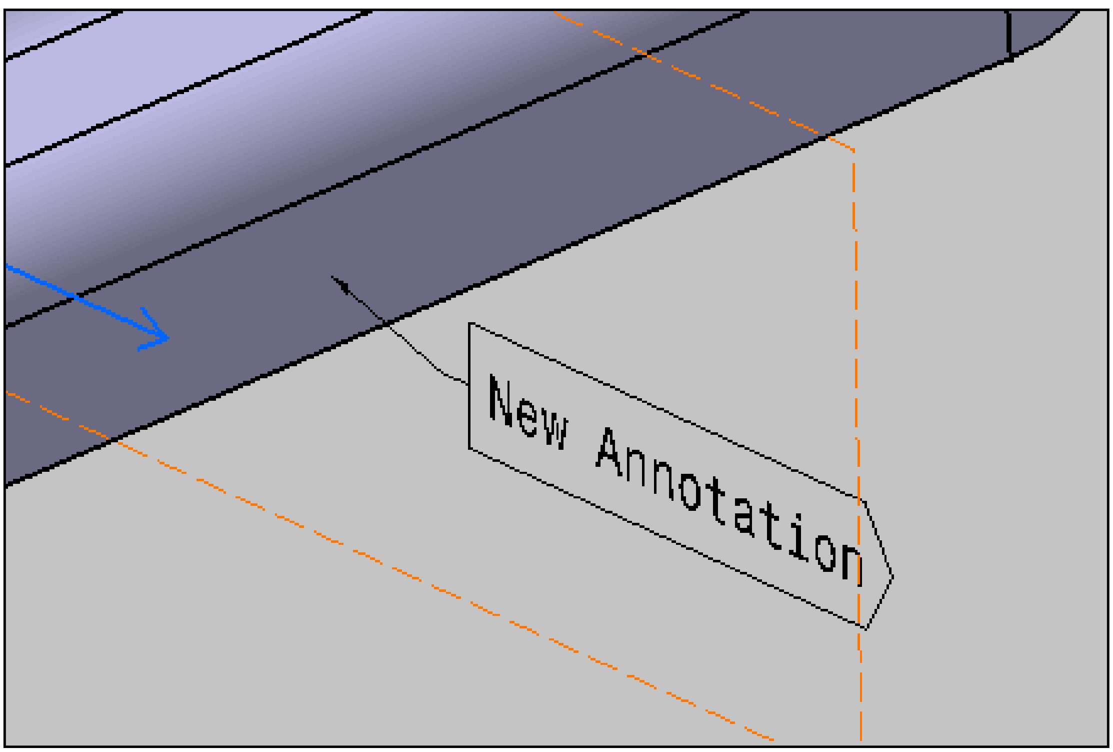
To edit one of them, select it and click the **Edit** button.

4. Enter your flag note name, for example "New Annotation" in the dialog box and specify a link: www.3ds.com



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 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.

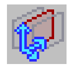
The flag notes (identified as Flag Note.xxx and its name between brackets) are added to the specification tree in the **Notes** group.

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

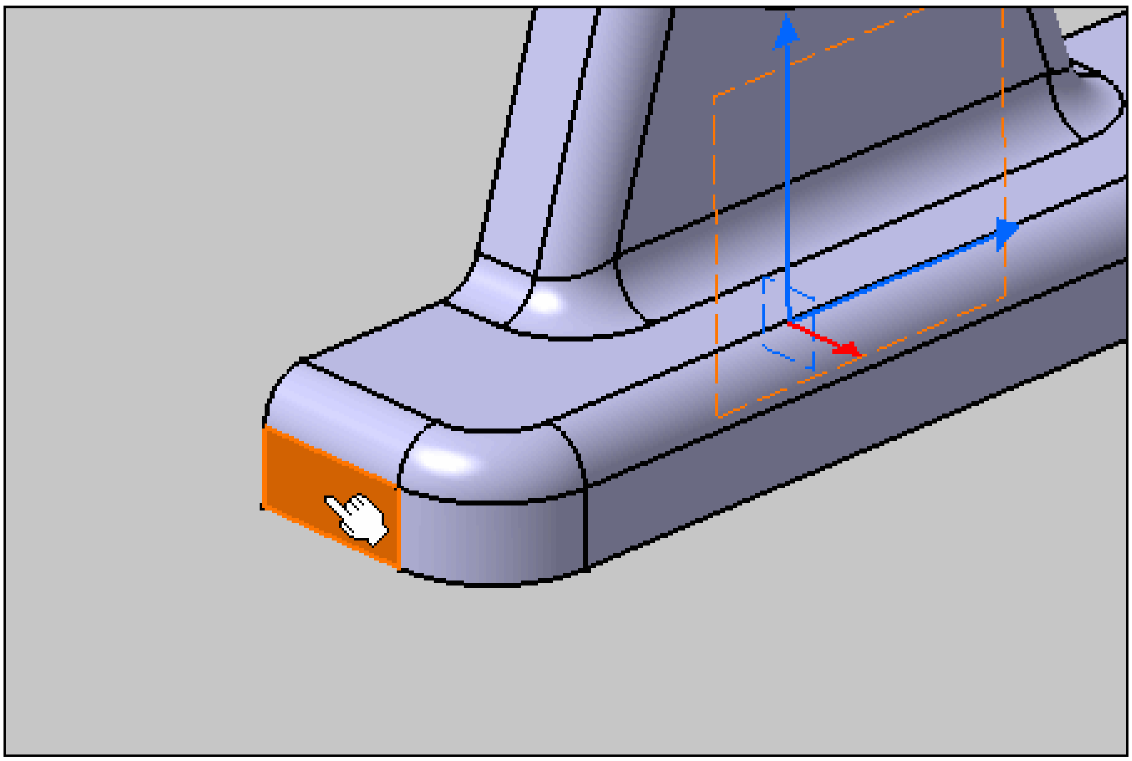
Creating a Projection View/Annotation Plane

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

Open the Common_Tolerancing_Annotations_01 CATPart document.

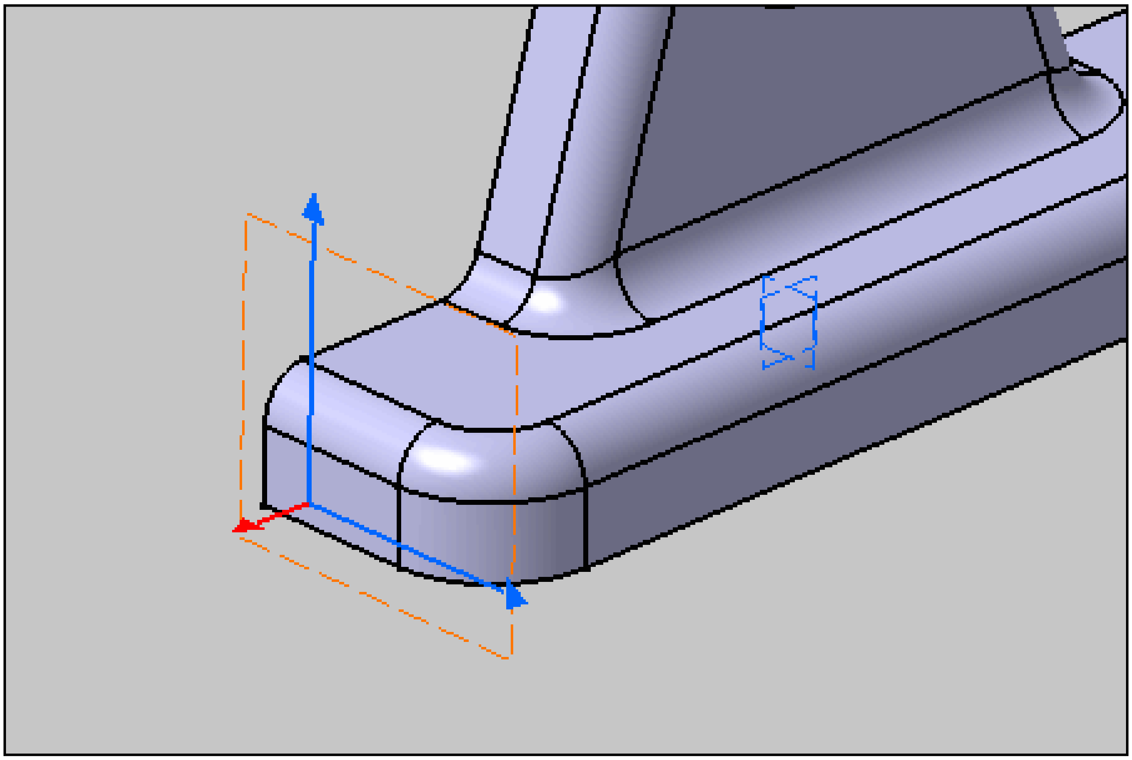
1. Click the **Projection View** icon: 

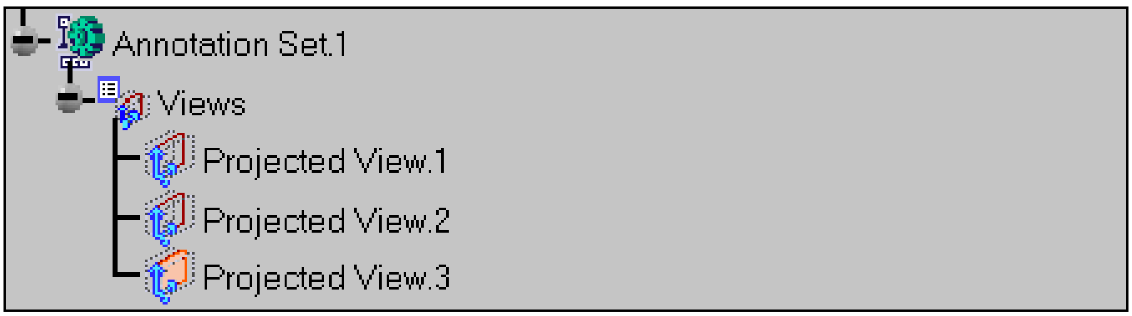
2. Select the face as shown.



You have to select a planar element only to perform this command.

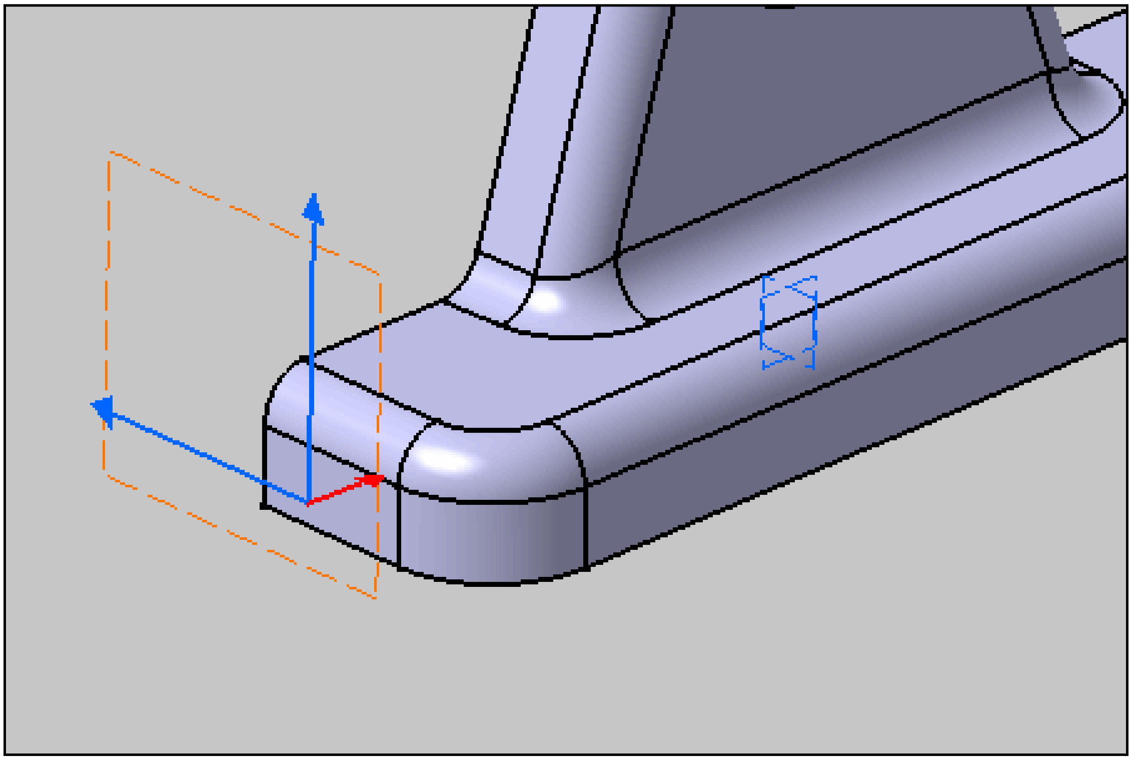
The projection view is created.
Projection views are represented by a blue reference axis, its normal axis is red until you create an annotation, and are identified as **Projection View.3** in the specification tree.





3. Right-click the annotation plane in the geometry or in the specification tree and select the **Invert Normal** contextual menu.

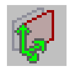
The projection view normal is reversed.



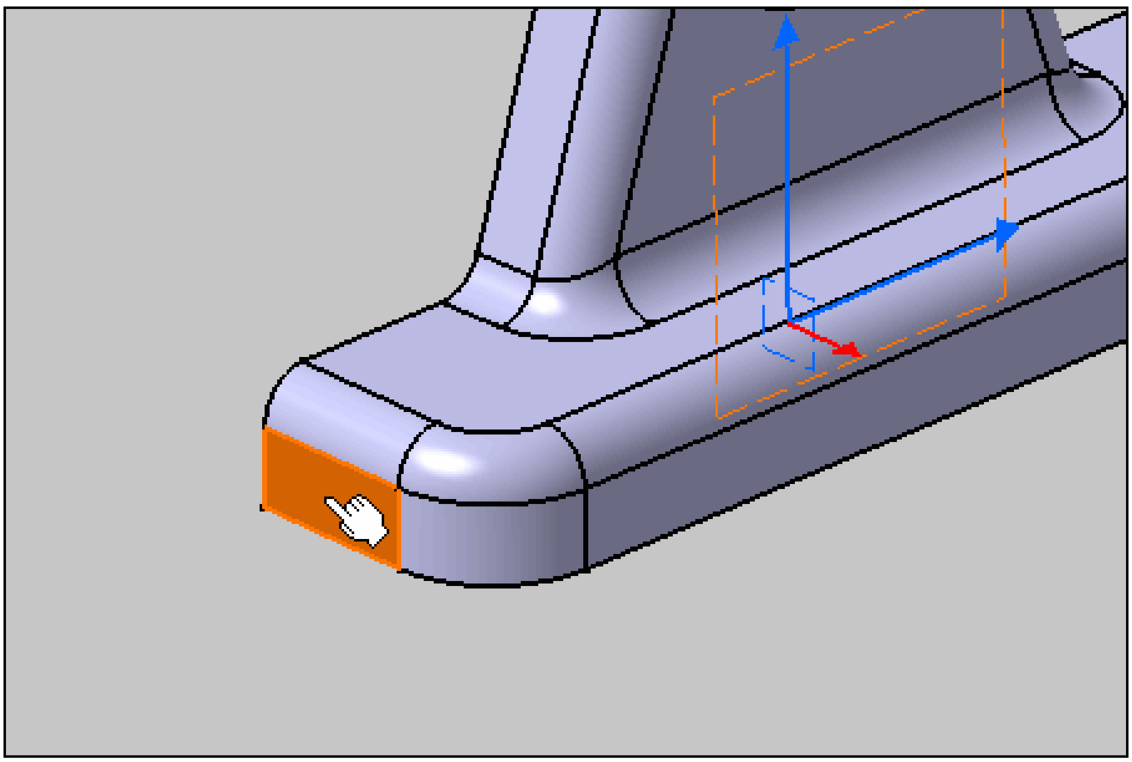
Creating a Section View/Annotation Plane

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

Open the Common_Tolerancing_Annotations_01 CATPart document.

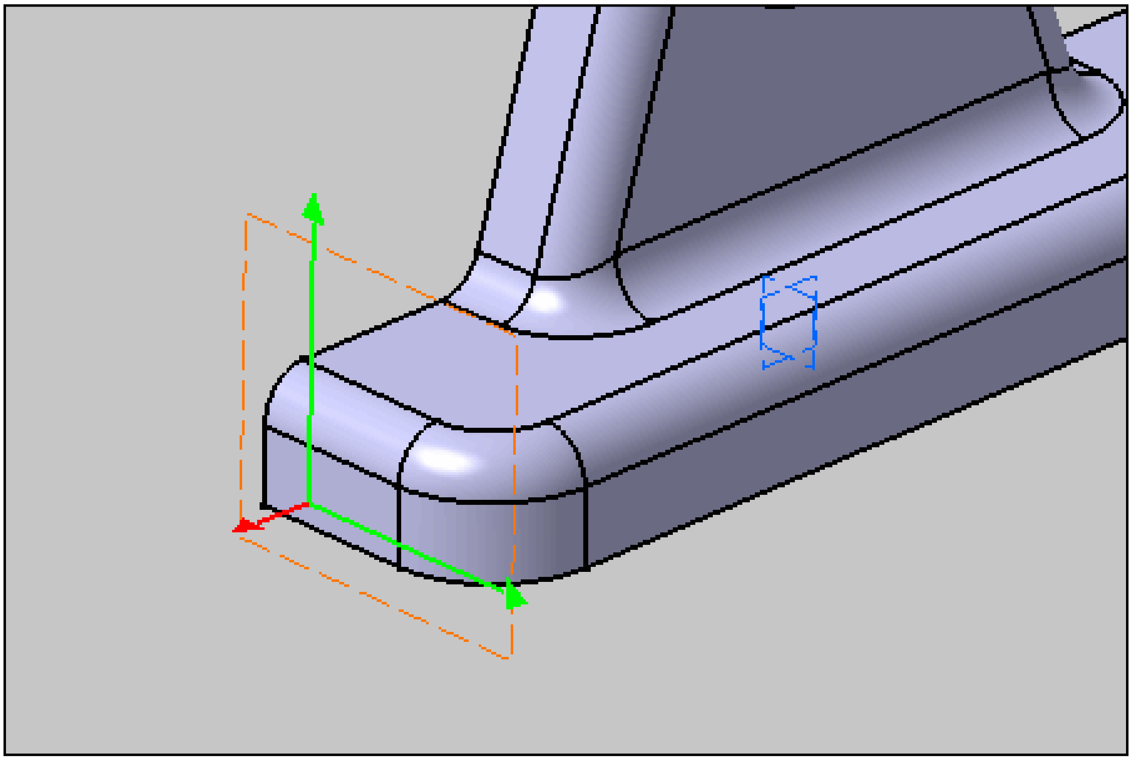
1. Click the **Section View** icon: 

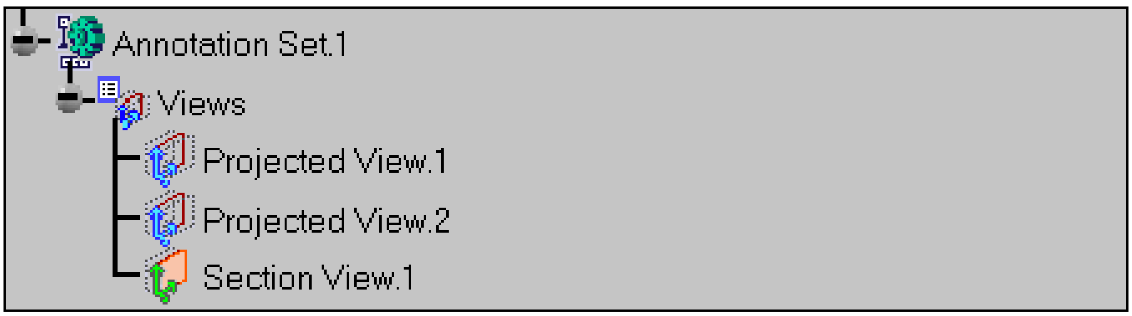
2. Select the face as shown.



You have to select a planar element only to perform this command.

The section view is created.
Section views are represented by a green reference axis, its normal axis is red until you create an annotation, and are identified as **Section View.1** in the specification tree.

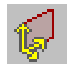



Annotation Set.1
Views
Projected View.1
Projected View.2
Section View.1

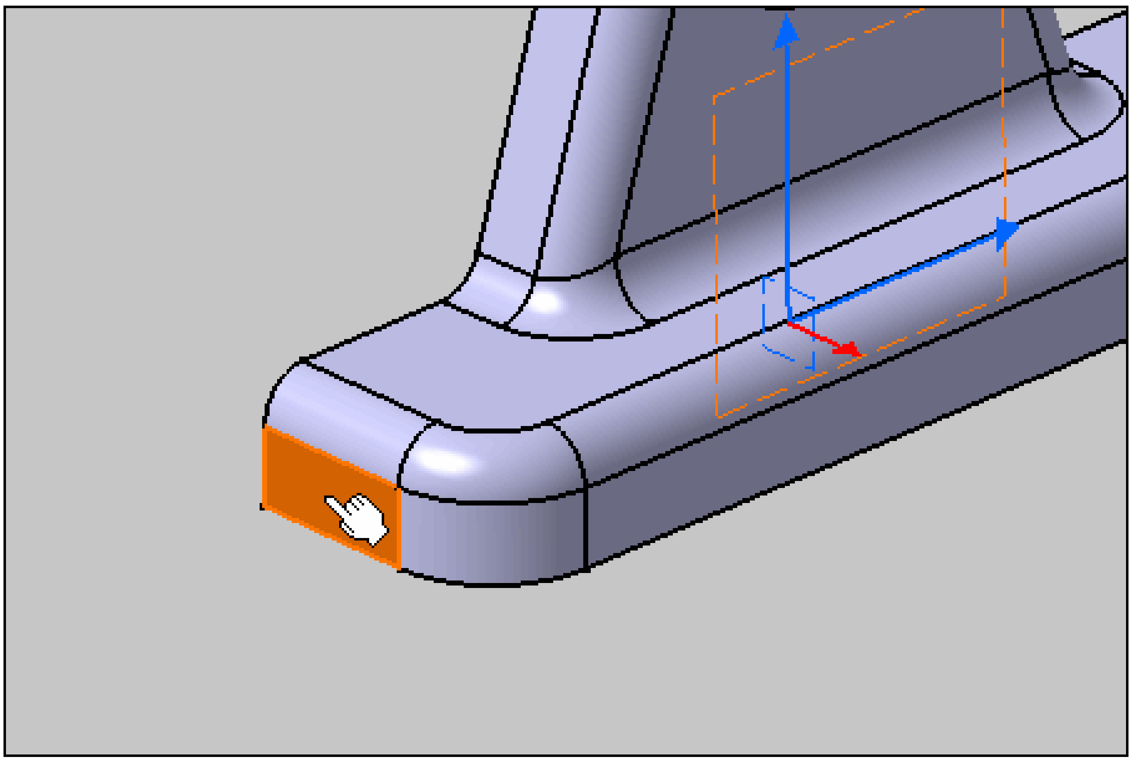
Creating a Section Cut View/Annotation Plane

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

Open the Common_Tolerancing_Annotations_01 CATPart document.

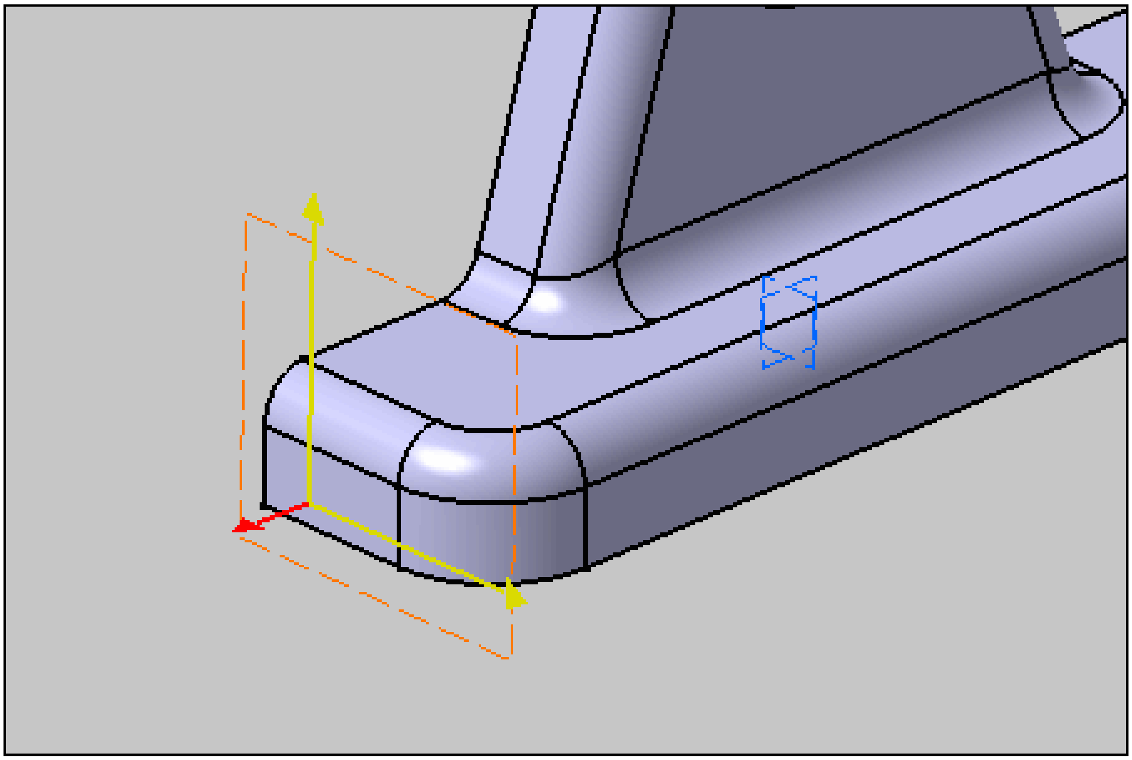
1. Click the **Section Cut View** icon: 

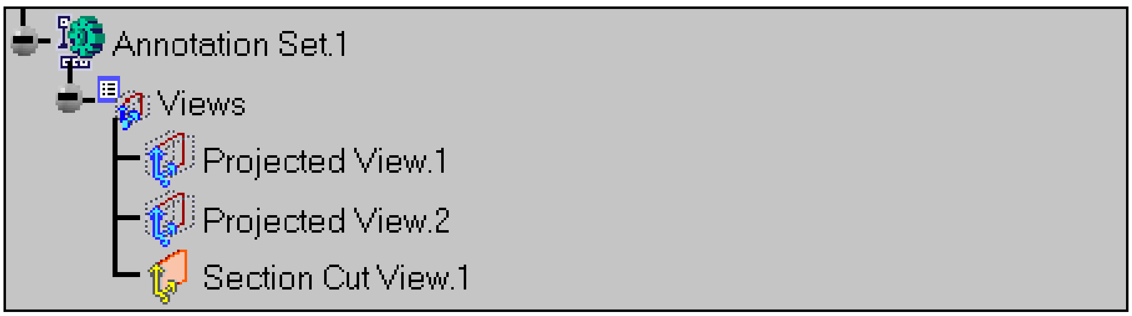
2. Select the face as shown.



You have to select a planar element only to perform this command.

The section cut view is created.
Section views are represented by a yellow reference axis, its normal axis is red until you create an annotation, and are identified as **Section Cut View.1** in the specification tree.




Annotation Set.1
Views
Projected View.1
Projected View.2
Section Cut View.1

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 Geometrical Set (in a .CATPart document). 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).

Materials applied to .CATPart, .CATProduct and .cgr documents can be saved in ENOVIAVPM.
For detailed information on ENOVIAVPM, refer to the *ENOVIAVPM User's Guide.*

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.

Open the ApplyMaterial.CATProduct document.

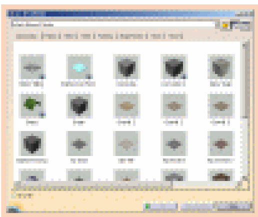
To visualize the applied material, select the **Shading with Material** icon from the View Toolbar.

1. Select the element(s) on which the material should be applied.

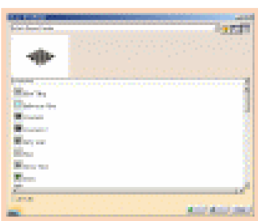
If you want to apply a material simultaneously to several elements, simply select the desired elements (using either the pointer or the traps) before applying the material.

2. Click the **Apply Material** icon .

The Library dialog box opens. It contains several pages of sample materials from which to choose.
From V5R14 onwards, two new families are provided with the default material library: Painting and Shape Review. In addition to these two new families, more materials have been added to the already existing families. For instance, when working with the Wood family you can now choose the Mahogany type, the Cedar type, the Kingwood type, etc.
For a complete description of the families provided with the default material library, refer to "Material Sample Library" in this guide.
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...



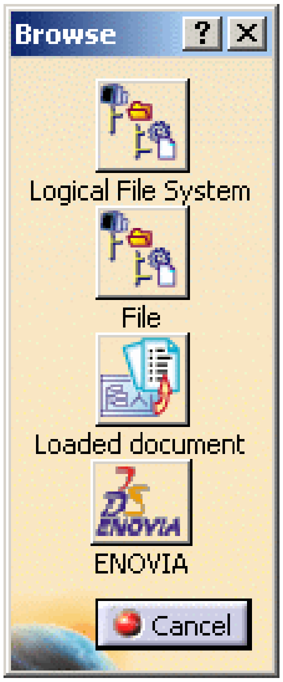
...or each page is identified by a material family name in a pulldown list if you select the Display list mode:



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.

Depending on the document environments (i.e. the method to be used to access your documents) you allowed in the Tools->Options->General->Document tab, an additional window such as the one displayed below may appear simultaneously to the File Selection dialog box to let you access your documents using an alternate method:



In our example, four document environments have been allowed among which the DLName environment. If you want to access your texture files using DLNames, for instance, just click the **Logical File System** button: this will open a specific dialog box dedicated to the DLName environment.
For detailed information on this dialog box, refer to Opening Existing Documents Using the Browse Window.

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.

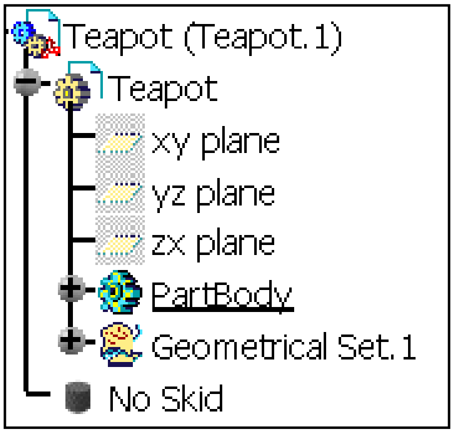
Note: You can edit linked materials. Doing so will modify the original material in the library. If you want to save changes made to the original material, use the **File->Save All** command.

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.

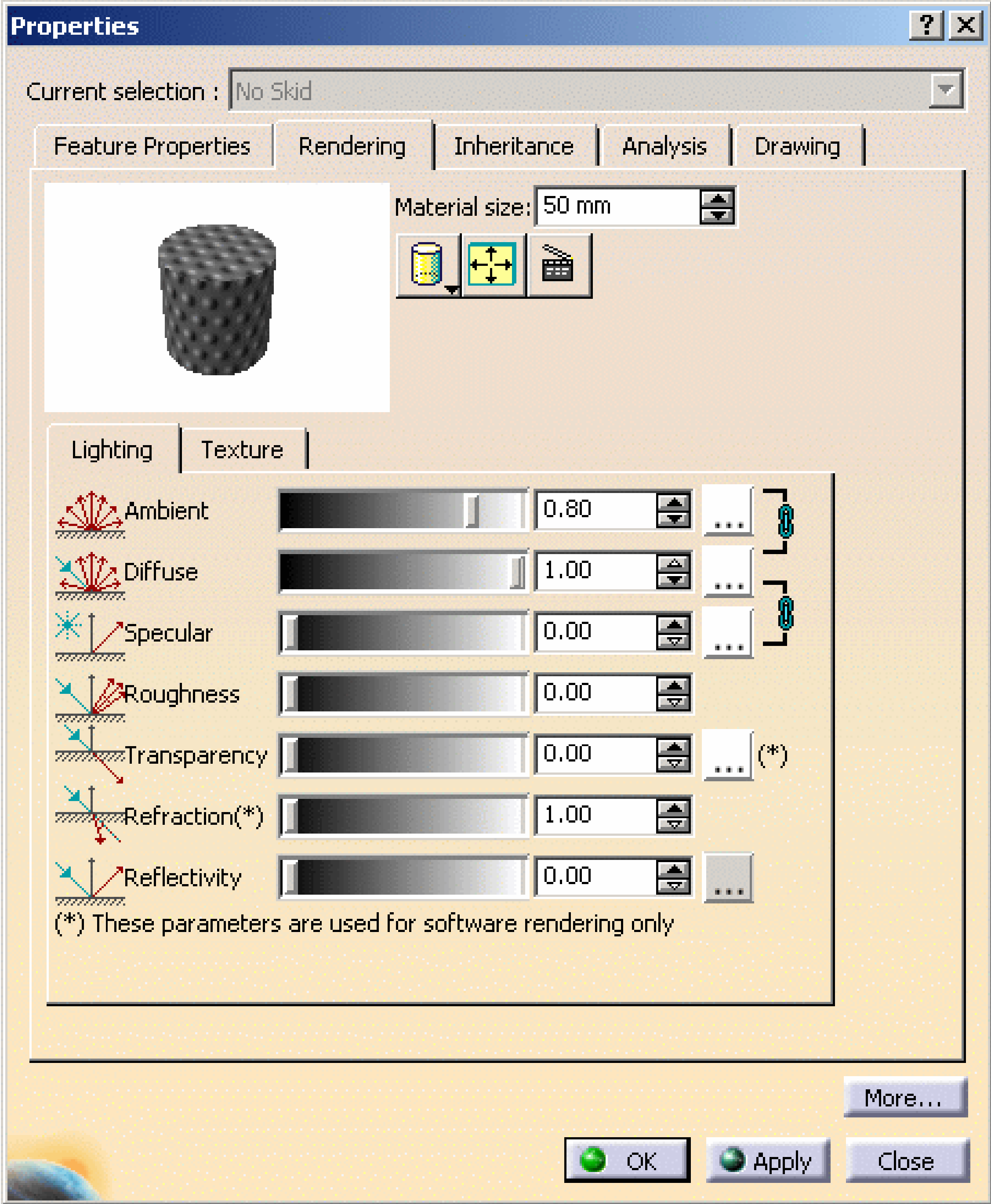
6. Click **OK** in the Library dialog box.

The object looks the following way:

Note: applying materials to elements affects 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.

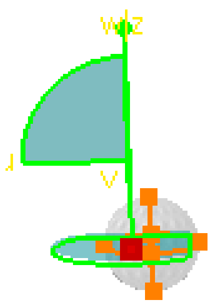
Note: Appropriate licenses are required to use the Analysis and Drafting tabs.

If you are working in "Materials" visualization mode (i.e. the "Materials" option is checked in the Custom View Modes dialog box) 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.

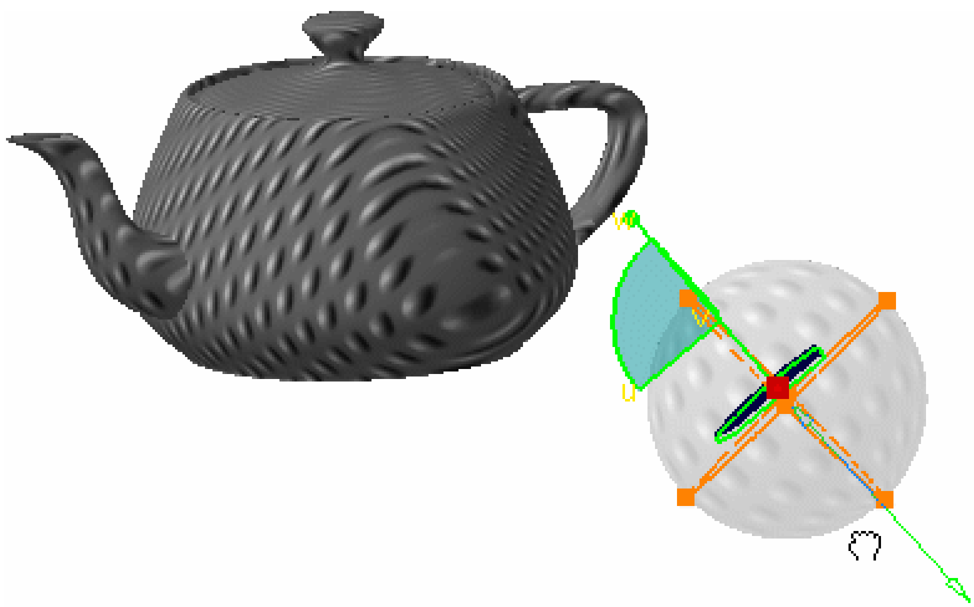
- Select the material in the specification tree:



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):

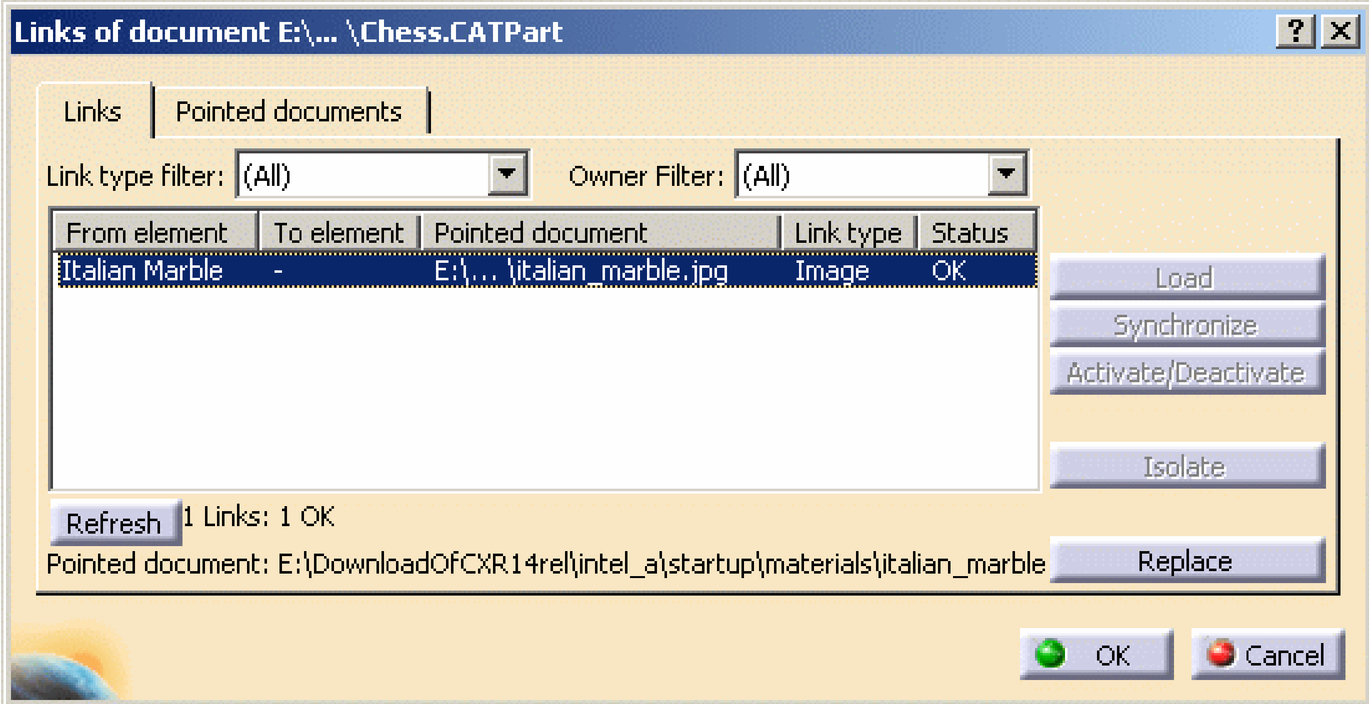
- Use the mapping support handles to stretch the material texture along u- and v- axes (as you can do it with the slider in the *Scale U, V* fields displayed in the Texture tab):



For more information on manipulating objects using the 3D compass, refer to the *Version 5 Infrastructure User's Guide*.

More about materials

Contrary to materials with no texture (such as "Gold"), materials with a texture (such as "Teak") are applied with an external link to their texture image. Therefore, this link is displayed when using the **File->Desk**, **Edit->Links...** or **File->Send To** command.
In the example below, "Italian Marble" has been applied onto Chess.CATPart and the link to the corresponding .jpg image appears when displaying the Links dialog box:


Links of document E:\... \Chess.CATPart
Links
Pointed documents
Link type filter:
(All)
Owner Filter:
(All)
From element
To element
Pointed document
Link type
Status
Italian Marble
-
E:\... \italian_marble.jpg
Image
OK
Load
Synchronize
Activate/Deactivate
Isolate
Refresh
1 Links: 1 OK
Pointed document: E:\DownloadOfCXR14rel\intel_a\startup\materials\italian_marble
Replace
OK
Cancel

Applying a Thickness

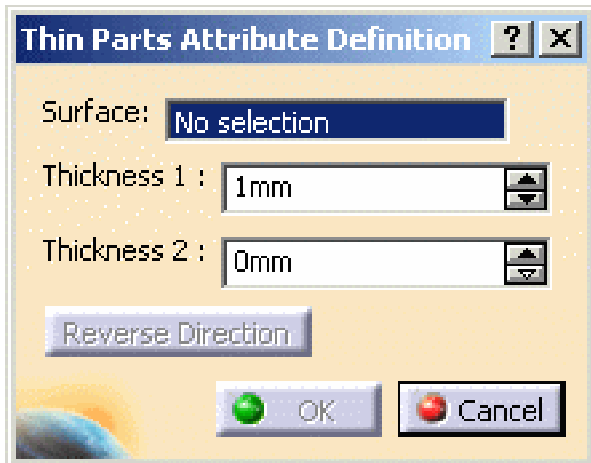
This command is only available with the Generative Shape Design 2 product.

This task shows how to apply a thickness onto a surface.

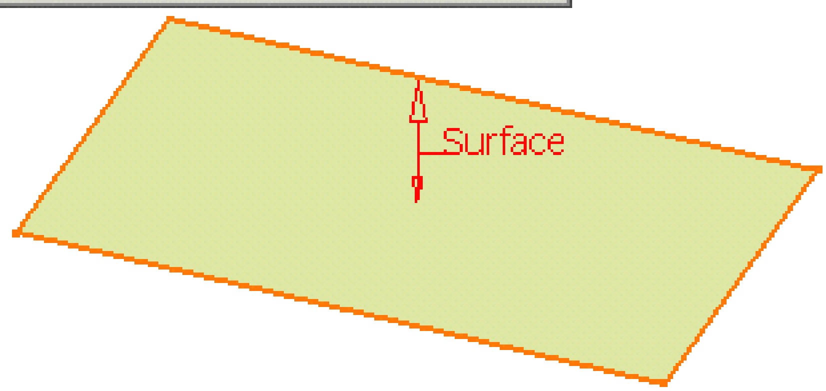
Open the Thickness1.CATPart document.

1. Select **Tools -> Thin Parts Attribute**.

The Thin Parts Attribute definition dialog box displays.



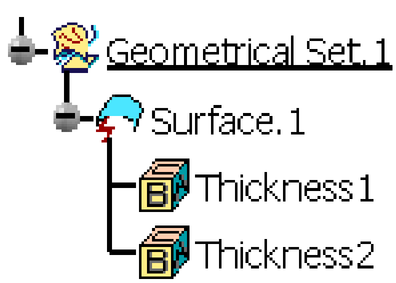
2. Select the surface.
3. Choose the input values for Thickness 1 (indicated by the arrow) and Thickness 2 using the spinners.



Check the **Reverse Direction** button to inverse the direction of thickness 1 and 2.

4. Click OK to apply the thickness.

Thicknesses 1 and 2 appear in the specification tree, as the surface's attributes.



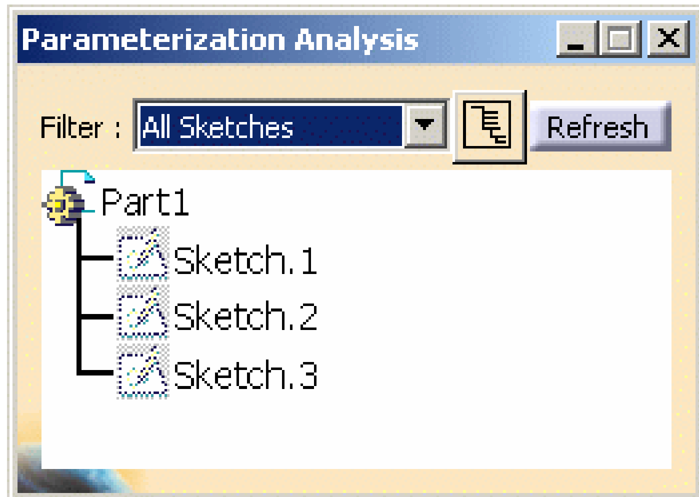
Analyzing Using Parameterization

This task enables to analyze the CATPart structure and shows how to isolate specific features within a part. This is particularly useful when managing power copies, for example.

Open the Parameterization1.CATPart document.

1. Select **Tools -> Parameterization Analysis...**

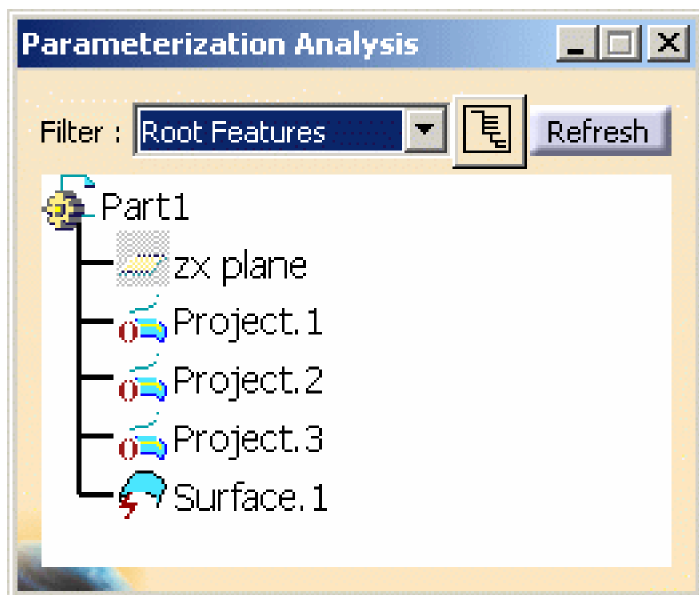
The Parameterization Analysis dialog box opens.

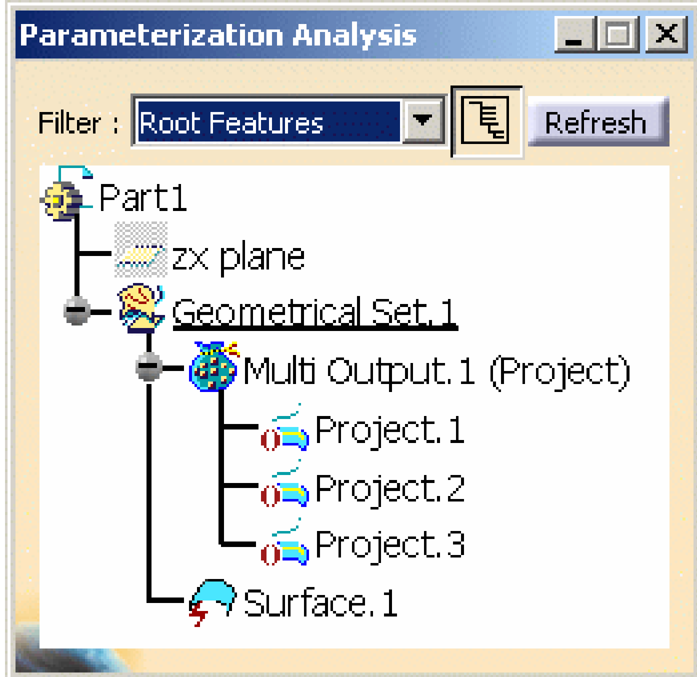


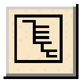
2. Use the Filter combo and choose to display Root Features.

The query is launched and the viewer automatically updates.

Deactivated features as well as datum features are displayed. However, if you want to display deactivated features only, select the Inactivated Features filter. Similarly, select the Isolated Features filter to display both deactivated and datum features.





3. Click the **Display Body Structure** icon  to display the graph keeping the tree structure.

Each feature is displayed within its own body.

- Features displayed in the viewer can be used in the same way as in the specification tree: double-click a feature to edit it, or use the contextual menu to reframe on or display its properties, for example.
- The viewer can still be open while performing other operations.
  Available filters are:
  - All Sketches
  - Over-constrained sketches
  - Fully-constrained sketches
  - Under-constrained sketches
  - Inconsistent sketches
  - External references
  - Inactivated features
  - Root features
  - Leaf features
  - Isolated features
  - Features in error
  - Waiting for update features
  - Features with stop update
  - Features with active stop update
  - Knowledge formulas, Rules, and Checks
  - Bodies

For further information about sketches, refer to the Analyzing and Resolving Over-Constrained or Inconsistent Sketches chapter in the *Sketcher* documentation.

For further information about Isolated features, refer to the Isolating Geometric Elements chapter.

For further information about Features with stop update and active stop update, refer to the Updating Your Design chapter.

For further information about knowledge formulas, refer to the *Knowledge Advisor* documentation.

For further information about Bodies, refer to the *Part Design* documentation.

Managing Groups

This command is only available with the Generative Shape Design 2 product.
It is not available with Ordered Geometrical Sets.

This task shows how to manage groups of elements in a Geometrical Set entity as follows:

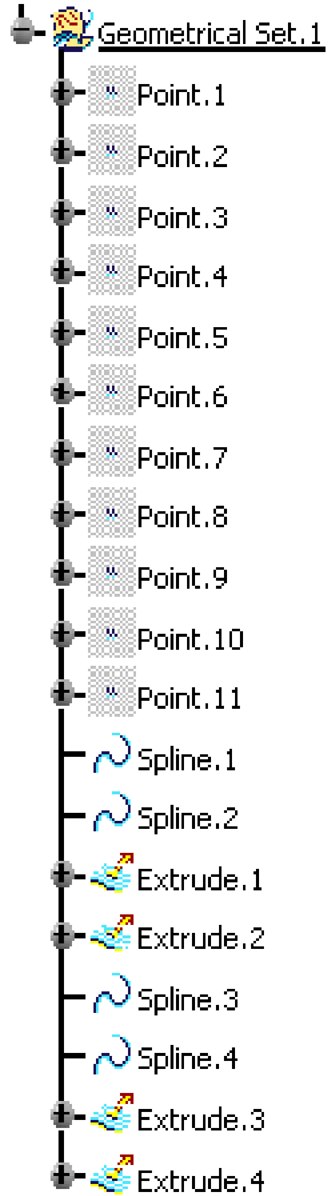
- creating a group
- editing a group
- collapsing and expanding a group
- moving a group to a new body

A group is a visualization element that applies on a geometrical set entity. Thus a group cannot exist without a geometrical set.
A group enables to reorganize the specification tree when it becomes too complex or too long and deals with the structure of the part being created.

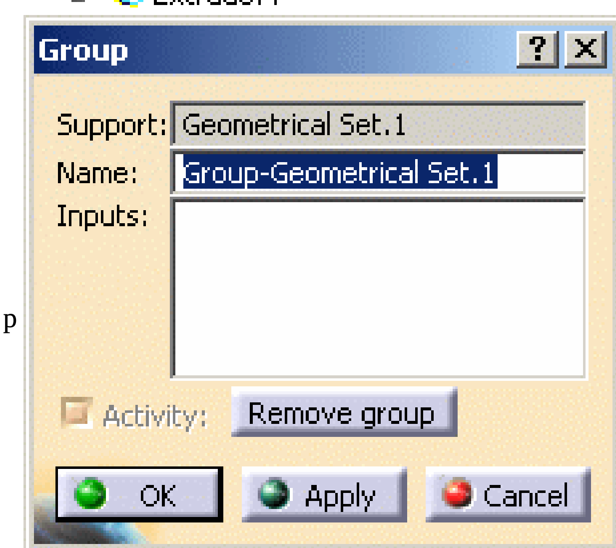
Open the Groups1.CATPart document.

Creating a group

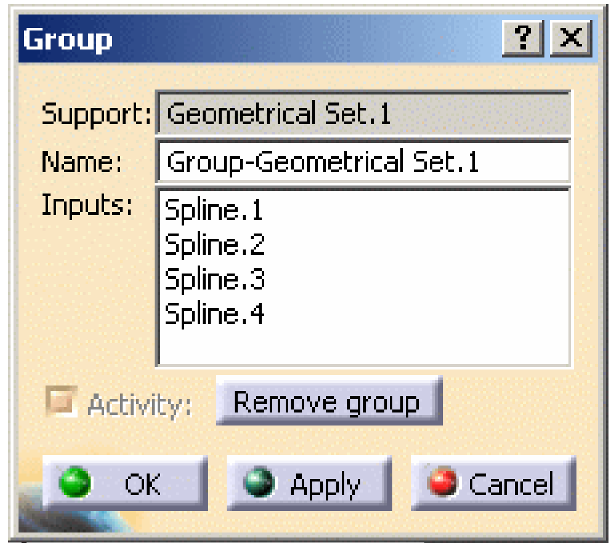
1. Right-click the desired Geometrical Set entity in the specification tree.
2. Choose the **Geometrical Set.x object -> Create Group** command from the contextual menu.



The Group dialog box appears.
The Support area indicates the name of the Geometrical Set entity where the group is to be created.

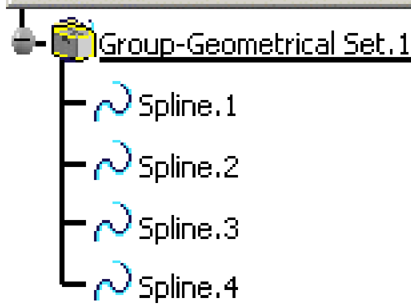


3. If needed, modify the proposed default group name that appears in the Name area.



4. Select entities to be included in the group and remain visible in the tree.

   Other entities become hidden.



5. Click **OK** to create the group.

In the Group dialog box you can:

- click the check box to specify whether group is expanded or collapsed.
- click the Remove Group button to reset the group definition.

Editing a group

1. Right-click the desired group in the specification tree and select the **Group-Geometrical Set.x object -> Edit Group...** command from the contextual menu.
2. You can then:

- rename the group
- remove the group
- add entities to the group.

Collapsing and expanding a group

1. To collapse a group, right-click the desired group in the specification tree and select the **Group-Geometrical Set.x object -> Collapse Group** command from the contextual menu.

   The portion of the specification tree related to the group appears reduced.

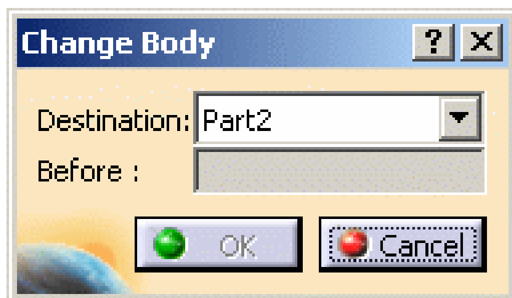
2. To expand a collapsed group, right-click the desired group in the specification tree and select the **Group-Geometrical Set.x object -> Expand Group** command from the contextual menu.

   All the entities belonging to the group are then visible in the specification tree.

Moving a group to a new body

1. Right-click the desired group in the specification tree and select the **Group-Geometrical Set.x object -> Change Geometrical Set** command from the contextual menu.

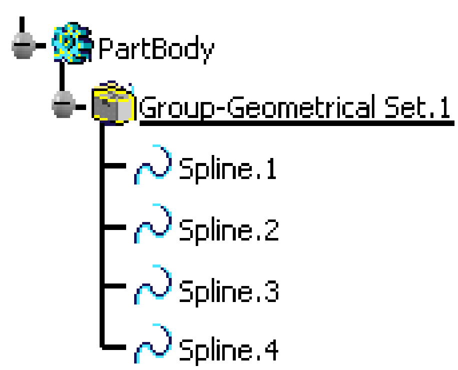
   The Change Body dialog box appears.



2. Select the new body where the group is to be located.

By default, if you select a body, the group is positioned last within the new body. However, you can select any element in the new body, before which the group will be located.

See also Moving Elements From an Geometrical Set.



3. Click OK to move the group to the new body.

You have the possibility, when creating a new feature, to integrate it or not as an input in a group.
Please refer to the *General Settings* chapter in the Customizing section.

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 (refer to the corresponding chapter in the documentation)
- 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 Body** option if you want all object instances in a separate body.
   A new Geometrical Set will be created automatically.

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

5. Click OK.
   The object is created as many times as required in the Object Repetition dialog box.

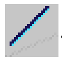
See each specific object creation for further details on what parameter is taken into account for the repetition.

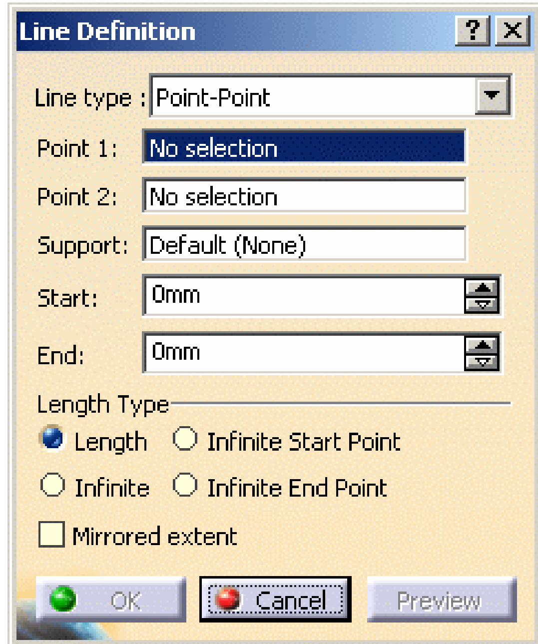
Stacking Commands

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 .

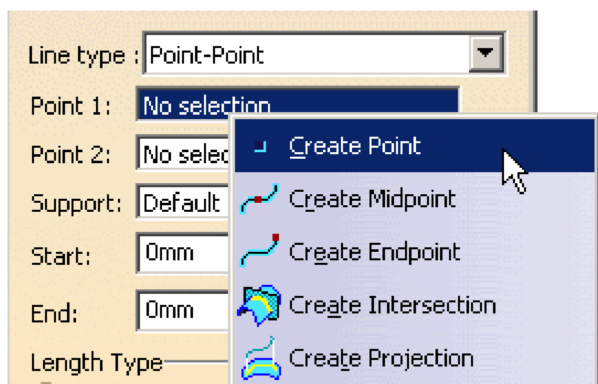


The Line Definition dialog box appears.

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

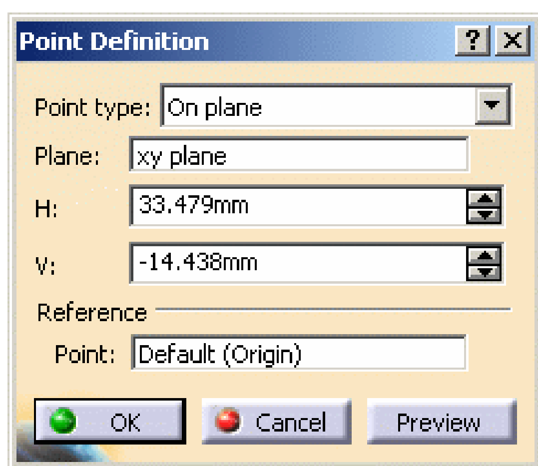
   Here we chose the **Point-Point** line type: two points are required to create the line.

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.

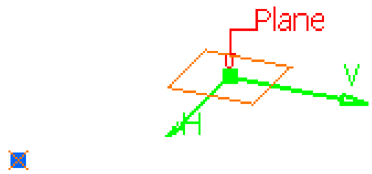
The Point Definition dialog box appears.



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

   Here we chose the **On Plane** type.

6. Choose the **Plane**.
7. Click **OK**.

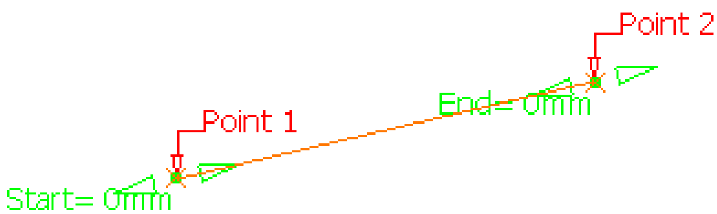


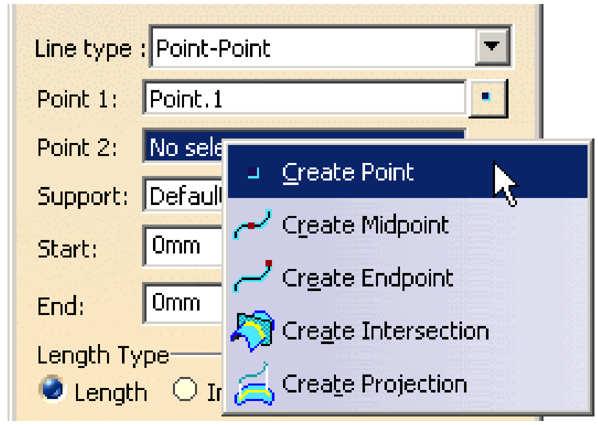
The Point Definition dialog box closes and you return to the Plane Definition dialog box.
The Point.1 field is valuated with the point you have just created.

8. Right-click the Point 2 field.

9. Repeat steps 4 to 7.

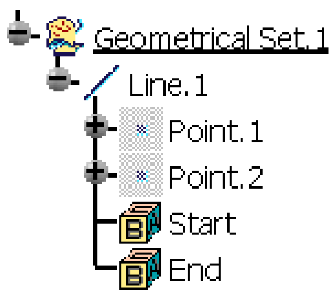
The Point Definition dialog box closes and you return to the Plane Definition dialog box.
The Point.2 field is valuated with the point you have just created and a line is previewed between Point 1 and Point 2.





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.



11. Edit the created line.

12. Right-click the Point 1 field.

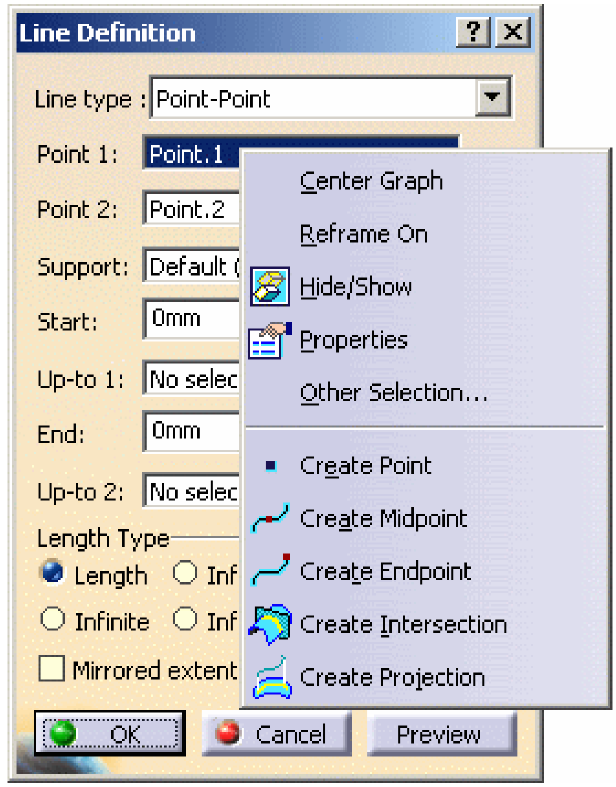
You can access generic contextual commands such as **Center Graph**, **Reframe On**, **Hide / Show**, **Properties**, and **Other Selection**.

For **Center Graph** and **Reframe On**, refer to the *Part Design User's Guide*.

For **Hide / Show**, refer to Hiding Objects, for **Properties**, refer to Displaying and Editing Graphic Properties, and for **Other Selection**, refer to Selecting Using the Other Selection... Command. All these chapters can be found in the *CATIA Infrastructure*

User's Guide.

These commands can also be accessed contextually from the specification tree.



Selecting Using Multi-Selection

This capability enables you to perform multi-selection of elements and validate the selection.

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.
- **Outside Trap Selection** : enables you to select elements outside the trap.
  Any object strictly outside the trap will be selected.
- **Intersecting Outside Trap Selection** : enables you to select elements outside the trap.
  Any object strictly outside or partially outside the trap will be selected.
- **Control Mode** : enables you to validate any selection at once.

- **List of selected items** : enables you to display the dialog box containing the list of selected items.
  The number of selected items is displayed in the field to the left of this icon.
  If there are less than one selected element, this button is disabled.

- **Finish** : enables you to finish and validate the multi-selection.

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.

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
- All transformations: translation, rotation, symmetry, scaling, affinity and axis to axis
- Developed wires

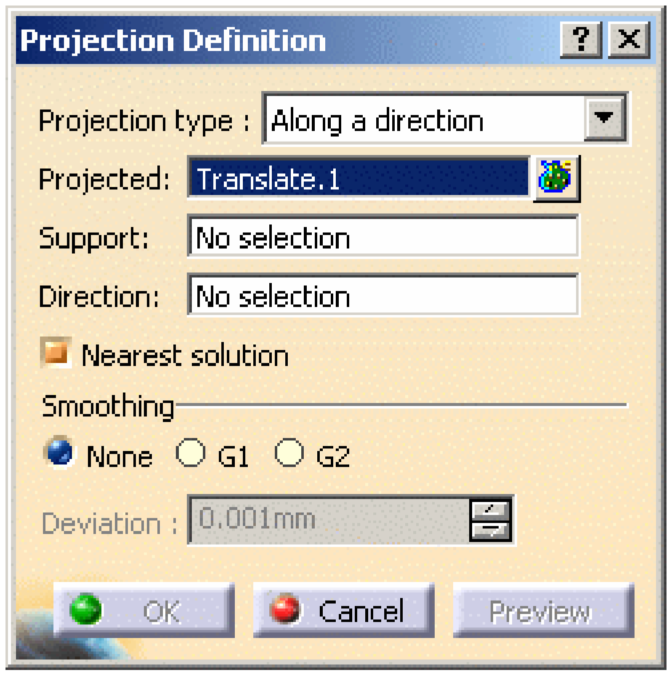
Let's take an example using the Projection and Translation functionalities.

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**.



3. Click the **Projected** field again, or

3. Click the bag icon to display the elements list.

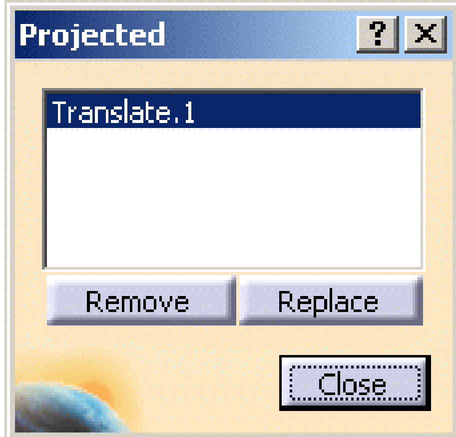
   The Projected dialog box opens.

4. Select as many elements as you need for your projection.

5. Click **Close** to return to the Projection Definition dialog box.

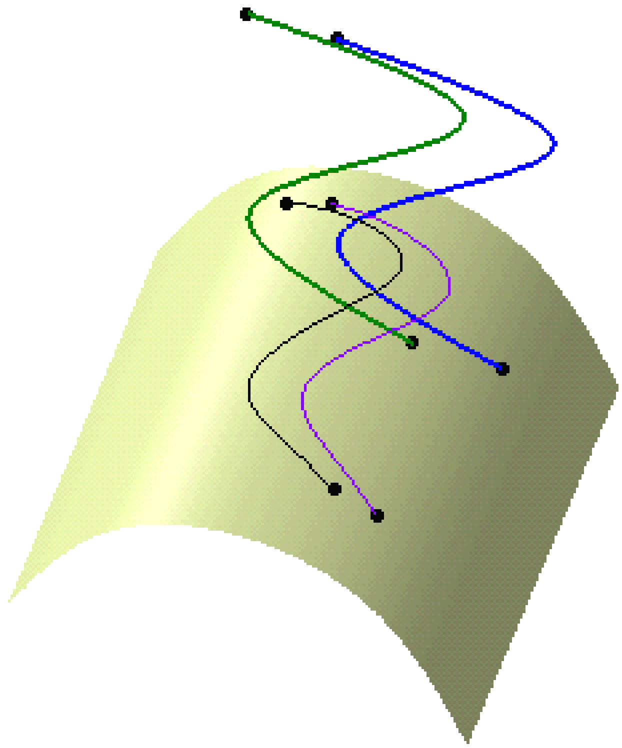
The number of selected elements is displayed in the

Projected field.



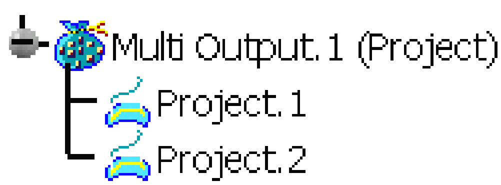
Use the **Remove** and **Replace** buttons to modify the elements list.

You can now select one or more geometrical sets and multi-outputs as inputs of the multi-selection.
In that case, all their direct children are selected.



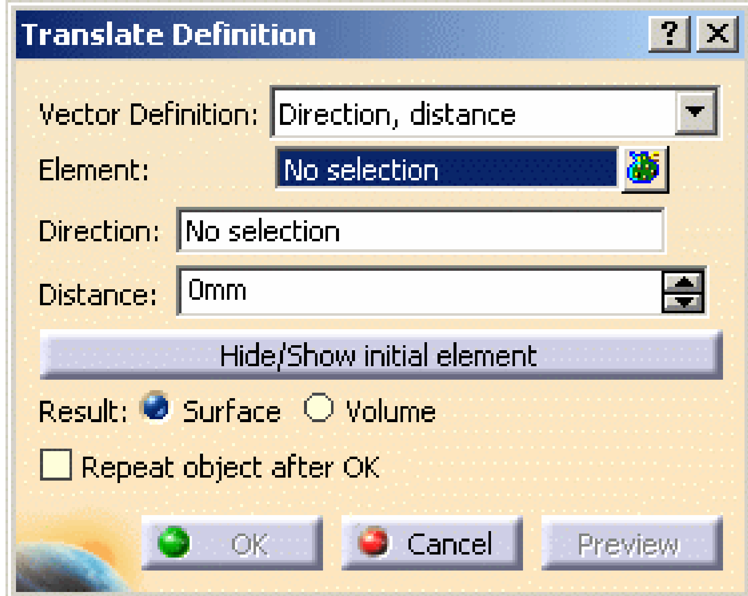
6. Select Extrude.1 as the **Support** element.
7. Select **Normal** as Projection type.
8. Click **OK** to create the projection elements.

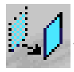
The projection is identified as Multi Output.1 (Project) in the specification tree.



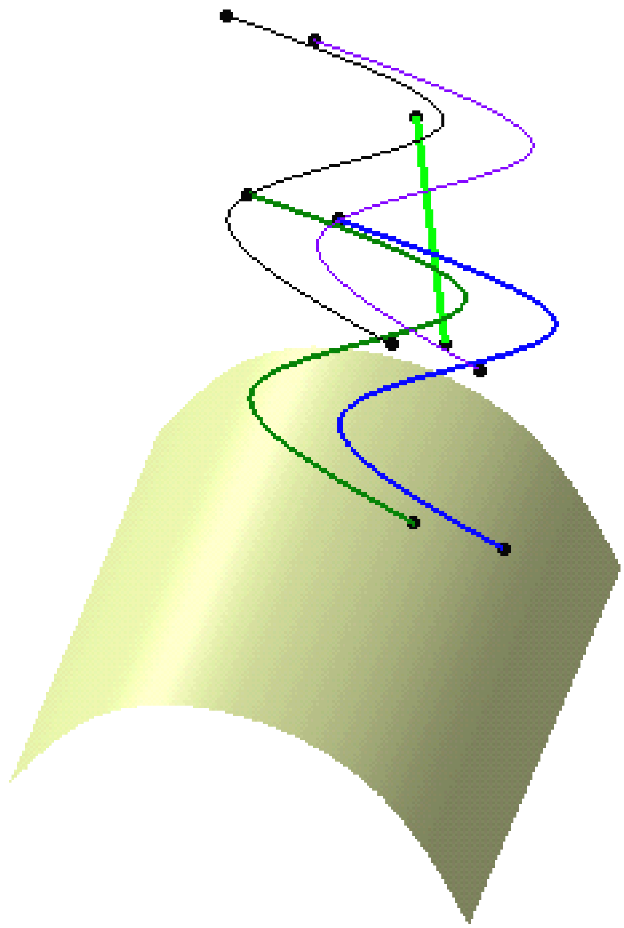
The created elements are aggregated under Multi Output.1.

You can create several multi-outputs in the specification tree, each one grouping one type of elements.



9. Click the **Translate** icon .

   The Translate Definition dialog box appears.



10. Select Translate.1 and Translate.2 as the **Elements** to be translated.
11. Select Direction, distance as the **Vector Definition**.
12. Select Extract.2 as the **Direction**.
13. Select -50mm as the **Distance**.
14. 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 displayed.

- You can deactivate all the elements of a multi-output. As a consequence, the multi-output feature disappears from the 3D geometry and erroneous elements can no longer be generated.
  Similarly, you can activate all the elements of a deactivated multi-output.
  To have further information on deactivation, please refer to the *Deactivating Features* chapter.

- 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.

Managing Multi-Result Operations

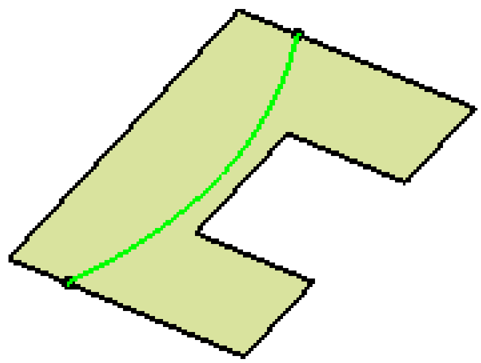
This task shows you how to manage the result of an operation in the case this result is not connex.

Several possibilities are offered:

- keep all the sub-elements
- keep one sub-element using the Near command
- keep one sub-element using the Extract command
- use pointing elements and select a sub-element to keep

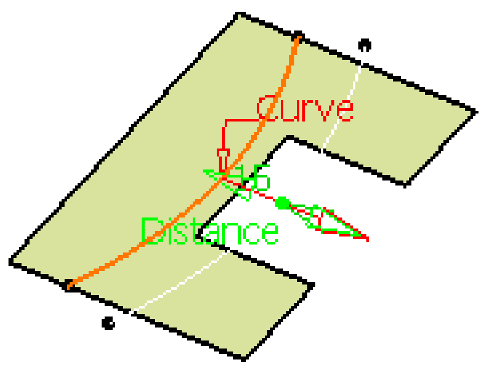
Keeping all sub-elements

A surface and a spline lying on this surface are created.
A parallel curve of the spline is to be created.



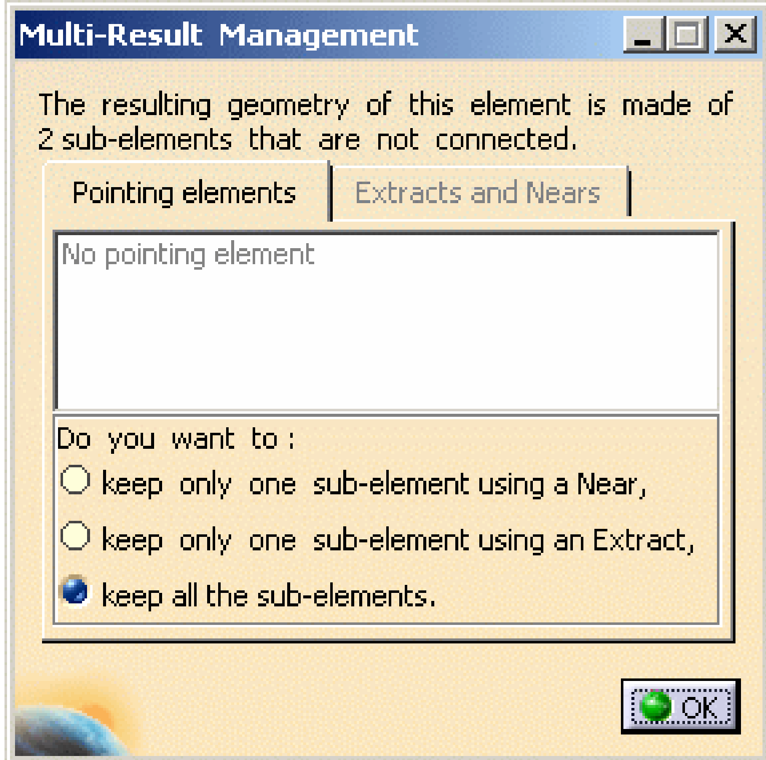
1. Click the **Parallel Curve** icon .

   The Parallel Curve Definition dialog box is displayed.



2. Select the spline as the **Curve**.
3. Select the surface as the **Support**.
4. Click OK.

The Multi-Result Management dialog box appears.



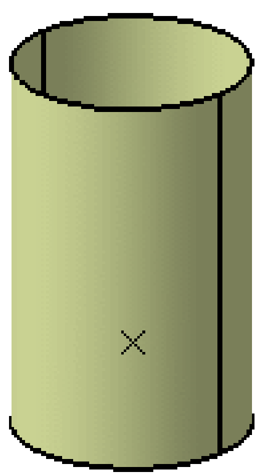
5. Check the **keep all the sub-elements** option to keep a non connex result.
6. Click OK.

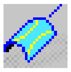
   The curve (identified as Parallel.xxx) is added to the specification tree.

Keeping one sub-element using the Near command

The multi-result feature has no children

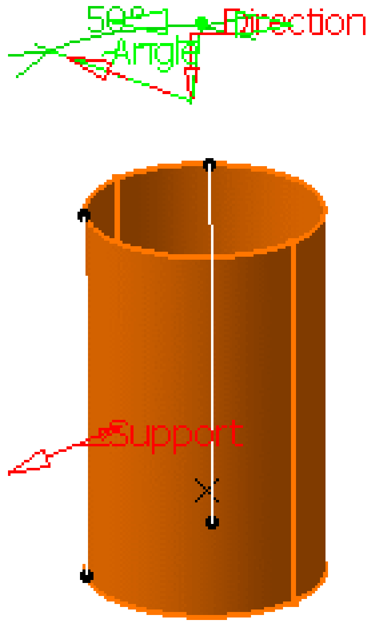
A cylinder is created.
Reflect lines on this cylinder are to be created.



1. Click the **Reflect Lines** icon .

   The Reflect Line Definition dialog box is displayed.

2. Select the cylinder as the **Support**.
3. Select a direction.
4. Define 50 as the **Angle** value.
5. Click OK.



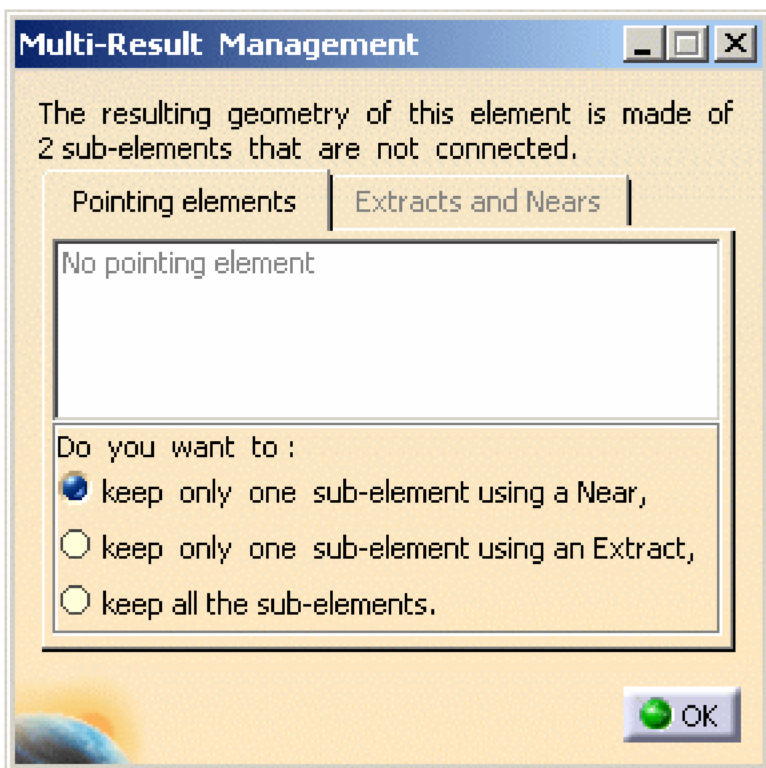
The Multi-Result Management dialog box appears.

6. Check the **keep only one sub-element using a Near** option to create a nearest entity of the multiple element, that is the reflect lines.
7. Click OK.

   The Near Definition dialog box appears and the reflect line is automatically filled in the **Multiple Element** field.

8. Select a plane as the **Reference Element**.
9. Click OK.

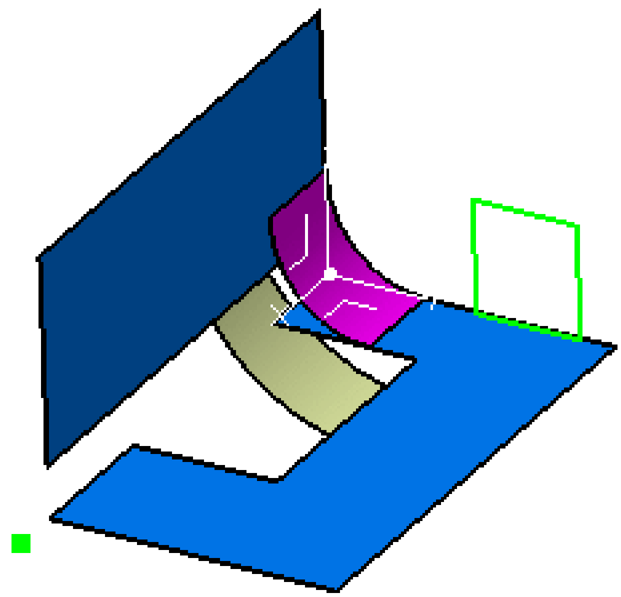
   The line (identified as Reflect Line.xxx) and the nearest element (identified as Near.xxx) are added to the specification



tree. The reflect line is put in no show.

The multi-result feature has several children

Open the Multi-Result1.CATPart document.

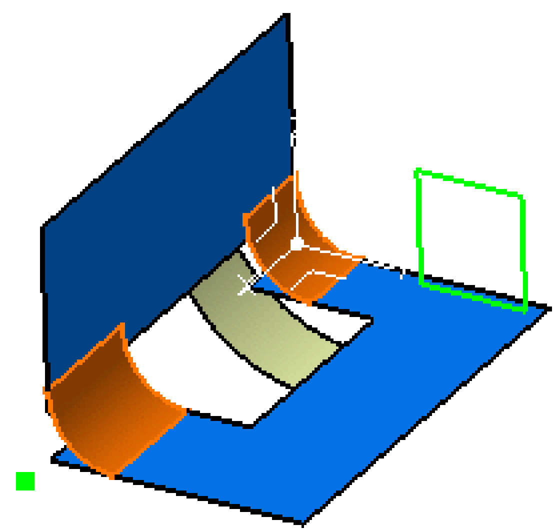


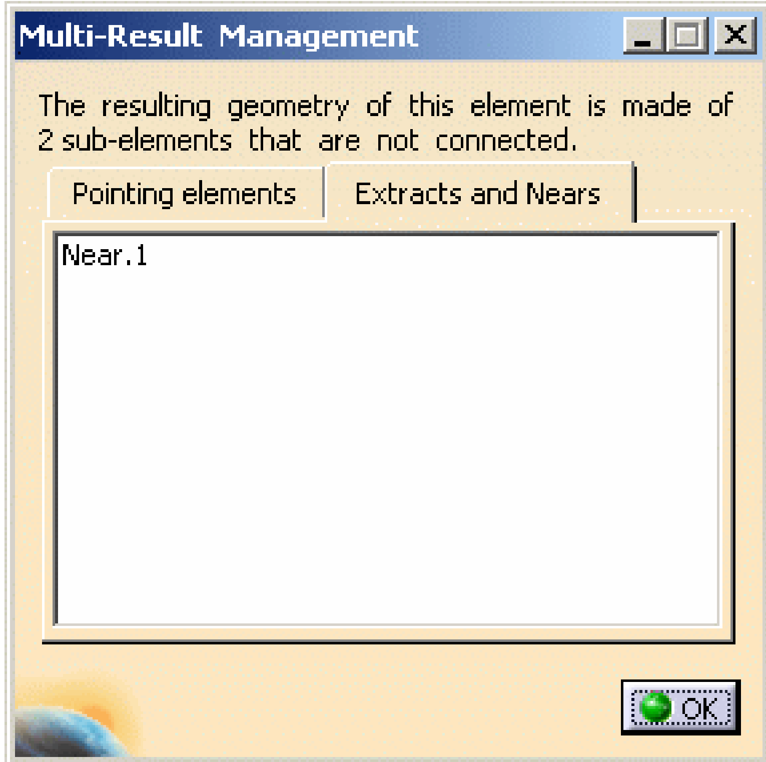
1. Double-click the Fillet.1 in the specification tree.

   The Fillet Definition dialog box opens.

2. Modify the radius value: set it to 15mm.

3. Click OK.





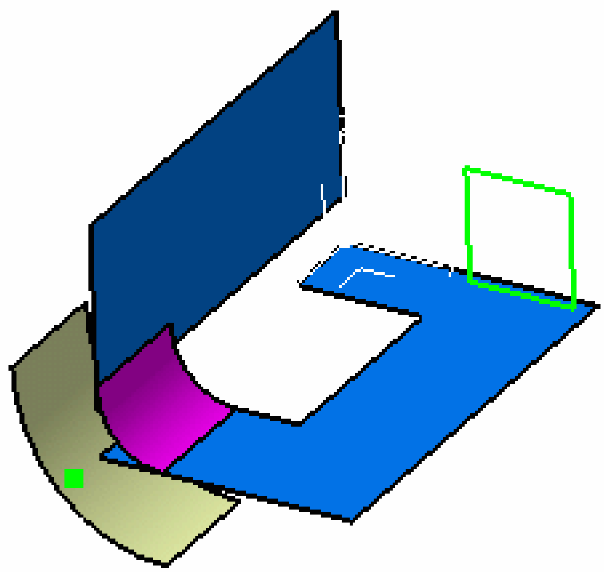
The Multi-Result Management dialog box appears.
The multi-result feature contains a near that is displayed in the Extracts and Nears tab.

4. Double-click Near.1.

   The Near Definition dialog box appears and the fillet is automatically filled in the **Multiple Element** field.

5. Select Point.1 as the **Reference Element**.

6. Click OK.



7. Click OK in the Multi-Result Management dialog box.

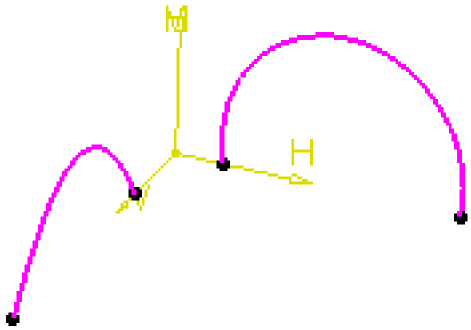
   The Offset element, that uses the Near element as the surface to be offset, is modified accordingly.

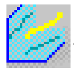
For further information about the Near command, refer to the Creating the Nearest Entity of a Multiple Element chapter in the *Generative Shape Design* documentation.

Keeping one sub-element using the Extract command

The multi-result feature has no children

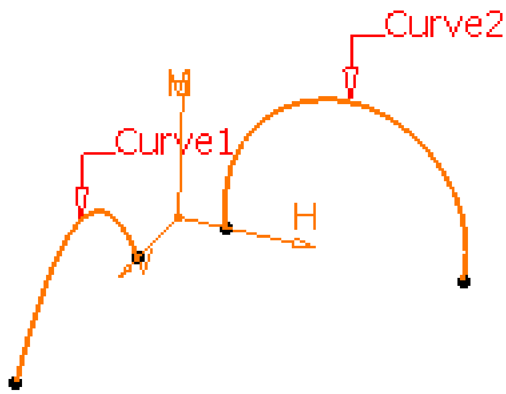
Two sketches are created.
A combine curve is to be created between them.



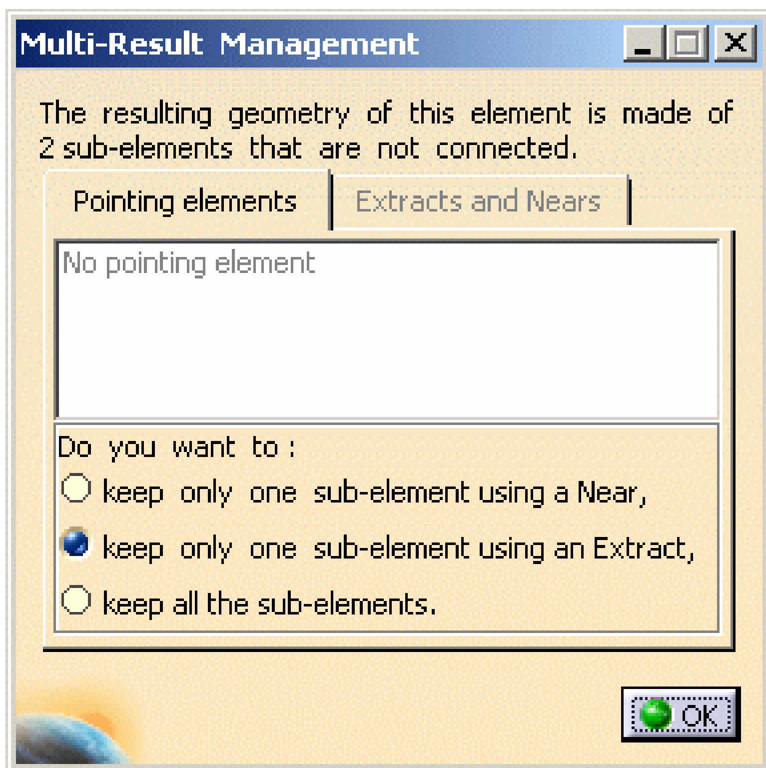
1. Click the **Combine** icon .

   The Combine Definition dialog box is displayed.

2. Select the **Normal** type.
3. Successively select the two curves to be combined.
4. Uncheck the **Nearest solution** option.
5. Click OK.



The Multi-Result Management dialog box appears.



6. Check the **keep only one sub-element using an Extract** option to create an extract of the multiple element, that is the combine curves.
7. Click OK.

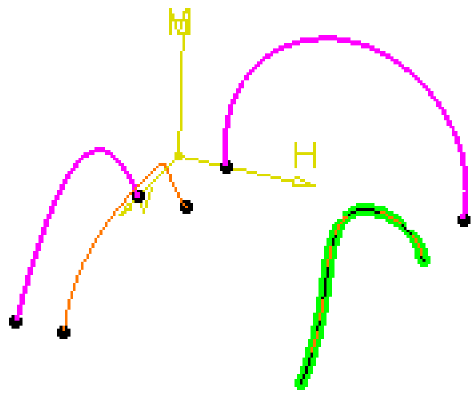
   The Extract Definition dialog box appears.

8. Select one of the combine curves in the 3D geometry as the Element to extract.

   The selected element is highlighted.

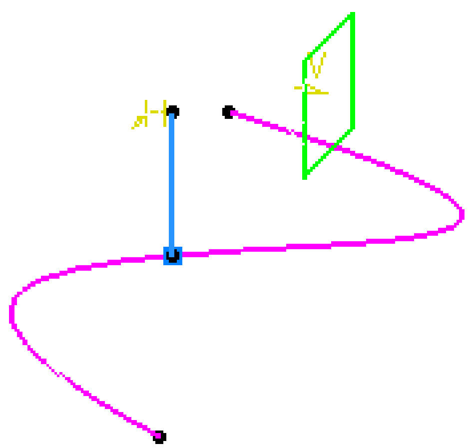
9. Click OK.

   The curve (identified as Combine.xxx) and the extracted element (identified as Extract.xxx) are added to the specification tree. The combine curve is put in no show.



The multi-result feature has several children

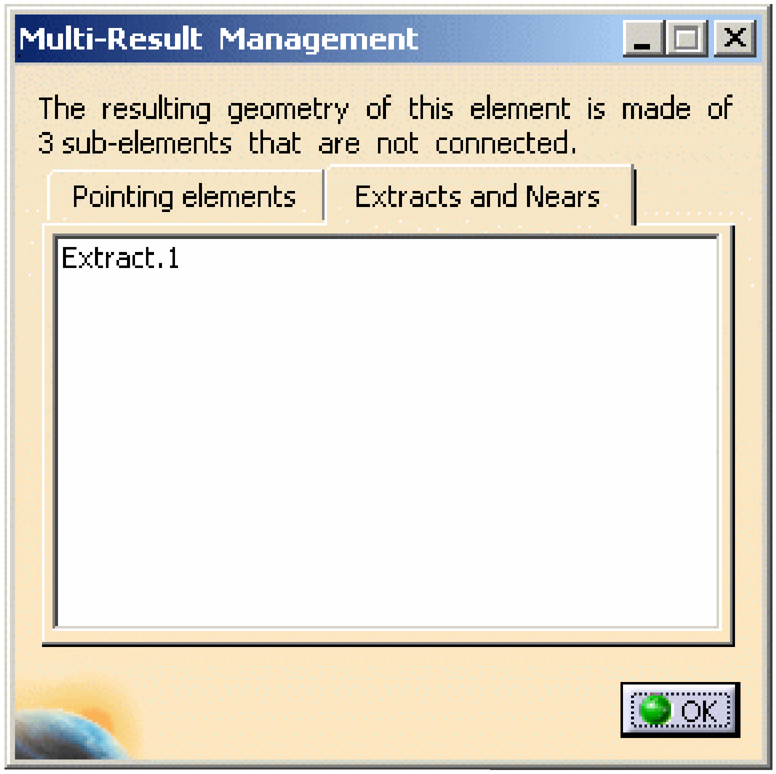
Open the Multi-Result2.CATPart document.



1. Double-click Intersect.1.

   The Intersection Definition dialog box appears.

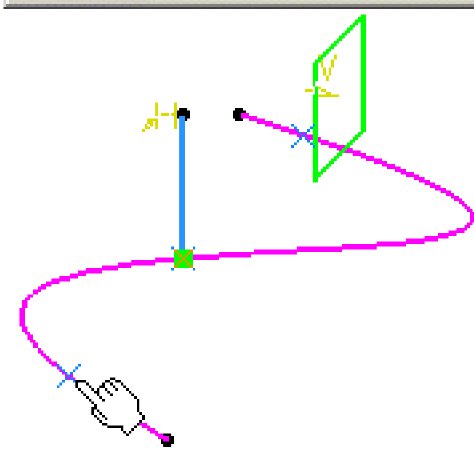
2. Click OK.



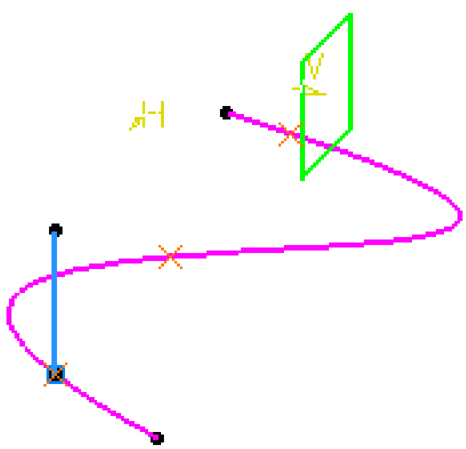
The Multi-Result Management dialog box appears.
The multi-result feature contains an extract that is displayed in the Extracts and Nears tab.

3. Double-click Extract.1.

   The Extract Definition dialog box opens.



4. Select another vertex as the **Element to extract**, as shown besides.
5. Click OK.



6. Click OK in the Multi-Result Management dialog box.

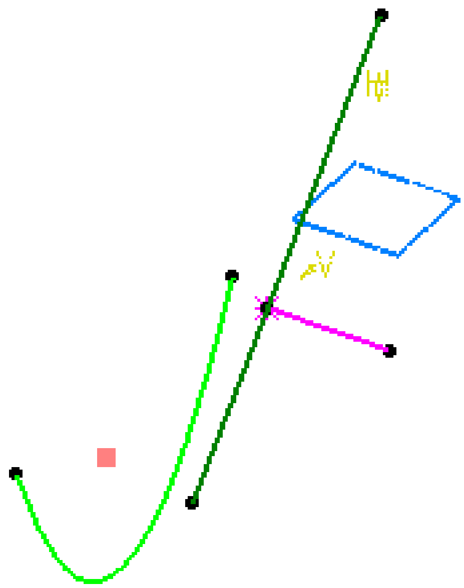
   The line, that uses the Extract element as the point, is modified accordingly.

For further information about the Extract command, refer to the Extracting Geometry chapter in the *Generative Shape Design* documentation.

Using pointing element and select a sub-element to keep

Using a Near

Open the Multi-Result3.CATPart document.

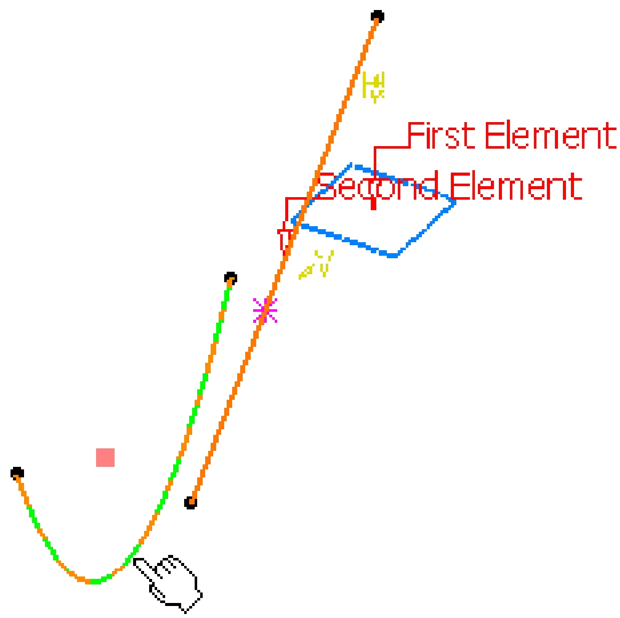


1. Double-click Intersect.1.

   The Intersection Definition dialog box appears.

2. Replace Line.1 by Spline.1 as the **Second element** by selecting it in the 3D geometry.

3. Click OK.



The Multi-Result Management dialog box appears.

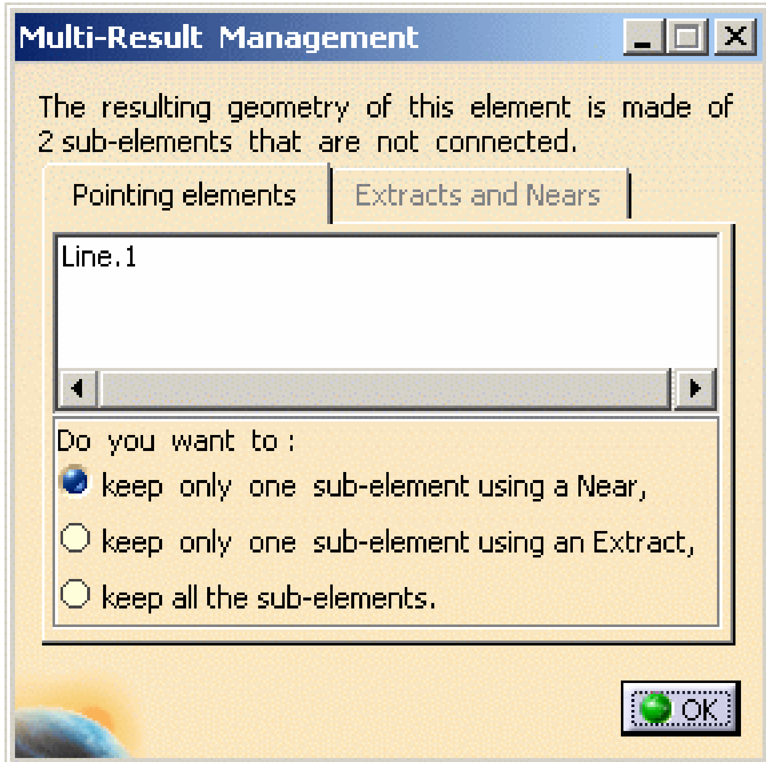
4. Double-click Line.1 to modify its specifications.

   The Line Definition dialog box opens.

5. Replace Intersect.1 by Point.1 as the **Point**.

6. Click OK.

   In the Multi-Result Management dialog box, there is no pointing element any more.



7. Check the **keep only one sub-element using a Near** option to create a nearest entity of the multiple element, that is Intersect.1.

8. Click OK.

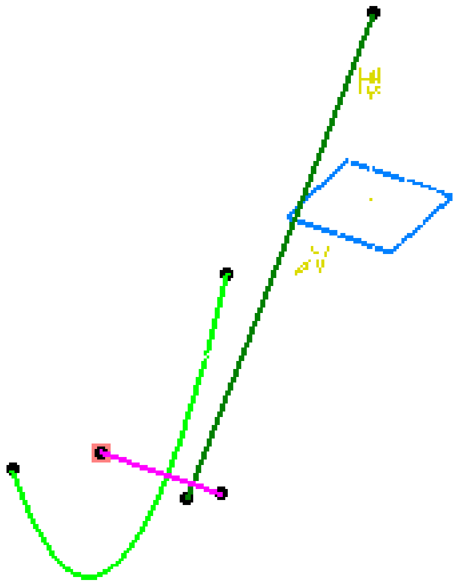
   The Near Definition dialog box appears and the intersect element is automatically filled in the **Multiple Element** field.

9. Select the axis system as the **Reference Element**.

10. Click OK.

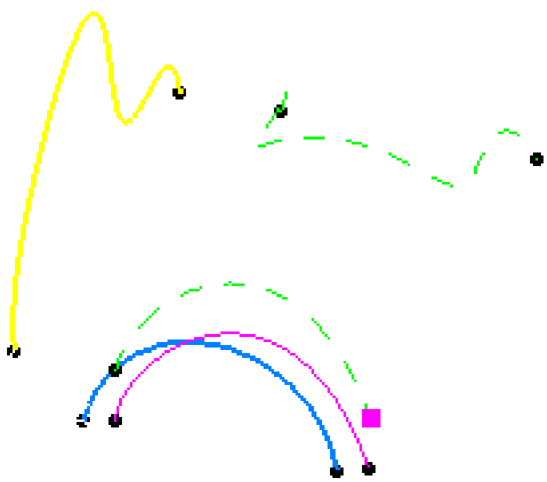
The nearest element (identified as Near.xxx) is added to the specification tree before the line and is set as current.

The line, that now uses Point.1, is modified accordingly.



Using an Extract

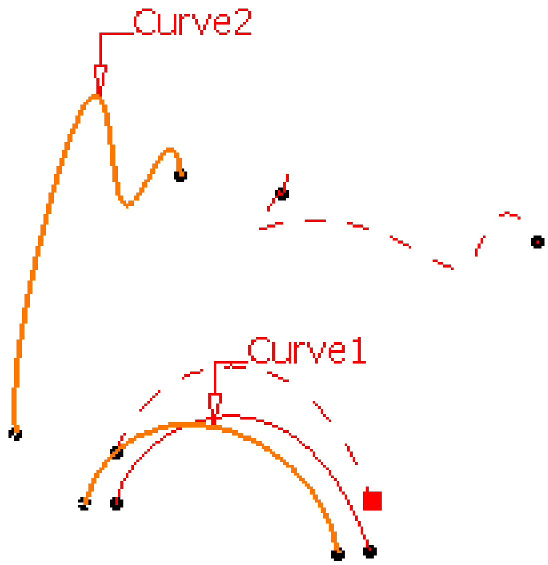
Open the Multi-Result4.CATPart document.



1. Double-click Combine.1.

   The Combine Definition dialog box appears.

2. Click OK.



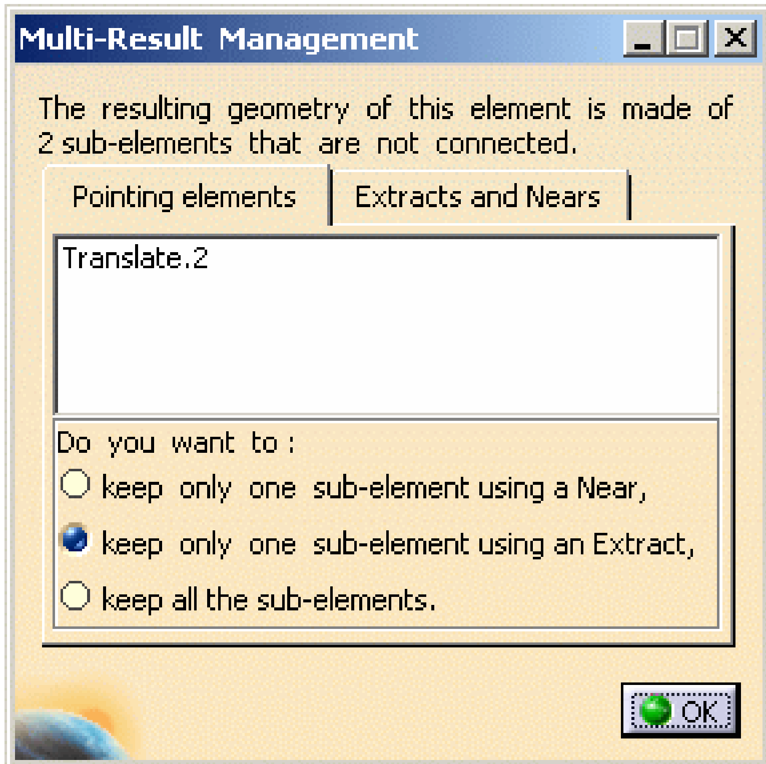
The Multi-Result Management dialog box appears.

3. Check the **keep only one sub-element using an Extract** option to create an extract of the multiple element, that is the combine curves.

4. Click OK.

   The Extract Definition dialog box appears and the combine element is automatically filled in the **Multiple Element** field.

5. Select the axis system as the **Reference Element**.



6. Select one of the combine curves in the 3D geometry as the **Element to extract**.

   The selected element is highlighted.

7. Click OK.

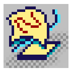
   The extracted element (identified as Extract.xxx) is added to the specification tree.

- Several multi-result features can be contained within a multi-output.
- The options set in the dialog box are retained when exiting then returning to the Multi-Result Management function.

Advanced Tasks

The advanced tasks you will perform in the Generative Shape Design workbench include managing the specification tree and interoperating with other workbenches.

Managing Geometrical Sets and Ordered Geometrical Sets



Manage geometrical sets: select a Geometrical Set in the specification tree, use the **Insert -> Geometrical Set** menu command, or **Remove Geometrical Set** or **Change Geometrical Set** contextual menus.

Manage ordered geometrical sets: select an Ordered Geometrical Set in the specification tree, use the **Insert -> Ordered Geometrical Set** menu command.

Insert a body into an ordered geometrical set: select the **Insert -> Body in a Set** menu command and enter the name of the body you wish to insert into the ordered geometrical set.

Duplicate geometrical sets: click this icon and select the geometrical set to be duplicated.

Hide/show geometrical sets or ordered geometrical sets and their contents: select the geometrical set or the geometrical set to be hidden and use the Hide/Show capabilities.

Managing Geometrical Sets

Geometrical sets enable to gather various features in a same set or sub-set and organize the specification tree when it becomes too complex or too long. You can put any element you wish in the geometrical set, it does not have to be structured in a logical way. The order of these elements is not meaningful as their access as well as their visualization is managed independently and without any rule.

This task shows how to manage geometrical sets within the specification tree. This involves:

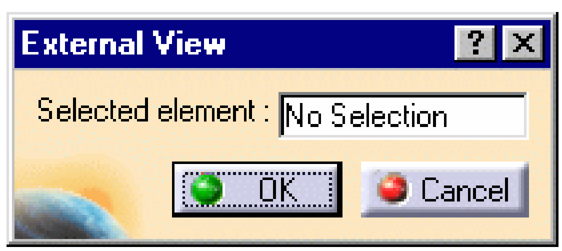
- inserting a geometrical set
- removing a geometrical set
- changing body
- sorting the contents of a geometrical set
- reordering components

You will find other useful information in the Managing Groups and Hiding/Showing chapters.

- You can insert and manipulate geometrical sets 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 geometrical sets can be used in a part edition context.
- When loading the Generative Shape Design workbench, a Geometrical Set automatically becomes the current body.
  This also means that only the results of the Hybrid Body, i.e. the result of all the operations performed on geometry, is visible and not any intermediate state of the Hybrid Body.
- You can define the Generative Shape Design feature that is to be seen when working with another application, such as Generative Structural Analysis for example.

To do this, while in the Generative Shape Design workbench:

1. Choose the **Tools -> External View...** menu item
   The External View dialog box is displayed.
2. Select the element belonging to a Geometrical Set that should always been seen as the current element when working with an external application.
3. Click OK in the dialog box.



The selected element will be the visible element in other applications, even if other elements are created later in the .CATPart document, chronologically speaking.

To check whether an external view element has already been specified, choose the **Tools -> External View...** menu item again. The dialog box will display the name of the currently selected element. This also allows you to change elements through the selection of another element. Note that you cannot deselect an external view element and that only one element can be selected at the same time.

Open any .CATPart document containing Geometrical Sets.
You can also open the GeometricalSets2.CATPart document.

Inserting a Geometrical Set

1. In the specification tree, select an element as the location of the new geometrical set.

   This element will be considered as a child of the new geometrical set and can be a geometrical set or a feature.

2. Select the **Insert -> Geometrical Set** menu command.

   The Insert Geometrical Set dialog box is displayed.

   The Features list displays the elements to be contained in the new geometrical set.

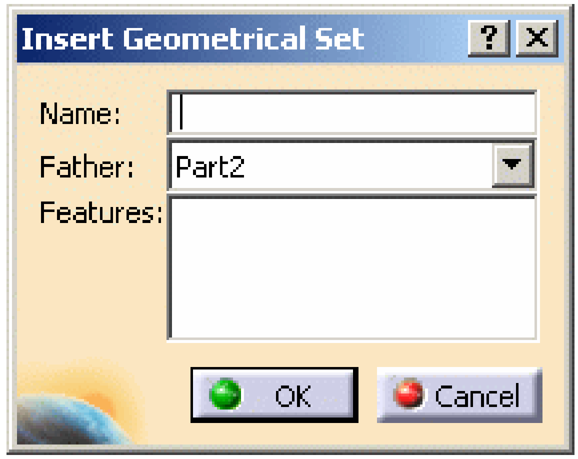
3. Enter the name of the new geometrical set.

4. Use the Father drop-down list to choose the body where the new geometrical set is to be inserted. All destinations present in the document are listed allowing you to select one to be the father without scanning the specification tree. They can be:
   - geometrical sets
   - parts

5. Select additional entities that are to be included in the new geometrical set.

If all selected entities belong to the same geometrical set, the father of the new geometrical set is automatically set to the father of these entities.

6. Click OK to create the geometrical set at the desired location.



The result is immediate. CATIA displays this new Geometrical Set.x, incrementing its name in relation to the pre-existing bodies, in the specification tree. It is created after the last current geometrical set and is underlined, indicating that it is the active geometrical set. The next created element is created within this geometrical set.

- You cannot create a geometrical set within an ordered geometrical set and vice versa.
- This Insert Geometrical Set dialog box is only available with the Generative Shape Design 2 product.

You can check the **Create a Geometrical Set when creating a new part** option in **Tools -> Options -> Infrastructure -> Part Infrastructure -> Part Document** tab if you wish to create a geometrical set as soon as you create a new part. For more information about this option, please refer to the Customizing section of the *Part Design User's Guide*.

Removing a Geometrical Set

Two methods are available:

- If you want to delete the geometrical set and all its contents:

1. Right-click the geometrical set then select the **Delete** contextual command.

- If you want to delete the geometrical set but keep its contents:
  This is only possible when the father location of the geometrical set is another geometrical set. This is not possible when the father location is a root geometrical set.

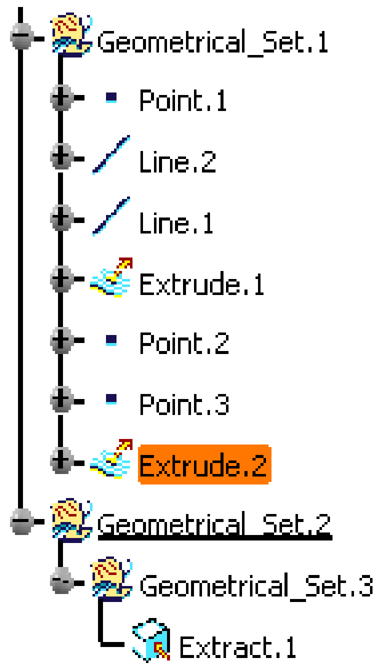
1. Right-click the desired geometrical set then select the **Geometrical Set.x object -> Remove Geometrical Set** contextual command.

The geometrical set is removed and its constituent entities are included in the father geometrical set.

You cannot delete a feature within a geometrical set created on the fly. Indeed this geometrical set is considered as private and can only be deleted globally.

Moving a Geometrical Set to a New Body

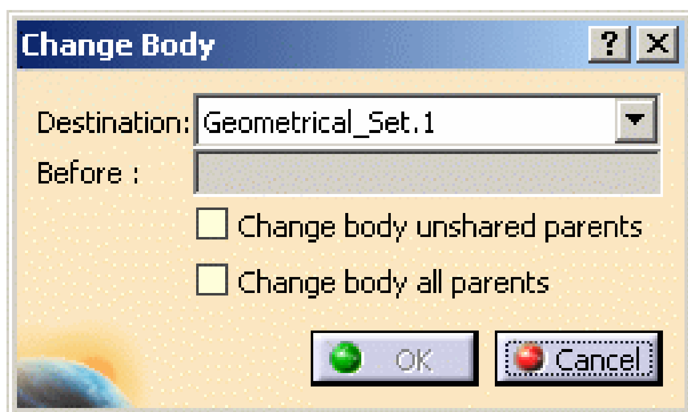
1. From the specification tree, select the element then choose the **Geometrical Set.object -> Change Geometrical Set...** item from the contextual menu.



Multi-selection of elements of different types is supported. However, note that in this case, the contextual menu is not available, and that you can access this capability using the **Edit** menu item.

The Change Body dialog box is displayed.

The list of destinations is alphabetically sorted.



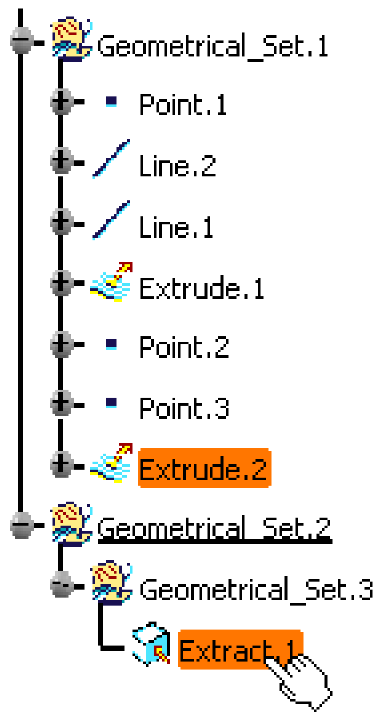
2. Select the **Destination** body where the geometrical set is to be located.

   Here we selected Geometrical_Set.3.

You can do so by selecting the Body in the specification tree, or using the drop-down list from the dialog box.

By default, if you select a body, the geometrical set is positioned last within the new body. However, you can select any element in the new body, before which the moved geometrical set will be located.

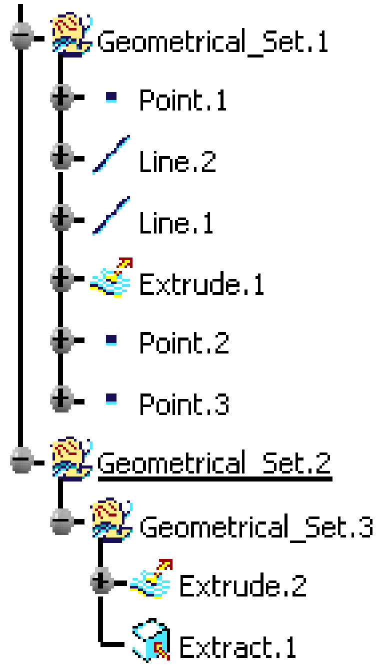
3. Select the element above which the one you already selected is to be inserted.



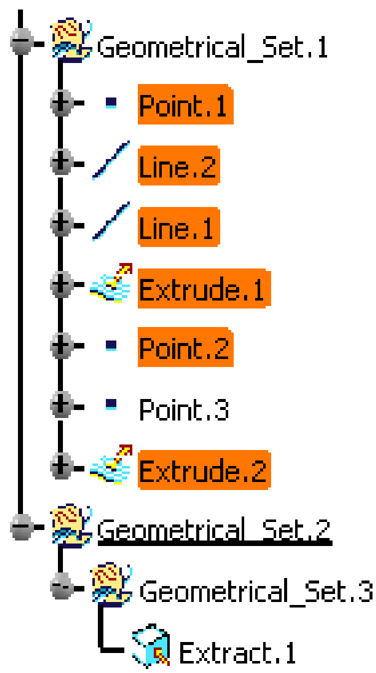
You can directly select this positioning element. In this case the **Destination** field of the Change Body dialog box is automatically updated with the Body to which this second element belongs.

4. Click OK to move the geometrical set to the new body.

The element selected first is moved to its new location in the specification tree, but geometry remains unchanged.

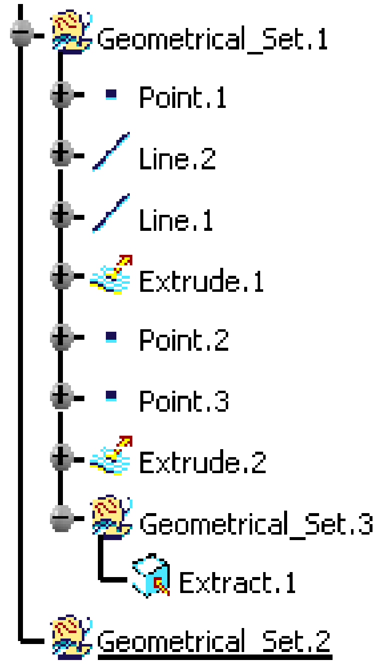


- Check the **Change body unshared parents** option to move all parents of the first selected element to its new location, provided these parents are not shared by any other element of the initial body.
  In this case, all the unshared parents are highlighted prior to the move.



- Check the **Change body all parents** option to move all parents of the first selected element to its new location, regardless of whether these parents are used (shared) by any other element of the initial body.
  In this case, all the parent elements are highlighted prior to the move.

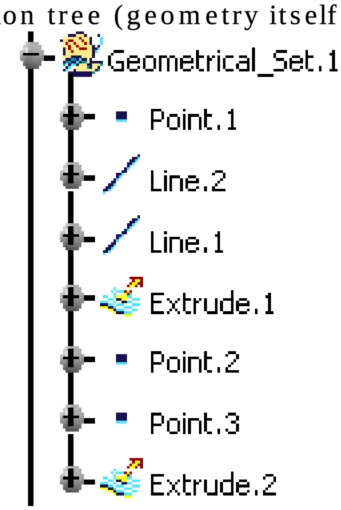
- You can move a whole branch, i.e. a whole body and its contents, at a time.
  Here we moved Geometrical_Set.3 last in Geometrical_Set.1.



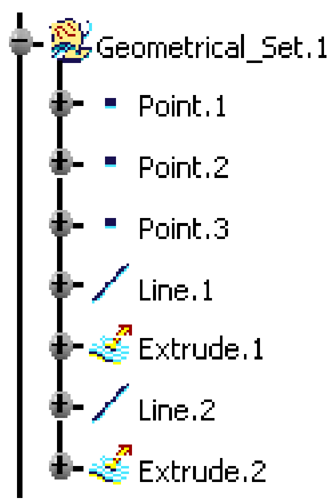
Sorting the Contents of a Geometrical Set

You may need to sort the contents of a Geometrical Set, when the geometric elements no longer appear in the logical creation order. In that case, use the Auto-sort capability to reorder the Geometrical Set contents in the specification tree (geometry itself is not affected).

The Geometrical_Set.1 contains two extruded surfaces based on point-point lines. The specification tree looks like this:



1. Right-click the Geometrical_Set.1 from the specification and choose the **Geometrical_Set.1 object -> AutoSort** command.



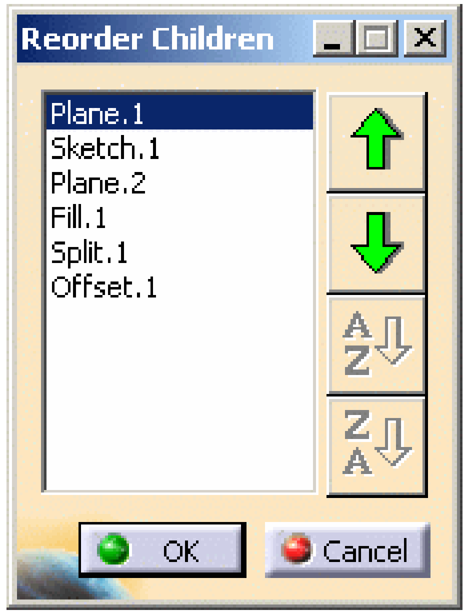
Instantly, the contents of the Geometrical Set are reorganized to show the logical creation process.

The geometry remains unchanged.

Reordering Components within a Geometrical Set

This capability enables you to reorder elements inside the same geometrical set.

1. Right-click the Geometrical_Set.1 from the specification tree and choose the **Ordered Geometrical Set.1 object -> Reorder Children** command.
   The Reorder Children dialog box is displayed.



2. Select an element.
3. Use the arrows to move an element up or down.

Reordering Features

The Reorder command allows you to move a feature in a Geometrical Set. These features can be:

- solids
- shape features
- sketches

Replacing Features

This capability is only available on shape features.

Please refer to the Replacing or Moving Elements chapter in the *Part Design User's Guide*.
To manage this capability, the **Do replace only for elements situated after the In Work Object** option is available in **Tools -> Options -> Part Infrastructure -> General** tab. It allows you to make the Replace option possible only for features located below the feature in Work Object and in the same branch.

Managing Ordered Geometrical Sets

Geometrical sets enable to gather various features in a same set or sub-set. The order of these features is not meaningful as their access as well as their visualization is managed independently and without any rule. However flexible, this structure does not fit the design process.
That is why ordered geometrical sets introduced notions of succession of steps that define the design, and absorption.
Creation features create a new object and modification features create a new state in an existing object as well as absorb the preceding state(s). Absorbed features are no longer visible nor accessible, as if "masked" by their absorbing feature.
In an ordered geometrical set, the order of apparition of features in the specification tree is consistent with the steps of creation of the design.
Unlike features within a geometrical set, features in an ordered geometrical set can be set as current: a given step of the design creation is chosen and what is located after it is not accessible nor visible.

This task shows how to manage ordered geometrical sets within the specification tree. This involves:

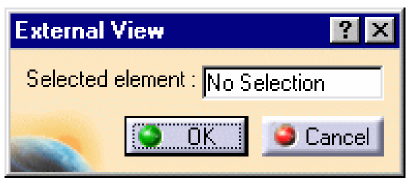
- inserting an ordered geometrical set
- defining an in work object
- visualizing features within an ordered geometrical set
- removing an ordered geometrical set
- removing a feature within an ordered geometrical set
- sorting the contents of an ordered geometrical set
- reordering components within an ordered geometrical set
- reordering features
- modifying children
- replacing features
- inserting and deleting inside and ordered geometrical set

You will find other useful information in the Managing Groups, Hiding/Showing, and Copying and Pasting chapters.

You can define the Generative Shape Design feature that is to be seen when working with another application, such as Generative Structural Analysis for example.

To do this, while in the Generative Shape Design workbench:

1. Choose the **Tools -> External View...** menu item.
   The External View dialog box is displayed.
2. Select the element belonging to a Geometrical Set that should always been seen as the current element when working with an external application.
3. Click OK in the dialog box.



The selected element will be the visible element in other applications, even if other elements are created later in the .CATPart document, chronologically speaking.

To check whether an external view element has already been specified, choose the **Tools -> External View...** menu item again. The dialog box will display the name of the currently selected element. This also allows you to change elements through the selection of another element. Note that you cannot deselect an external view element and that only one element can be selected at the same time.

Open any .CATPart document containing Geometrical Sets.
You can also open the OrderedGeometricalSets1.CATPart document.

Inserting an Ordered Geometrical Set

1. In the specification tree, select an element as the location of the new ordered geometrical set.

   This element will be considered as a child of the new ordered geometrical set.

   Inserting an Ordered Geometrical Set does not break the succession of steps as the order applies to all the elements of a same root ordered geometrical set.

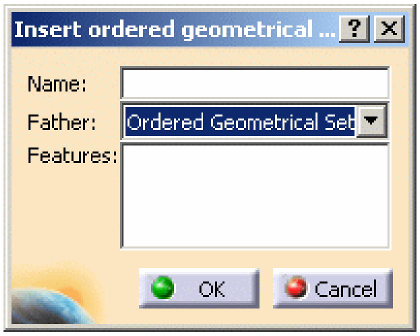
2. Select the **Insert -> Ordered Geometrical Set** menu command.

   The Insert ordered geometrical set dialog box is displayed.

   The Features list displays the elements to be contained in the new ordered geometrical set.

3. Enter the name of the new ordered geometrical set you wish to insert.

4. Use the Father drop-down list to choose the body where the new ordered geometrical set is to be inserted. All destinations present in the document are listed allowing you to select one to be the father without scanning the specification tree. They can be:
   - ordered geometrical sets
   - parts



By default the destination is the father of the current object. By default the ordered geometrical set is created after the current feature.

5. Select additional entities that are to be included in the new ordered geometrical set.

If all selected entities belong to the same ordered geometrical set, the father of the new ordered geometrical set is automatically set to the father of these entities.

6. Click OK to create the ordered geometrical set at the desired location.

The result is immediate. CATIA displays this new Ordered Geometrical Set.x, incrementing its name in relation to the pre-existing bodies, in the specification tree. It is created after the last current ordered geometrical set and is underlined, indicating that it is the active ordered geometrical set.

- You can insert an ordered geometrical set after the current feature.
- You cannot create an ordered geometrical set within a geometrical set and vice versa.

You can now insert a body into an ordered geometrical set.
For further information, refer to the Inserting a Body into an Ordered Geometrical Set chapter.

This Insert ordered geometrical set dialog box is only available with the Generative Shape Design 2 product.

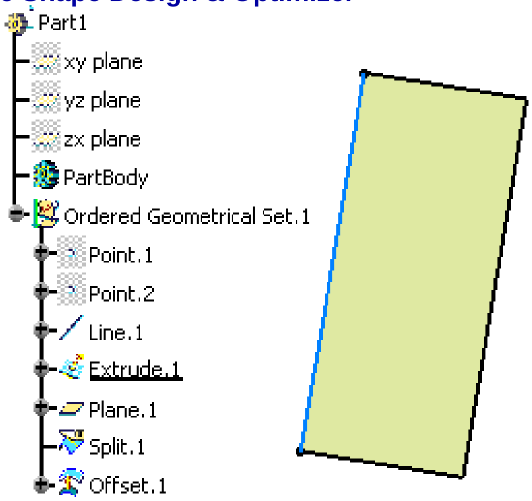
Defining an In Work Object

The next created element is created after the In Work object.
If the new feature to be inserted is a modification feature, features after the In Work object may be rerouted to the new created feature.

Visualizing features in an Ordered Geometrical Set

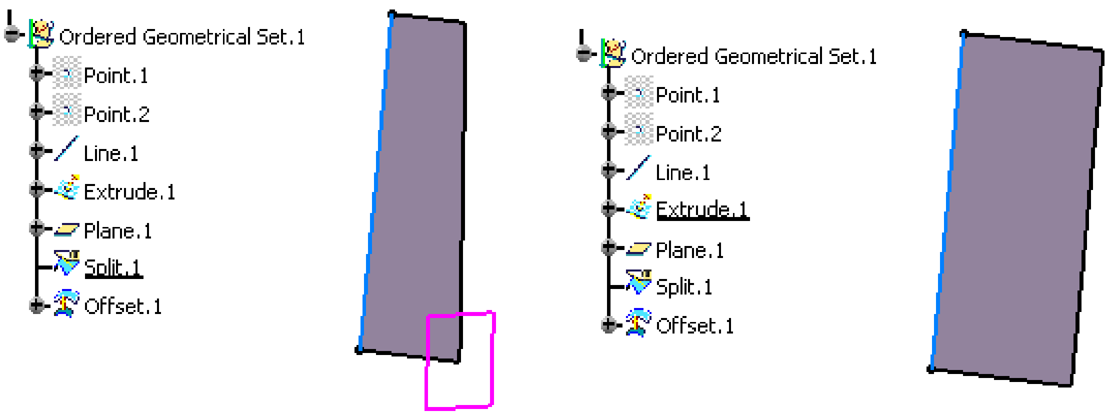
It can be useful to temporarily see its future geometry.
To do so, you can check the Future geometry option in **Tools -> Options -> Part Infrastructure -> Display** tab. It allows you to also display the geometry located after the current feature.

- Only features that come before the current object and that are not absorbed by any feature preceding the current object are visualized in the specification tree.



- A color assigned to a feature is propagated to all the features that successively modify this feature and so on. This is why it is possible to set a specific color only on creation features. Therefore, changing the color of a modification feature modifies the color of the initial state.

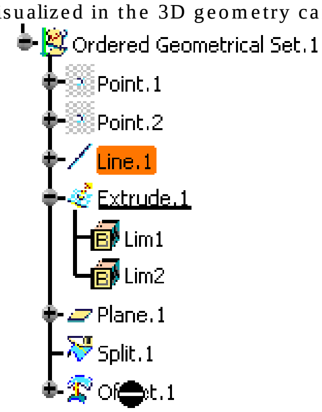
Here Extrude.1 is absorbed by Split.1. Therefore the color of Extrude.1 is propagated onto Split.1.



The same behavior applies on Show/No show attributes.

Selecting Features within an Ordered Geometrical Set

The selection of features located after the current feature or absorbed features is not possible.
Here, for instance, when editing Extrude.1, the selection of Offset.1 is not possible because Offset.1 is located after Extrude.1 which is the current object. A black sign indicates that this selection is not possible. Additionally, the application displays a tooltip explaining why it is not possible.
To ensure the consistency between the visualization in the 3D geometry and the selection in the specification tree, features that cannot be visualized in the 3D geometry cannot either be selected in the specification tree.



However, if you want to allow the selection of geometrical elements absorbed by other elements or the selection of geometrical elements located after the element being created, you can check the **Enable selection of drawn geometry** or **Enable the selection of future geometry** options in **Tools -> Options -> Part Infrastructure -> General** tab.
As a consequence, the succession of steps of the ordered geometrical set is no longer respected. We advise you not to check these options but rather work in a geometrical set environment.
If these options are unchecked, selecting elements whose geometry is not visible any more is not possible.

Removing an Ordered Geometrical Set

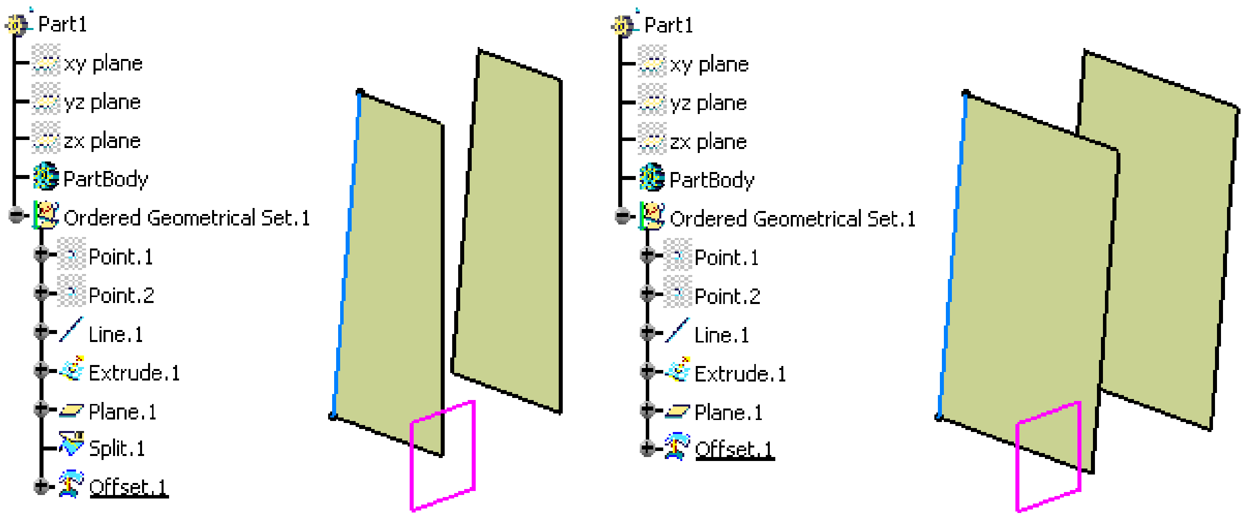
1. Right-click the ordered geometrical set then select the **Delete** contextual command.

   The ordered geometrical set and all its contents are deleted.

Removing a Feature within an Ordered Geometrical Set

1. Right-click the feature then select the **Delete** contextual command.

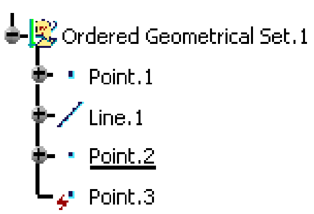
- deletion of a modification feature: the system reroutes the children on the element that is modified. Therefore the deleted feature will be replaced by the modified feature of upper level.
  In our scenario, Split.1 is deleted. As a consequence, Offset.1 now points Extrude.1.



- deletion of a creation feature: no reroute is possible.

Sorting the Contents of an Ordered Geometrical Set

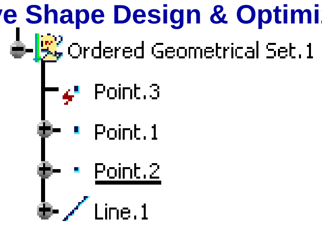
You may need to sort the contents of an ordered geometrical set, when the geometric elements no longer appear in the logical creation order. It may be the case if you enabled the selection of drawn or future geometry options. In that case, use the Auto-sort capability to reorder the ordered geometrical set contents in the specification tree.
The Ordered Geometrical Set.1 contains a line based on two points lines. The specification tree looks like this:



1. Right-click the Geometrical_Set.1 from the specification and choose the **Ordered Geometrical Set.1 object -> AutoSort** command.

Instantly, the contents of the Ordered Geometrical Set are reorganized to show the logical creation process.

The geometry remains unchanged. Datum features are put first in the specification tree.



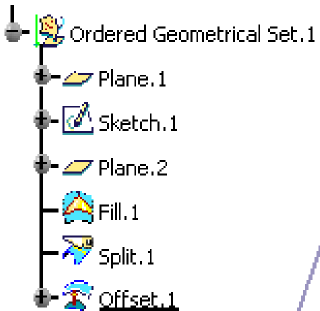
Reordering Components within an Ordered Geometrical Set

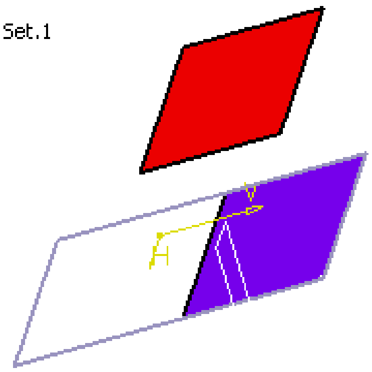
This capability enables you to reorder elements inside the same ordered geometrical set.

Reordering a creation feature based upon a modification feature

Open the Reorder1.CATPart document.

The Ordered Geometrical Set contains Split.1 (in blue) that splits Fill.1 by a purple vertical plane, and Offset.1 (in red) is an offset of Split.1.





1. Right-click the Ordered Geometrical Set.1 from the specification tree and choose the **Ordered Geometrical Set.1 object -> Reorder Children** command.

   The Reorder Children dialog box is displayed.

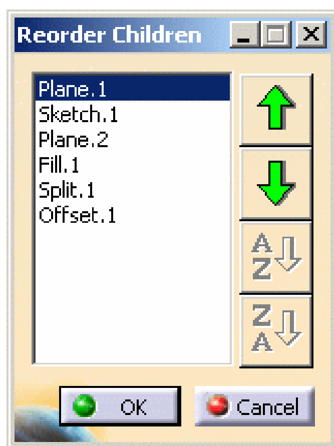
2. Select the element to be rerouted.

   Here we chose to reorder Offset.1 (creation feature) before Split.1 (modification feature).

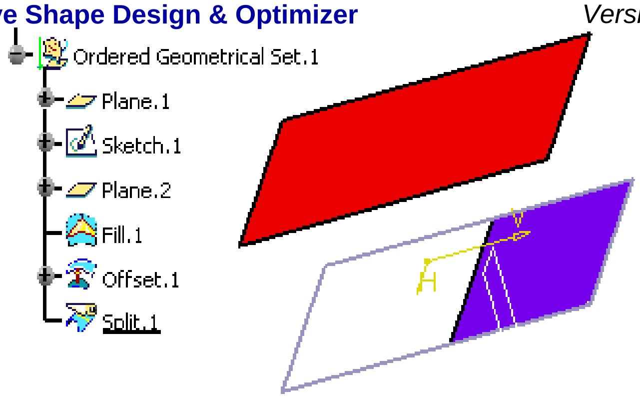
3. Use the arrow to move Offset.1 up.

4. Click OK.

   Offset.1 is now located before Split.1 in the specification tree.



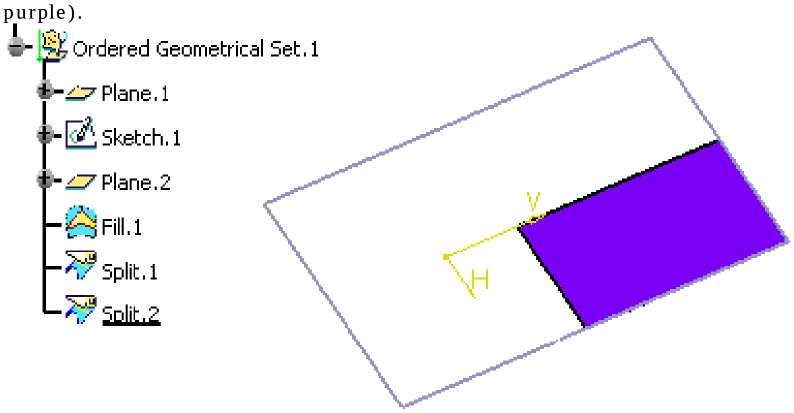
If you define Split.1 as the In Work object, you can see that Offset.1 is now based upon Fill.1.
Split.1 was not rerouted since Offset.1 does not modify Fill.1.



Reordering a modification feature based upon a modification feature

Open the Reorder2.CATPart document.

The Ordered Geometrical Set contains Split.2 (in blue) that splits Split.1 by a vertical plane. Split.1 itself splits Fill.1 (delimited by Sketch.1 in purple).



1. Right-click the Ordered Geometrical Set.1 from the specification tree and choose the **Ordered Geometrical Set.1 object -> Reorder Children** command.

   The Reorder Children dialog box is displayed.

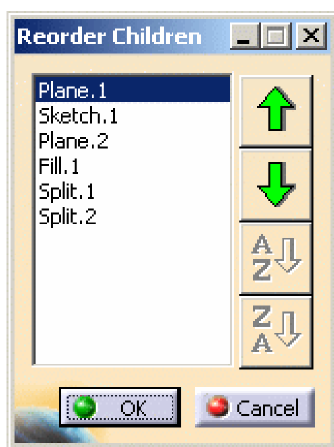
2. Select the element to be rerouted.

   Here we chose to reorder Split.2 (modification feature) before Split.1 (modification feature).

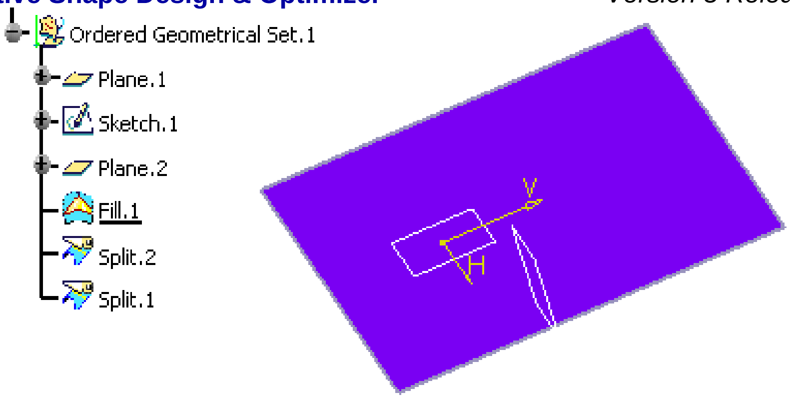
3. Use the arrow to move Split.2 up.

4. Click OK.

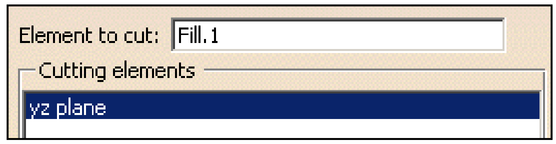
   Split.2 is now located before Split.1 in the specification tree.



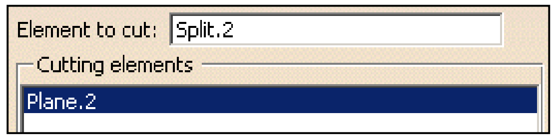
Split.2 is rerouted onto the input feature modified by Split.1, that is Fill.1 (in blue). Otherwise Split.2 would still split Split.1 which comes after in the specification tree.



Indeed, when you edit Split.2, you can notice that the Split.2 was rerouted onto Fill.1.



And since Split.2 now modifies Fill.1, Split.1 was rerouted onto Split.2.



An error message is issued if you try to move an element towards a position that breaks the order rules.

Note that the feature defined as the In Work object after the Reorder operation is not affected by this operation from an update point of view:

- when reordering upward, the feature located just before the new position of the reordered feature becomes the In Work object.
- when reordering downward, the feature just before the original position of the reordered feature becomes the In Work object.

You can use the Scan command after the Reorder operation to see what moved step by step.

Reordering Features

The Reorder command allows you to move a feature in an Ordered Geometrical Set. These features can be:

- Generative Shape Design features
- sketches

For further information, please refer to the Reordering Features chapter in the *Part Design User's Guide*.

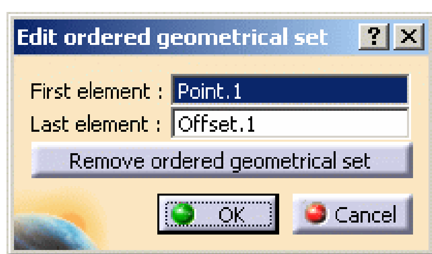
Modifying Children

The Modify Children command allows you to modify the contents of an ordered geometrical set by selecting its first and last component, as well as destroy it.

This command is only available on sub-ordered geometrical sets.

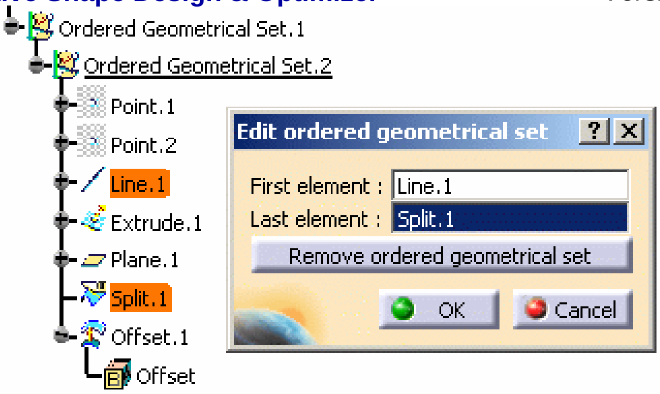
1. Right-click the sub-ordered geometrical set from the specification tree and choose the **Ordered Geometrical Set.x object -> Modifying children**.

   The Edit dialog box opens with the **First Element** and **Last element** fields automatically valuated with the first and last elements of the ordered geometrical set.



2. Select the elements you wish to place first and last.

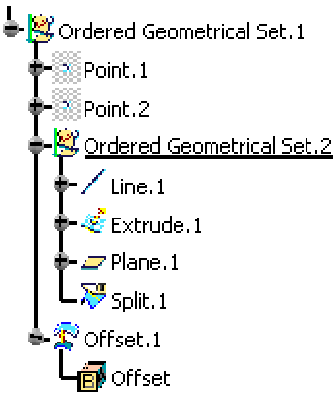
   In our scenario, we chose Line.1 as the first element and Split.1 as the last element.



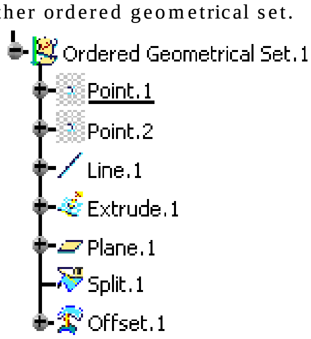
3. Click OK.

The specification tree is modified consequently.

Elements before or after the first and last elements are rerouted in the father ordered geometrical set.



The Modify children command also allows you to remove the sub-ordered geometrical set. As a consequence, elements are rerouted in the father ordered geometrical set.



Replacing Features

Please refer to the Replacing or Moving Elements chapter in the *Part Design User's Guide*.

The **Do replace only for elements situated after the In Work Object** option is available in **Tools -> Options -> Part Infrastructure -> General** tab. It restricts the Replace capability only on features located before the feature in Work Object and in the same branch.
As a consequence, the succession of steps of the ordered geometrical set is no longer respected. We advise you not to check this option but rather work in a geometrical set environment.

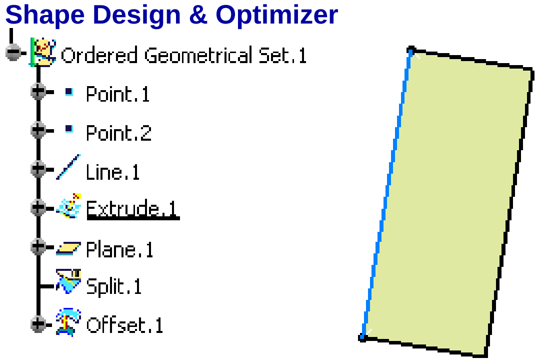
Switching From Ordered Geometrical Set to Geometrical Set

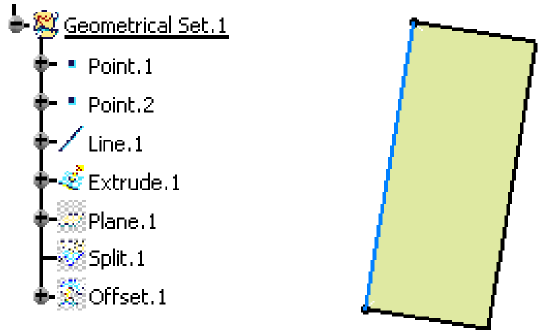
While in an ordered geometrical set environment, you may want to switch to a geometrical set environment (for instance, if you do not want to work in an ordered environment any more).

1. Right-click the Ordered Geometrical Set.1 from the specification tree and choose the **Ordered Geometrical Set.1 object -> Switch to geometrical set** command.

The Ordered Geometrical Set.1 becomes Geometrical Set.1.

Features after the current object that were not visualized in the ordered geometrical set are put in no show in the geometrical set.





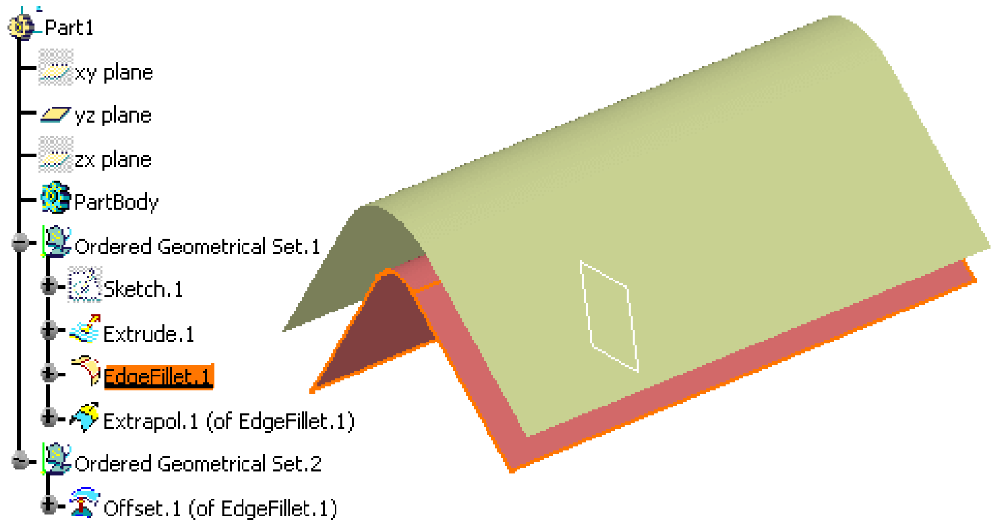
- This command is only available on a root ordered geometrical set.
- Switching from geometrical set to ordered geometrical set is not possible.
- Colors may be modified.

Inserting and Deleting Inside an Ordered Geometrical Set

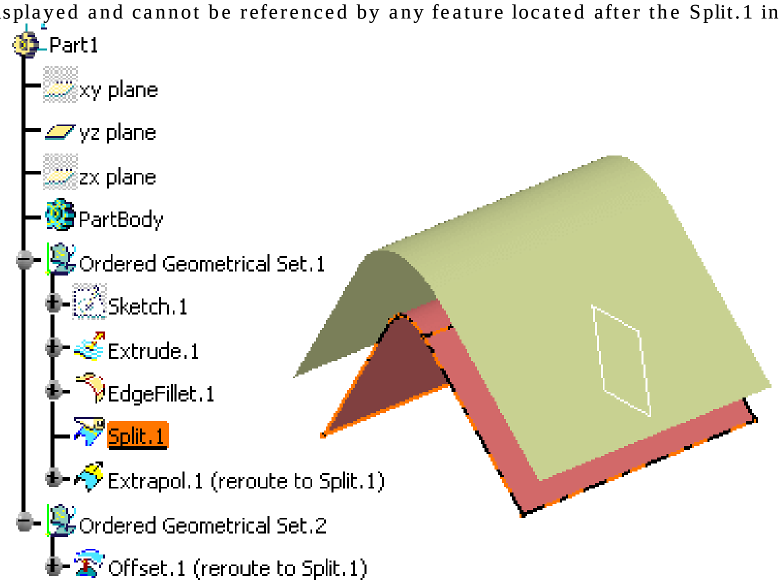
Inside an ordered geometrical set, the Insert and Delete commands may have impacts that result in replace actions based on absorption rules.

Open the OrderedGeometricalSets2.CATPart document.

Here, the edge fillet (Edge Fillet.1) is the current object.



A split feature (Split.1) is inserted just after EdgeFillet.1. This new feature absorbs EdgeFillet.1 and therefore the latter is no more displayed and cannot be referenced by any feature located after the Split.1 in the specification tree.



To ensure the ordering rule, the links to the absorbed feature (EdgeFillet.1) must be rerouted to the inserted feature (Split.1). This replacement applies to all the features inside the root ordered geometrical set (Ordered Geometrical Set.1) located after the inserted feature and to all the features located inside other ordered geometrical sets roots (here, Ordered Geometrical Set.2).

This replace action may not be applicable; in this case a warning message is issued.
Using our example, had we selected the other side of Split.1, the replacement of the edge to extrapolate (defined in Extrapol.1 feature) would not have been possible.

As a consequence of the replace action, the affected features (that is Extrapol.1 and Offset.1) become "not updated". The update following the insertion may also produce an error and in this case the design will have to be modified so that the inserted feature is compatible with the entire design.

The replace actions performed by the Delete command are generally the opposite of the replace actions performed by Insert command. Using our example, deleting Split.1 leads to the replacement of Split.1 by EdgeFillet.1. Nevertheless, bear in mind that deleting a feature can lead to a configuration different from the one preceding the insertion of a feature (for instance, if inserting a Trim feature, all inputs will be replaced by this feature but if deleting it, the Trim feature will be replaced by its main input).

Based on this mechanism stand two methodologies for:

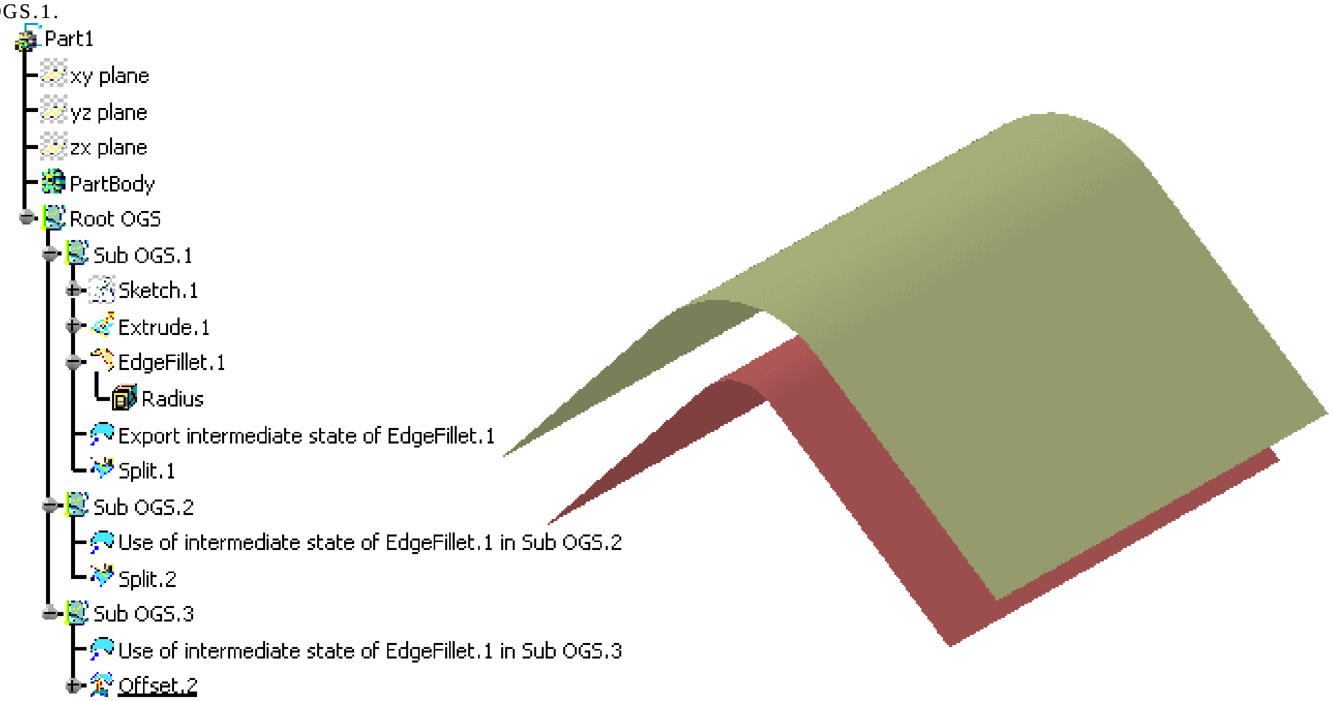
- multiple references to an intermediate state of design inside an ordered geometrical set,
- external links to the "end design" specified inside an ordered geometrical set.

Multiple references

Inside a root ordered geometrical set, a feature can be the input of several features (all creation features, except for the last feature, according to the order in the specification tree, which can be a modification feature). In some cases, the design may require to create several modification states of a same feature. To do so, it is necessary to create copies (Copy/Paste As Result With Link).

Open the OrderedGeometricalSets3.CATPart document.

This example shows how to allow multiple modifications of EdgeFillet.1 feature, considered as an "intermediate state of design". A copy of the feature, representing the exported view, is inserted just after it. In the beginning of every sub-set where this state of design will be used, a copy of the copy is created, representing the imported view.
Using this construction, modifications applied to EdgeFillet.1 or to the copies of the copy (imported views) will affect only the design in Sub OGS.1.

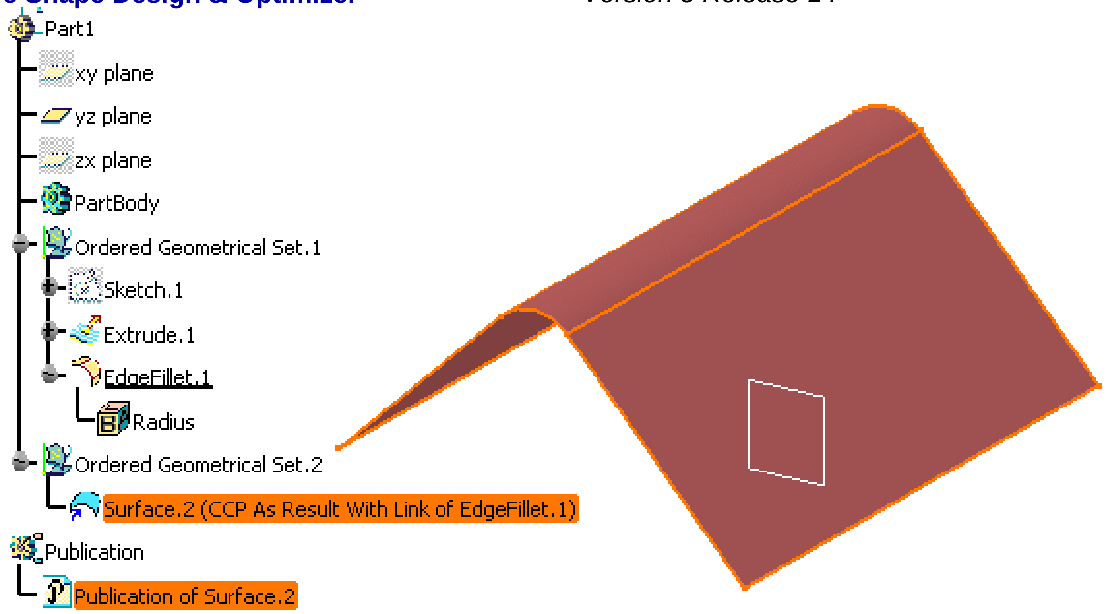


External Links

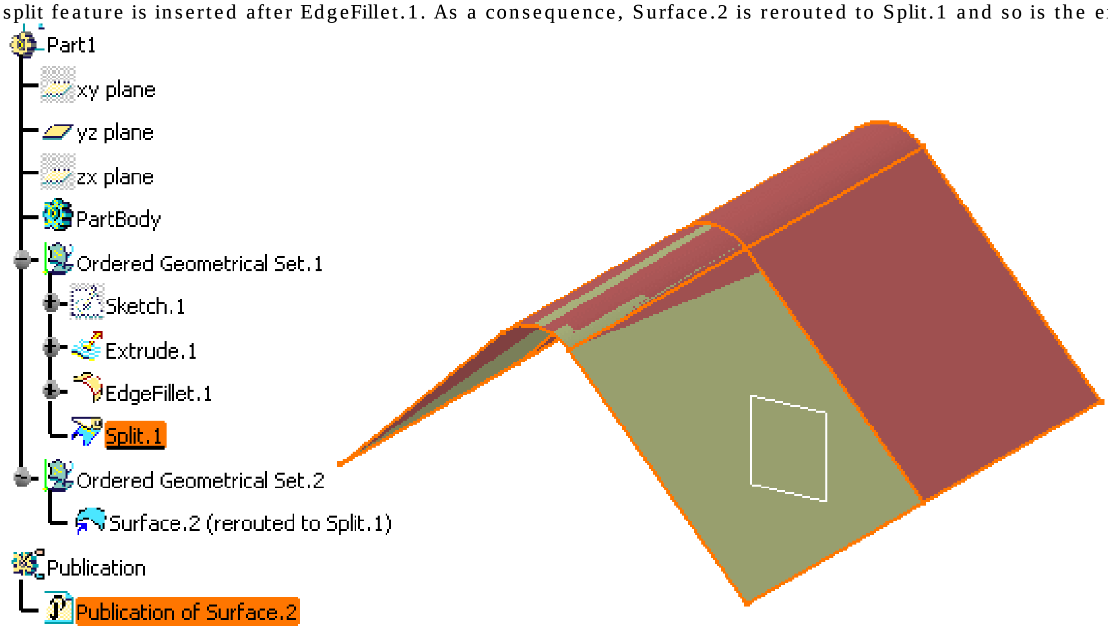
The replace actions due to design modifications (insertion and deletion) do not affect external links (that is the links between an external element from the .CATPart document and a feature inside an ordered geometrical set). To ensure that the links will always reference the last state of design, it is necessary to create a copy (Copy/Paste As Result With Link) of the last current feature in a new ordered geometrical set. This copy can possibly be published. As a consequence, the external link will have to reference this copy or its publication.

Open the OrderedGeometricalSets4.CATPart document.

In this example, Surface.2 is a copy of EdgeFillet.1.
The external link has to reference Surface.2 or its publication.



A split feature is inserted after EdgeFillet.1. As a consequence, Surface.2 is rerouted to Split.1 and so is the external link.



Inserting a Body into an Ordered Geometrical Set

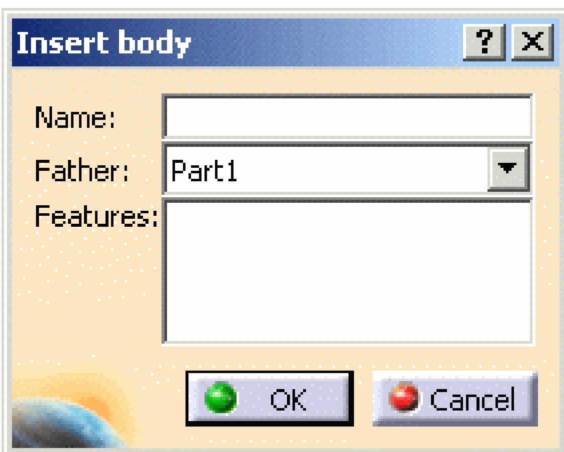
This task shows you how to insert a body into an ordered geometrical set.

Open the OrderedGeometricalSets1.CATPart document.

1. Select the **Insert -> Body in a Set...** command.

   The Insert body dialog box is displayed.

2. Enter the name of the body you wish to insert into the ordered geometrical set. Our part contains no bodies, so enter a name as you are creating the body. For example, enter New Body.



3. Use the Father drop-down list to choose the body where the new ordered geometrical set is to be inserted. In our example, set Ordered Geometrical Set.1. All destinations present in the document are listed allowing you to select one to be the father without scanning the specification tree. They can be:
   - ordered geometrical sets
   - parts

By default the destination is the father of the current object. By default the body is created after the current feature.

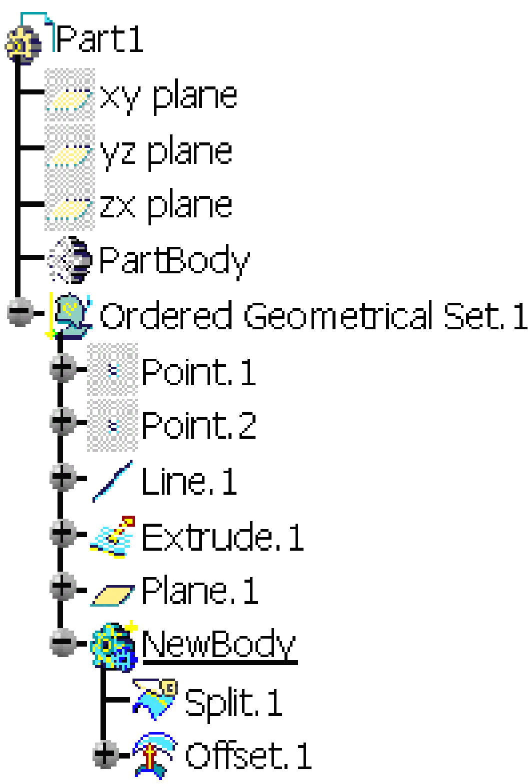
4. Set the Father option to the name of the ordered geometrical set you need. In our example, set Ordered Geometrical Set.1.

   Possible destinations are the part and all Ordered Geometrical Sets already defined in the part. By default the destination is the father of the current object. By default the body is created after the current feature.

5. It is possible to select elements of the Ordered Geometrical Set to put these elements inside the body when creating it. Only consecutive elements can be selected. Volumes and bodies cannot be selected. In case of selection of elements, the destination became automatically the father of the selected elements and cannot be changed any more. Select for example, Split.1 and Offset.1.

5. Click **OK** to confirm the operation.

   The result is immediate.



You can now create the features you need in the new body inserted into the Ordered Geometrical Set.

Duplicating Geometrical Sets and Ordered Geometrical Sets

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

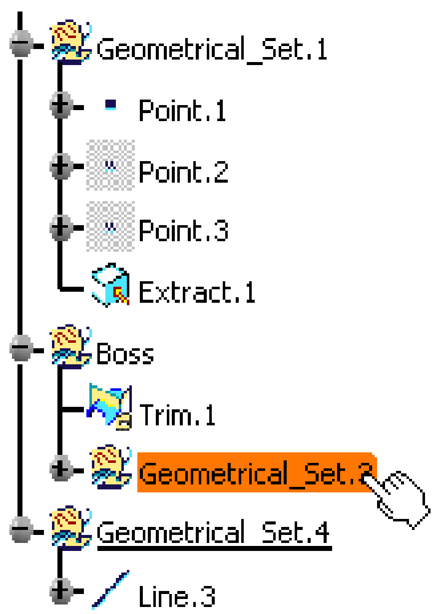
This task shows how to duplicate geometrical sets and ordered geometrical sets from the specification tree.

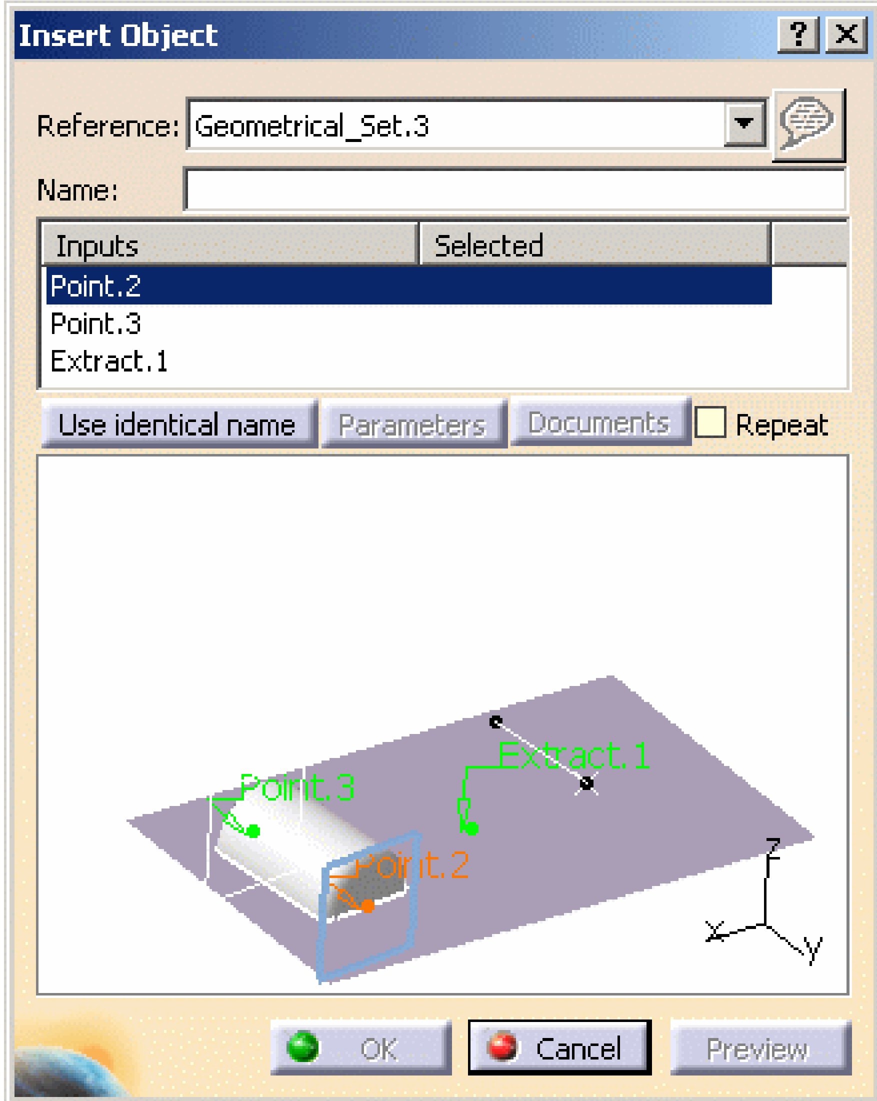
You will find other useful information in the Managing Geometrical Sets and the Managing Ordered Geometrical Sets sections.

Open the PowerCopyStart1.CATPart document.

1. Click the **Duplicate Geometrical Set** icon, or select the **Insert -> Advanced Replication Tools -> Duplicate Geometrical Set** menu item.

2. Select the geometrical set to be duplicated. It can be selected within the current .CATPart document, or from any other .CATPart document.



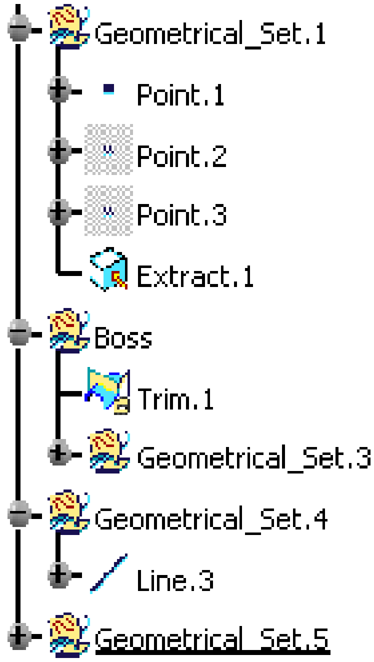


3. Complete the **Inputs** within the dialog box by selecting the adequate element in the geometric area.
4. If needed, click on the **Use identical name** button to automatically select all the elements with the same name. This is especially useful when the input is the same one repeated several time.

Check the **Repeat** option to be able to repeat the duplication.
In this case, once you have clicked OK in the Insert Object dialog box, the latter remains open, the Geometrical Set'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. Click OK.

   The new geometrical set is created in the specification tree.



The identifiers of copied elements are incremented with respect to the original elements.
The original elements and copied elements can be edited independently.

Hiding/Showing Geometrical Sets or Ordered Geometrical Sets and Their Contents

This task shows how to use the Hide/Show command on different level of geometrical sets and ordered geometrical sets and for different purposes. Indeed you can:

- hide/show a complete geometrical set or ordered geometrical set
- hide/show contents of a geometrical set or an ordered geometrical set
- hide/show an element while in a command
- hide/show an element belonging to an ordered geometrical set

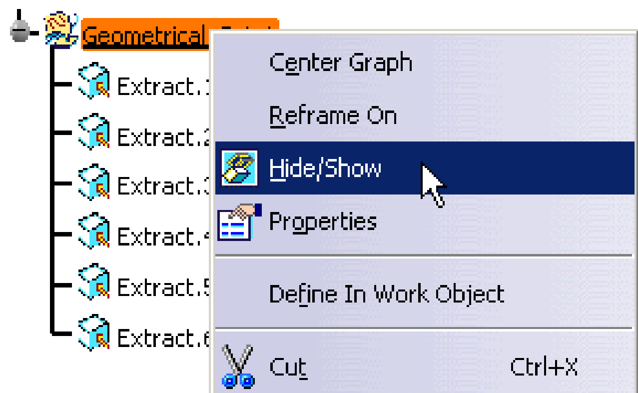
Open any .CATPart document containing Geometrical Sets or Ordered Geometrical Sets.
You can also open the GeometricalSets1.CATPart document to have an example with Geometrical Sets and the OrderedGeometricalSets1.CATPart document to have an example with Ordered Geometrical Sets.

Hiding/Showing a Geometrical Set or an Ordered Geometrical Set

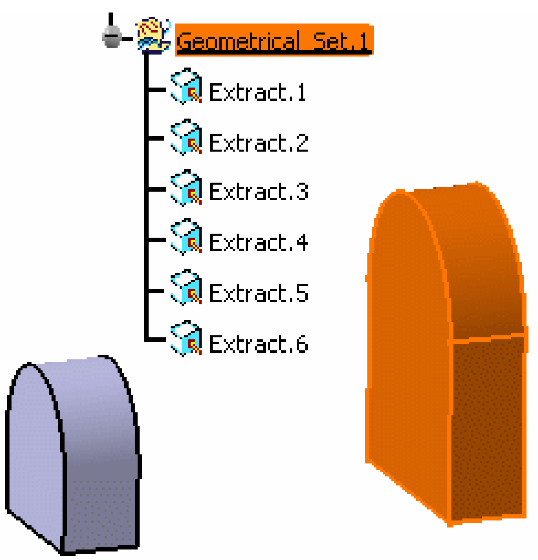
This contextual menu allows you to hide/show a geometrical set or an ordered geometrical set whether current or not.

1. In the specification tree, select the geometrical set or ordered geometrical set you wish to hide/show

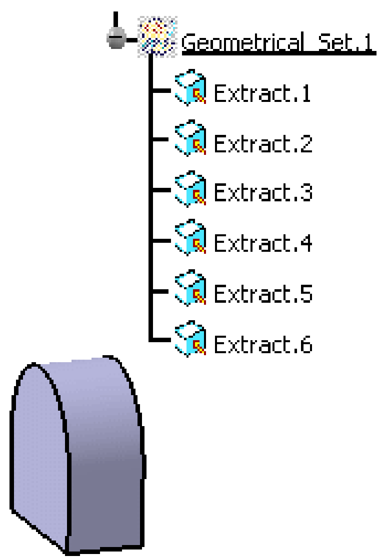
2. Right-click to display the contextual menu and choose the **Hide/show** command.



The geometrical set or ordered geometrical set is hidden, if it was visible, or becomes visible, if it was hidden.



Visible geometrical set



Hidden geometrical set

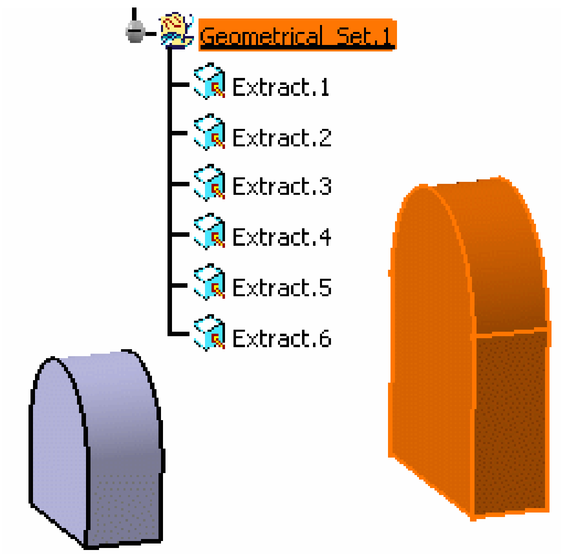
Hiding or Showing a geometrical set or an ordered geometrical set as a whole can also be done using the Hide/Show  icon.

It is not possible to hide a sub-ordered geometrical set using the **Hide/Show** contextual command. However you can use the **Hide components** contextual command as explained hereafter.

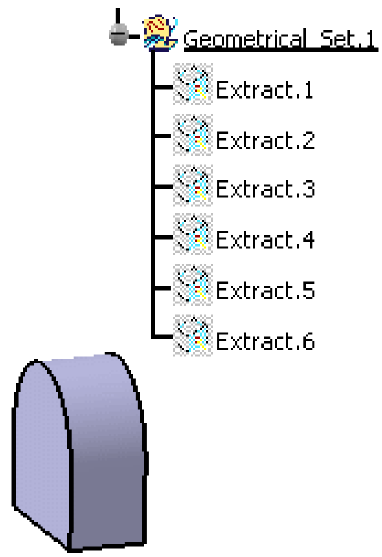
Hiding/Showing Contents of a Geometrical Set or Ordered Geometrical Set

This contextual menu allows you to hide/show all features in a geometrical set or an ordered geometrical set (even sketches), whether current or not.

1. In the specification tree, select the geometrical set or the ordered geometrical set whose solid elements you want to hide/show.
2. Right-click and choose **Geometrical_Set.x object -> Show components** contextual command to restore the view if the elements were hidden, or **Geometrical_Set.x object -> Hide components** contextual command to hide visible elements.



Visible contents



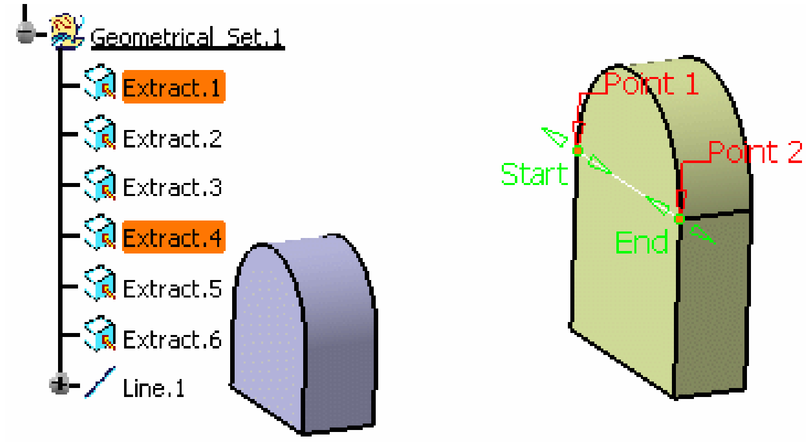
Hidden contents

It is advised to use this method to hide contents of a geometrical set or an ordered geometrical set, rather than using the **Hide/Show** contextual command: indeed when a geometrical set or an ordered geometrical set is in show, its contents are as well. This method enables to quickly show an element of a geometrical set or an ordered geometrical set .

Hiding/Showing an element while in a command

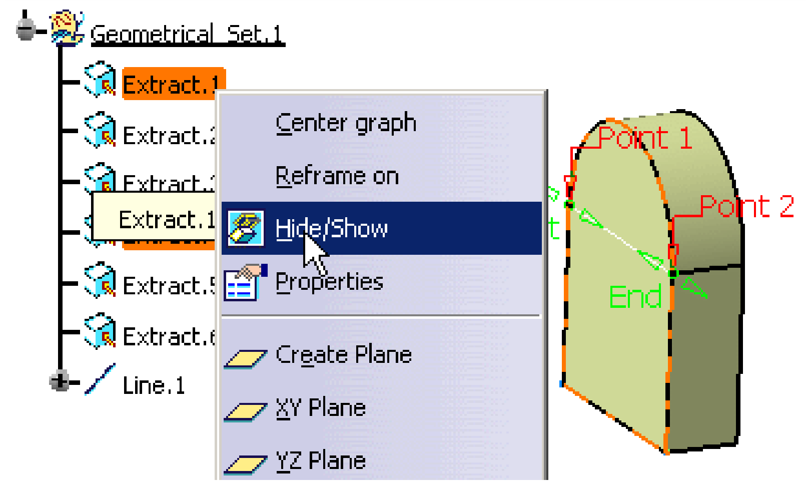
This contextual menu allows you to hide/show an element of the current geometrical set or ordered geometrical set, while using a command.

1. Click the **Line** icon and select two points to create a line.

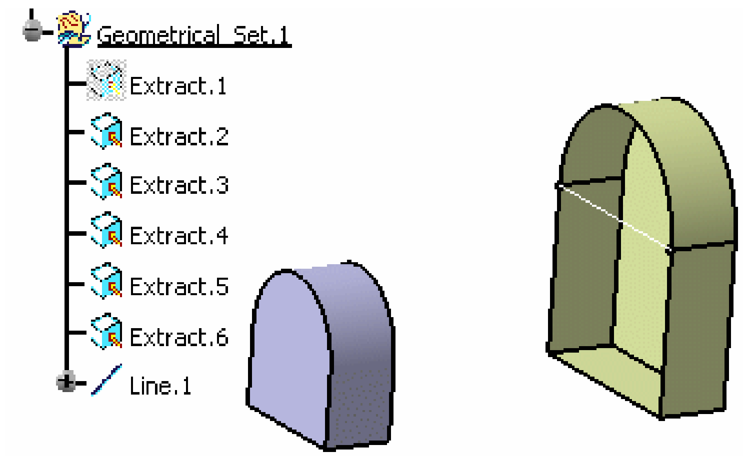


2. Right-click the element to be hidden from the specification tree or the geometry, and choose the **Hide/Show** contextual command.

   The selected element is hidden without exiting the currently active command.



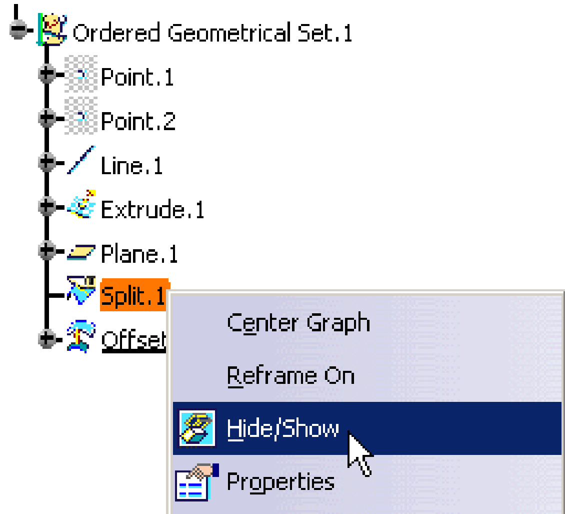
3. Click OK in the Line dialog box to create the line.
4. Repeat the operation on the element again to re-display it.



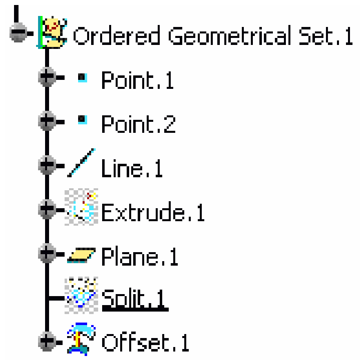
Hiding/Showing an element belonging to an Ordered Geometrical Set

This contextual menu allows you to hide/show a modification feature.
If a modification feature is put in no show, all features absorbed by this feature are in no show too.

1. Right-click the element (Split.1) to be hidden from the specification tree or the geometry, and choose the **Hide/Show** contextual command.



As Split.1 is absorbed by Extrude.1, Extrude.1 is also put in no show.



Note that you can hide/show all elements of a document, according to their type. To do this, simply use the **Tools -> Show** or **Tools -> Hide** menu and choose the adequate element type (**All Points**, **All Lines**, **All Curves**, **All Sketches**, **All Surfaces**, **All Planes**, **All Geometrical Sets**, **All Bodies**, **All Axis Systems**, **All Elements**, **All Selected Elements**, **All Except Selected Elements**).

Creating a Curve From Its Equation

This task shows how to create a curve by defining its equation as a law.

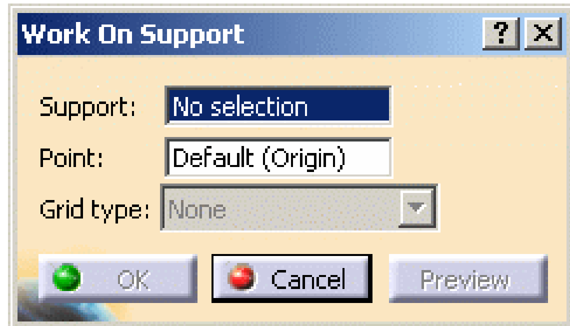
You must have access to the *Knowledge Advisor* product.

Make sure the **Relations** and **Parameters** options are active in the **Tools -> Options -> Infrastructure -> Part Infrastructure -> Display** tab.

Open a new .CATPart document.

1. In the Generative Shape Design workbench, define a working support using the **Work on Support** icon .

The Work on Support dialog box appears.



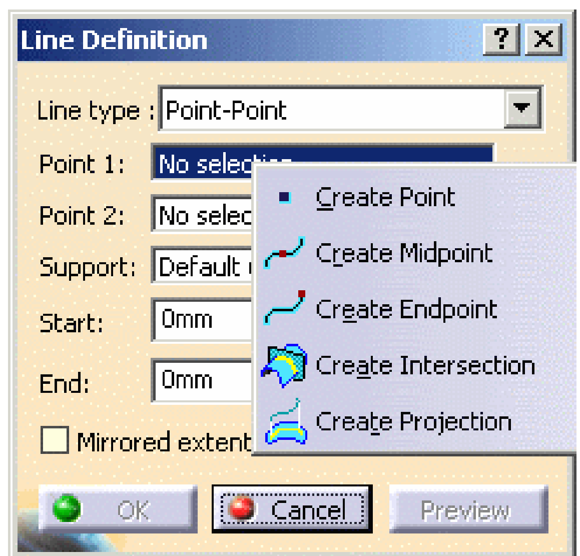
2. Select the yz plane, for example, and click OK in the updated Work on Support dialog box without modifying any other parameter.

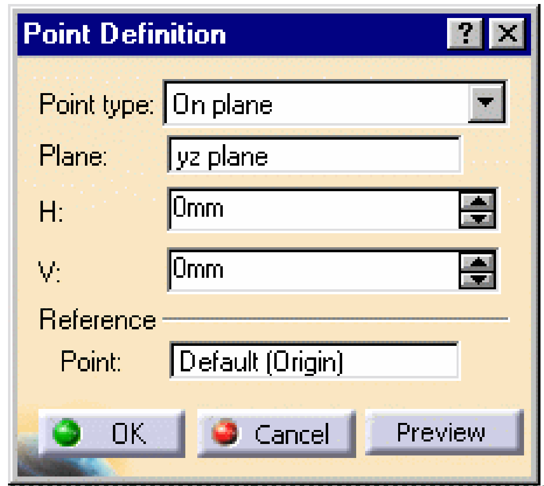
The Working Support.1 is created, and the system automatically moves into this plane.
You now want to create a horizontal line as the abscissa axis.

3. Click the **Line** icon .

The Line dialog box is displayed.

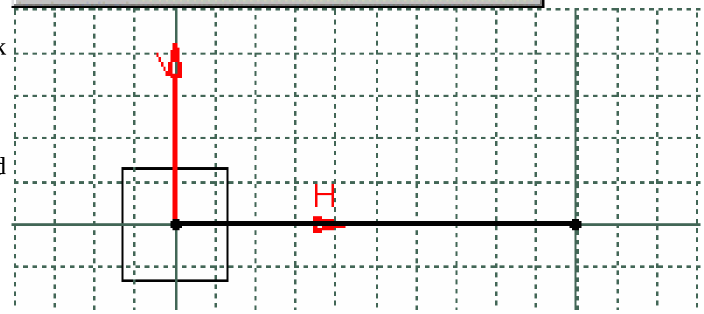
4. Right-click in the Point 1 field, and choose the Create point contextual menu.





The Point Definition dialog box is displayed, the **Point type** and **Plane** fields being automatically filled.

5. Create a point at **H:0mm** and **V:0mm**, and click OK.



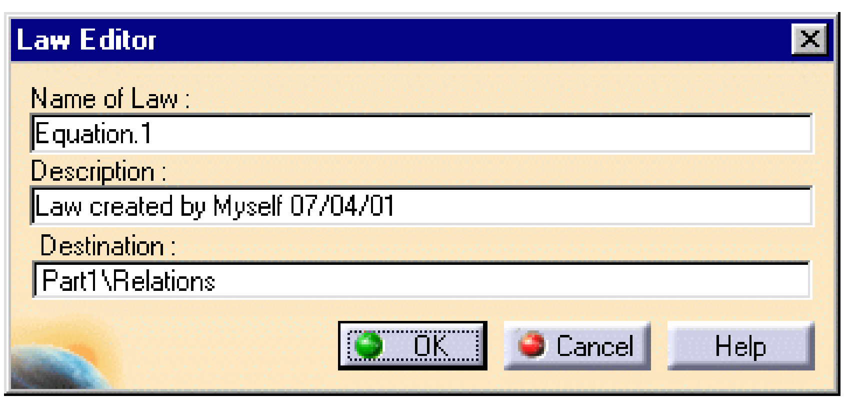
6. Repeat the operation, right-click the Point 2 field from the Line dialog box to create another point at **H:100mm** and **V:0mm**, then click OK in the Point Definition dialog box.

7. Click OK in the Line dialog box to create the line

You may want to hide the grid by checking the Hide grid option from the Work On Support dialog box.

8. From the Knowledge toolbar, click the **Law**  icon.

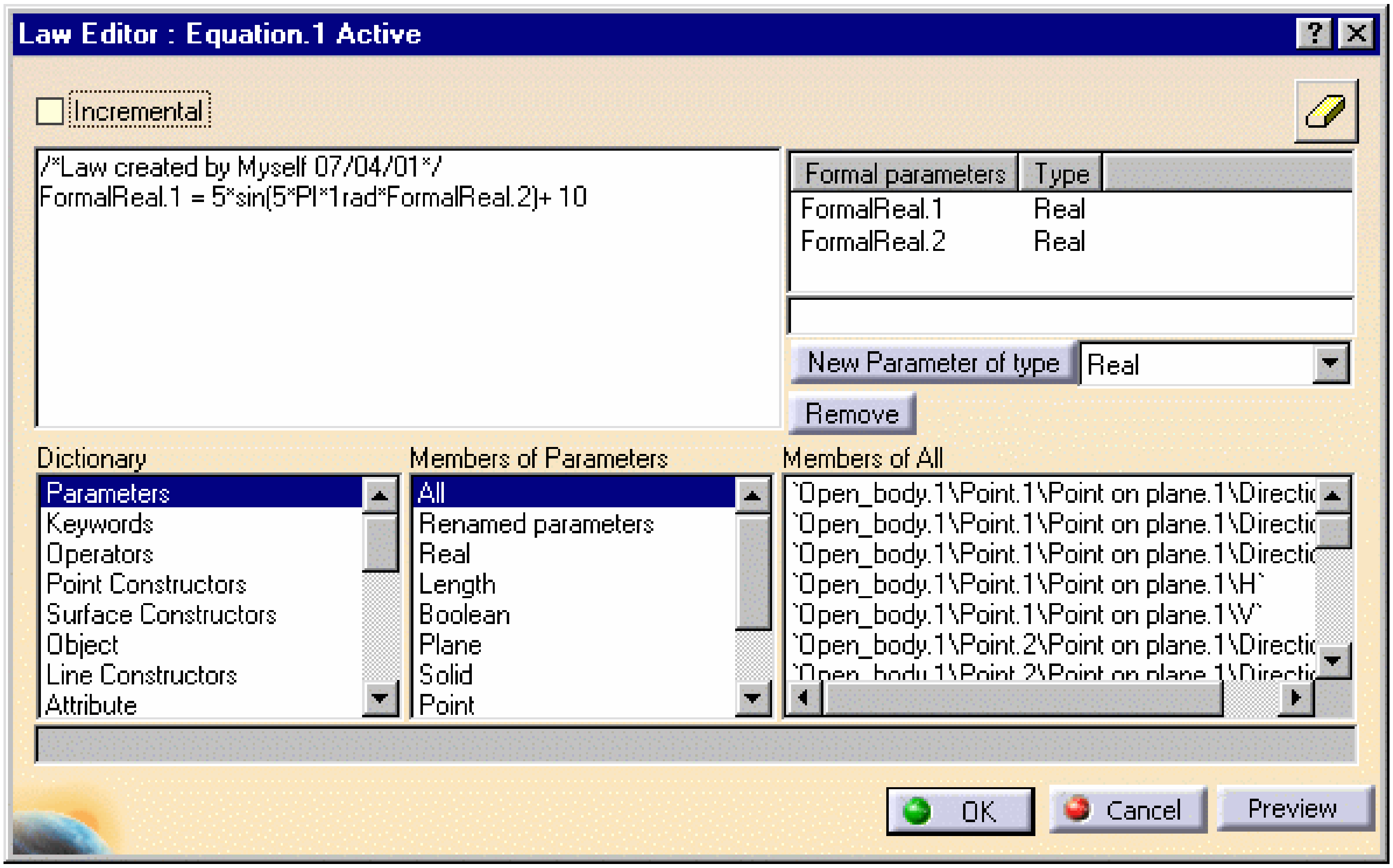
The Law Editor dialog box is displayed in which you name the law to be created, give it a description and a storage location.



9. Click OK.

The Law Editor dialog box is updated. The right-hand part allows you to create the parameters to be used in the law. The left-hand part is the law edition box.

10. Create two real type parameters FormalReal.1 and FormalReal.2, then enter the law below into the edition window: `FormalReal.1 = 5*sin(5*PI*1rad*FormalReal.2)+ 10`



11. Click OK to create the law.

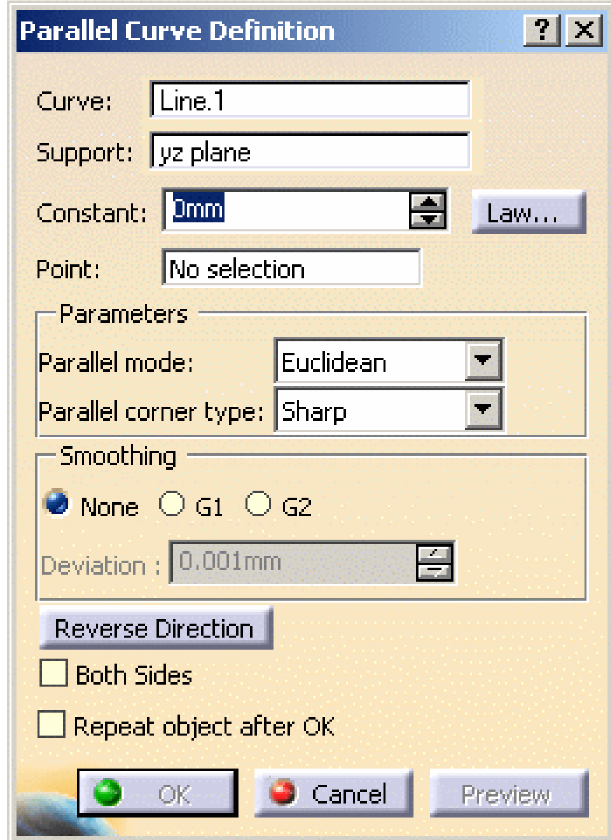
12. In the Generative Shape Design workbench, click the **Parallel Curve** icon .

The Parallel Curve Definition dialog box is displayed.

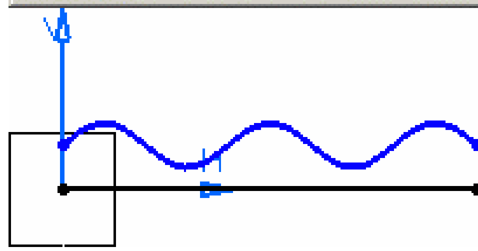
13. Select the line created in Step 7 as the reference **Curve**.

14. Click the **Law** button and select the Law.1 you have juste created from the specification tree.

15. Click OK.



A curve parallel to the selected one is created, taking the law into account, i.e. it is defined by the equation entered as a law using the Knowledge Advisor.



Creating a Parameterized Curve

This task shows how to create a planar curve defined by two formulas.

You must have access to the *Knowledge Advisor* product.

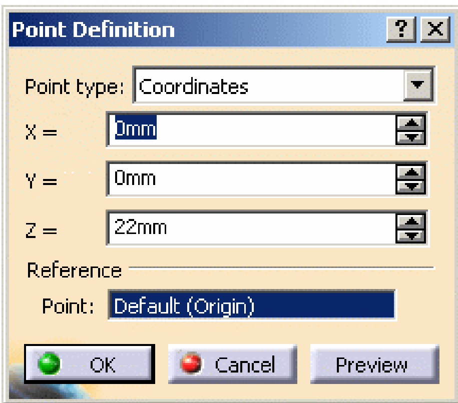
Make sure the **Relations** and **Parameters** options are active in the **Tools -> Options -> Infrastructure -> Part Infrastructure -> Display** tab.

Open a new .CATPart document.

1. Create two points using the following coordinates:

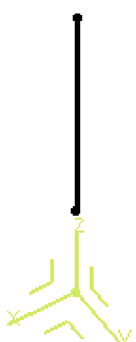
   Point.1: X=0, Y=0, and Z=22

   Point.2: X=0, Y=0, and Z=75



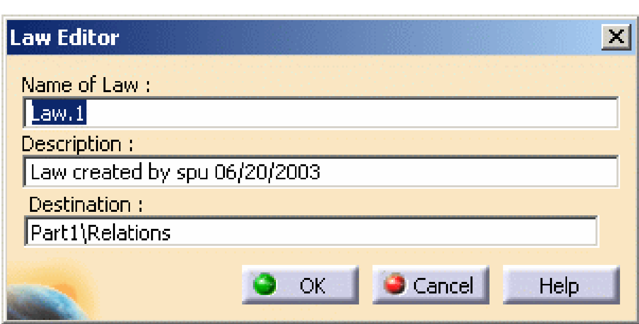
2. Define line between these two points.

   It is created along the the Z axis of the axis-system, using the Point-Point option.



3. From the Knowledge toolbar, click the **Law** icon.

The Law Editor dialog box is displayed in which you name the law to be created, give it a description and a storage location.

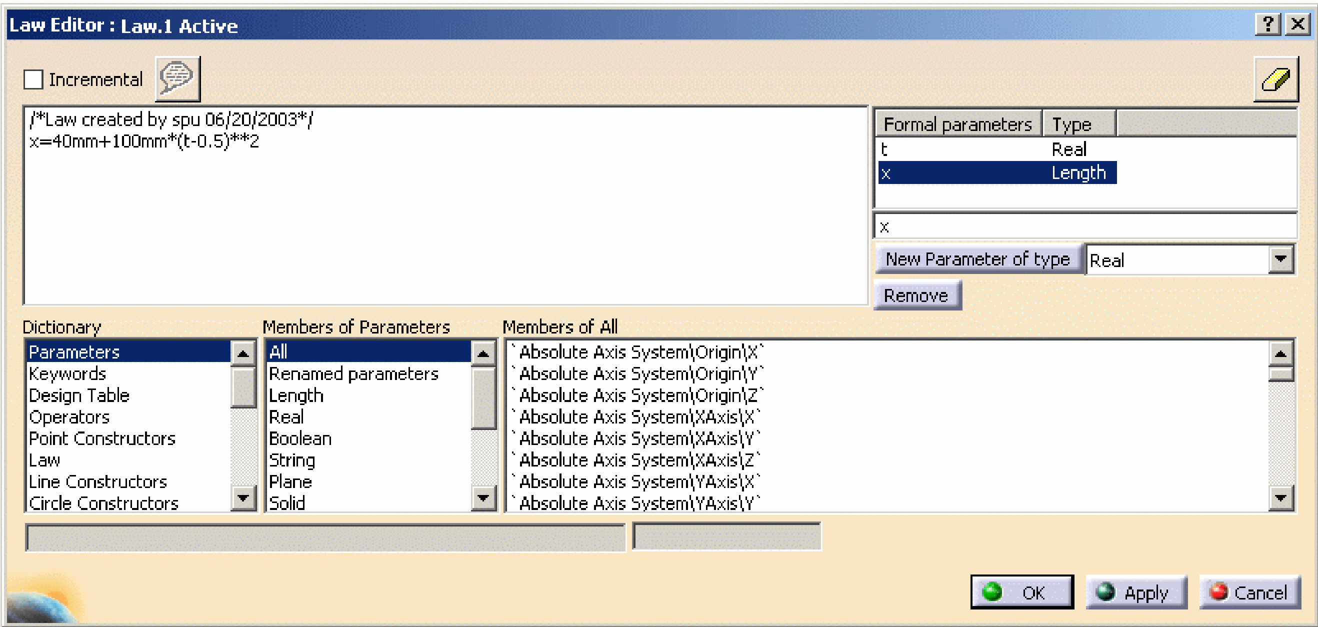


4. Click OK.

The Law Editor dialog box is updated. The right-hand part allows you to create the parameters to be used in the law. The left-hand part is the law edition box.

5. Create one real type parameter t and one length type parameter x, then enter the law below into the edition window:

   `x=40mm+100mm*(t-0.5)**2`



6. Click OK to create the law.

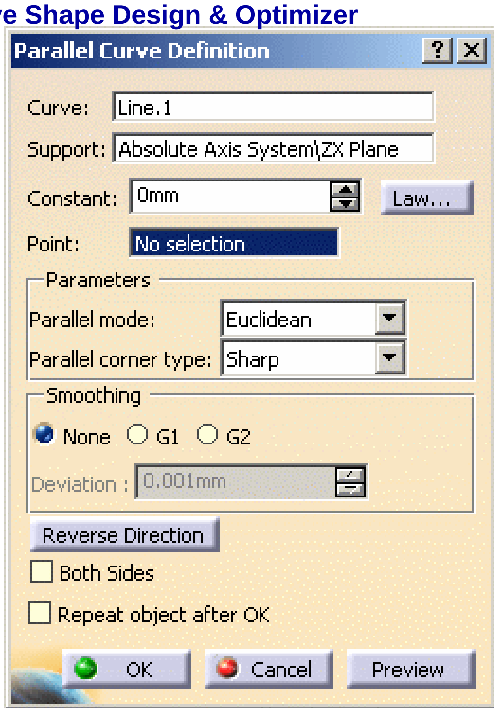
7. Create a second law using one real type parameter t and one length type parameter y, then entering the following law into the edition window: `y=40mm+200mm*(t-0.5)*(0.25-t)*(0.75-t)`

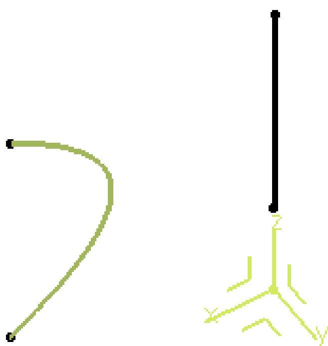
8. Create a parallel curve, using the Line.1 created in step 2 as the reference **Curve** to be offset, and the ZX plane of the axis system as the **Support** on which the reference curve lies.

9. Click the **Law...** button and select the Law.1 you have just created from the specification tree.

10. Click OK.

A curve parallel to the selected one is created, taking the law into account, i.e. it is defined by the equation entered as a law using the Knowledge Advisor.

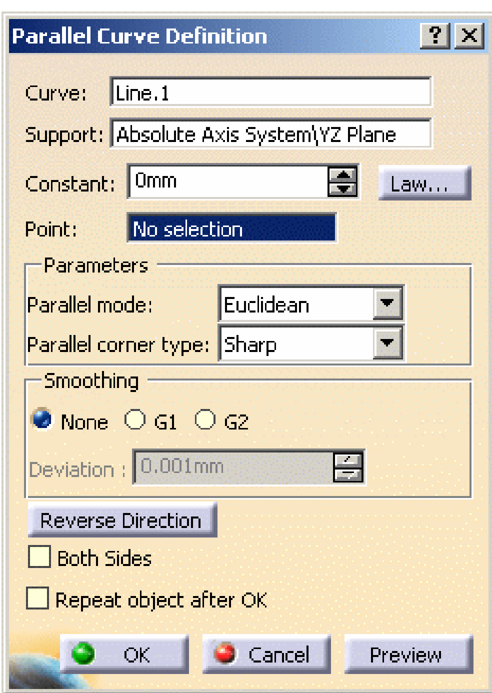


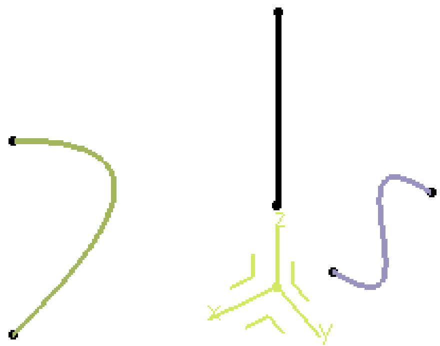


11. Create a second parallel curve, using Line.1 as the reference **Curve** and the YZ plane as the **Support**.

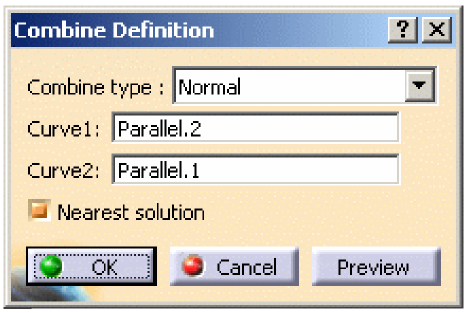
12. Click the **Law...** button and select Law.2.

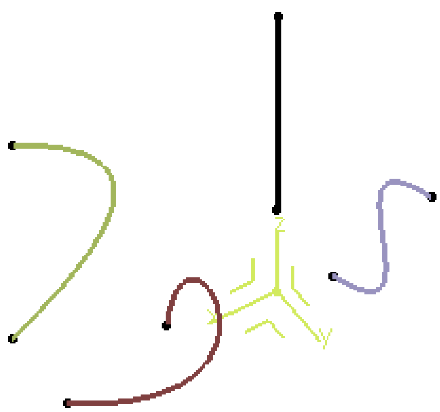
13. Click OK to create the parallel curve with a variable law.





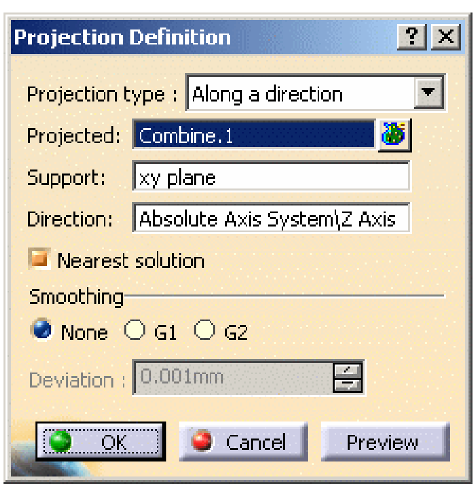
14. Use the Combine command to create a curve resulting from the intersection of the extrusion of the two parallel curves.

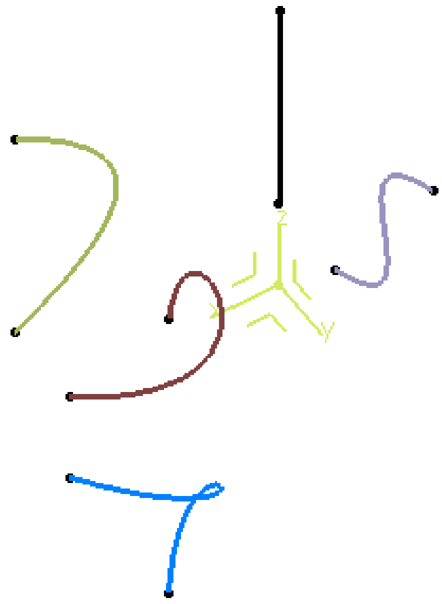




15. Use the Project command to project the combined curve onto the xy plane:

Choose the combined curve as the element to be projected, the xy plane as the Support, and the Z axis as the Direction.

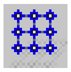


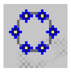


Here is the parameterized curve (in blue) obtained by two formulas:

It is not advised to create closed parameterized curves.

Patterning

 Create rectangular patterns: select the element to be duplicated, define the creation directions, choose the parameters you wish to define and set these parameters

 Create circular patterns: select the element to be duplicated, define the axial reference, the creation direction, choose the parameters you wish to define and set these parameters

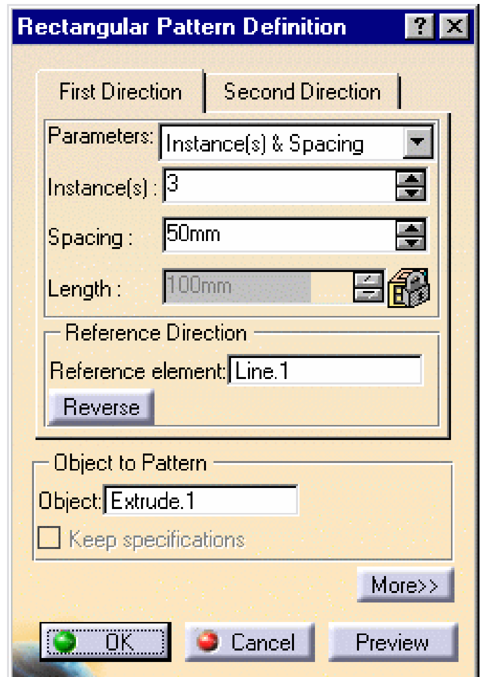
Creating Rectangular Patterns

This task shows how to use create rectangular patterns, that is to duplicate an original wireframe or surface-type element at the location of your choice according to a rectangular arrangement.
This means that you will need to define a 2-axis system using two directions.

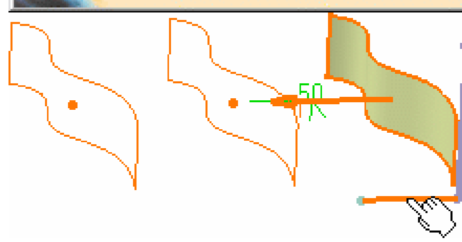
Open the Pattern1.CATPart document.

1. Click the **Rectangular Pattern** icon.

2. Select the element you wish to replicate as a pattern.



The Rectangular Pattern Definition dialog box is displayed. Each tab is dedicated to a direction you will use to define the location of the duplicated element.



3. Click the **Reference element** field and select a direction to specify the first direction of creation.

To define a direction, you may select a line, a planar face or surface edge.
You can reverse this direction by clicking the Reverse button.

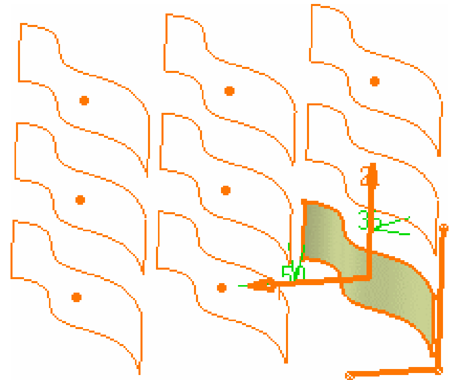
4. Set the duplication parameters by choosing the number of instances, the spacing between instances, or the total length of the zone filled with instances.

   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.

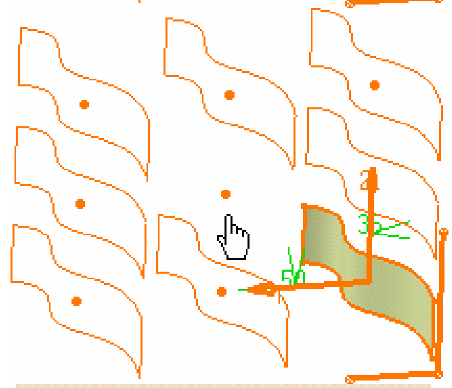
If you set **Instances & Length** or **Spacing & Length** parameters, note that you cannot define the length by using formulas.



5. Click the **Second Direction** tab to define the same parameters along the other direction of the rectangle.

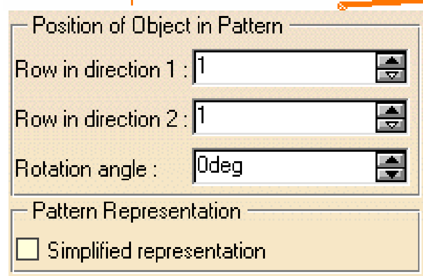
You can delete instances of your choice when creating or editing a pattern. To do so, just select the points materializing instances in the pattern preview.

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.



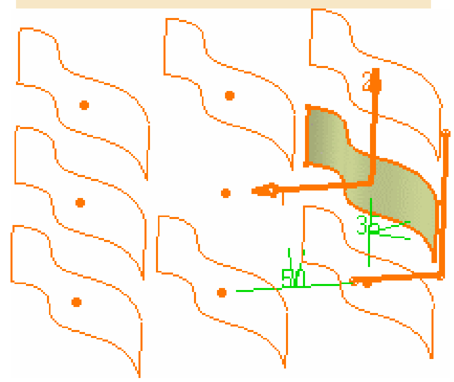
6. Click the **More>>** button to display further options.

   These options let you position the instances in relation to the first selected element.

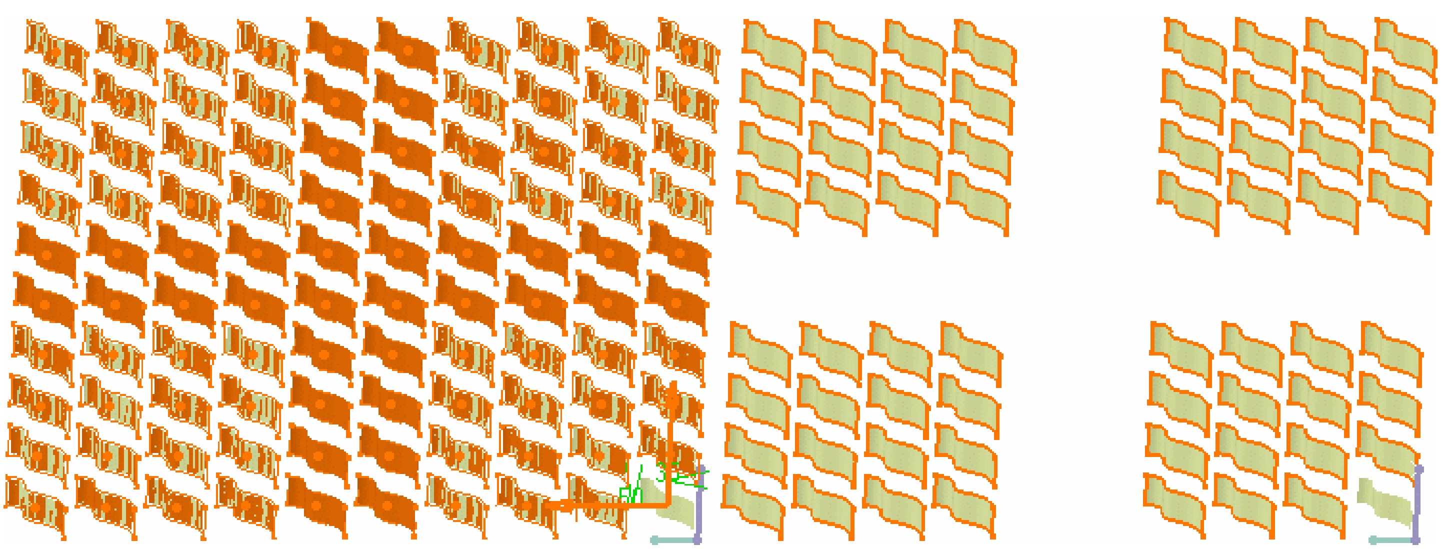


7. Increase the Row in direction 2 to 2.

   You notice that the first selected pattern now is the second instance in the vertical direction, as this was the second selected direction.



The **Simplified representation** option lets you lighten the pattern geometry, when more than 15 instances are generated. What you need to do is just check the option, and click Preview. The system automatically simplifies the geometry:

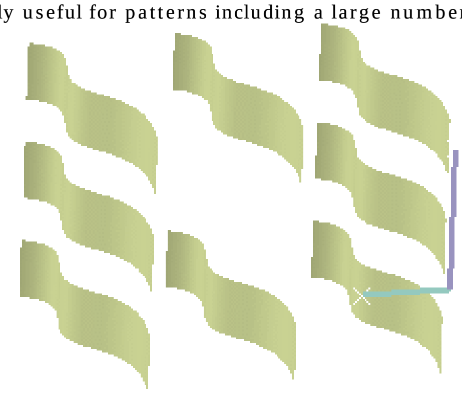


Previewed simplified geometry *Simplified geometry*

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.

8. Click OK to create the pattern.

The pattern (identified as RectPattern.xxx) is added to the specification tree.

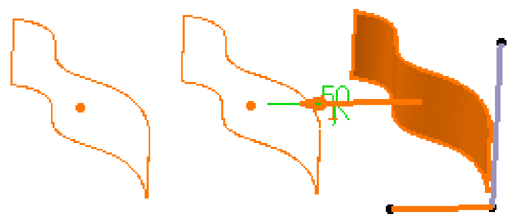


Patterning Volumes

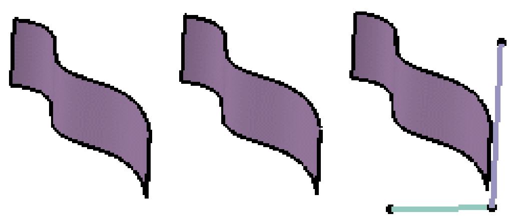
Open the PatterningVolumes1.CATPart document.

1. Click the **Rectangular Pattern** icon.
2. Select the element you wish to replicate as a pattern.

   The Rectangular Pattern Definition dialog box is displayed.
3. Click the **Reference element** field and select a direction to specify the first direction of creation.
4. Set the duplication parameters by choosing the number of instances, the spacing between instances, or the total length of the zone filled with instances.



5. Click OK to create the pattern.

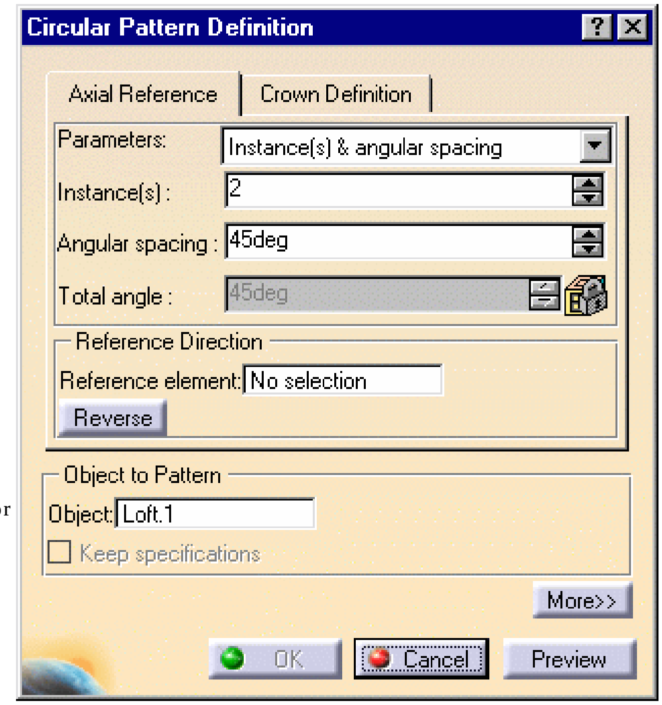


Creating Circular Patterns

This task shows how to use create circular patterns, that is to duplicate an original wireframe or surface-type element at the location of your choice according to a circular arrangement.

Open the Pattern2.CATPart document.

1. Click the **Circular Pattern** icon.



2. Select the element to replicate as a pattern.

   Here we selected the multi-sections surface.

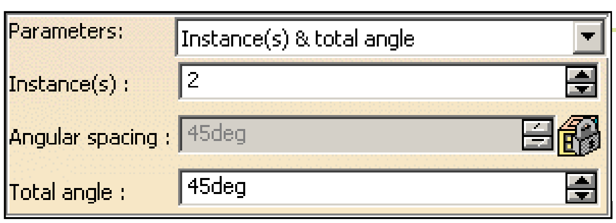
The Circular Pattern Definition dialog box is displayed.

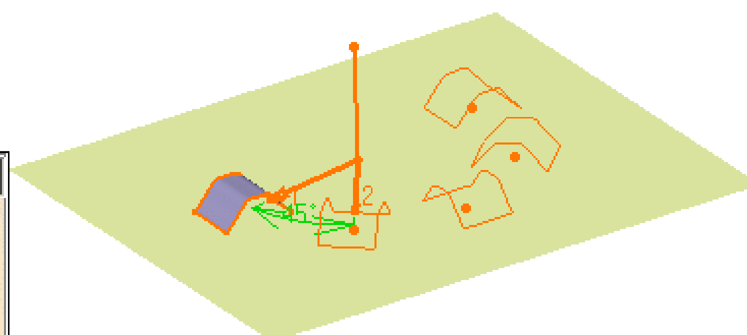
3. Click the **Reference element** field and select a direction to specify the first direction of creation, that is the rotation axis (Line.2).

- To define a direction, you can select a line, an edge or a planar face.
  Should you select a face, the rotation axis would be normal to that face.
- You can click the Reverse button to inverse the rotation direction.

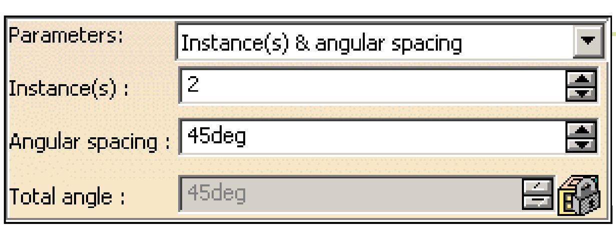
4. Define the **Axial Reference** by choosing the **Parameters** type:

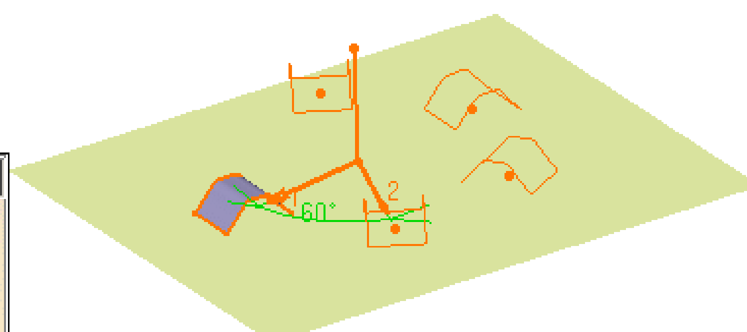
- **Instance(s) & total angle**: the number of patterns as specified in the instances field are created, in the specified direction, and evenly spread out over the total angle.



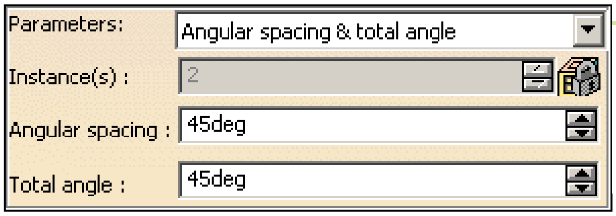


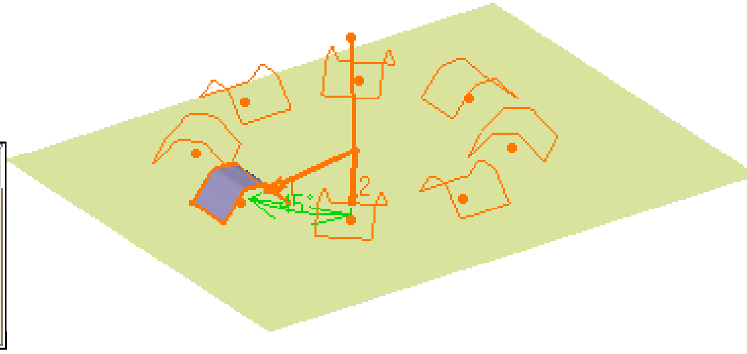
- **Instance(s) & angular spacing**: the number of patterns as specified in the instances field are created in the specified direction, each separated from the previous/next one of the angular angle value.



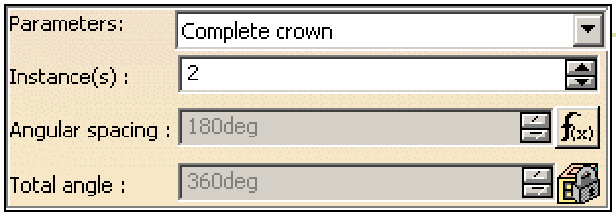


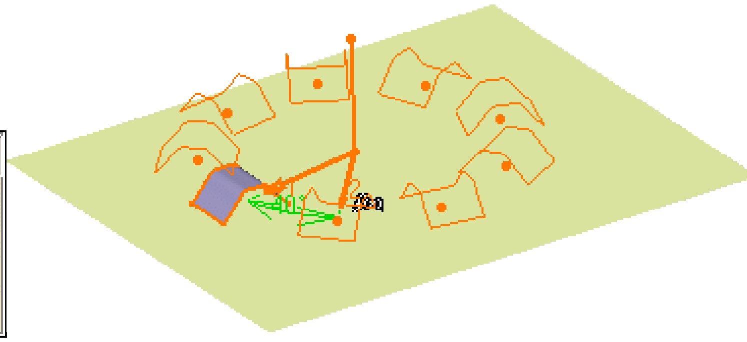
- **Angular spacing & total angle**: as many patterns as possible are created over the total angle, each separated from the previous/next one of the angular angle value.





- **Complete crown**: the number of patterns as specified in the instances field are created over the complete circle (360°deg).

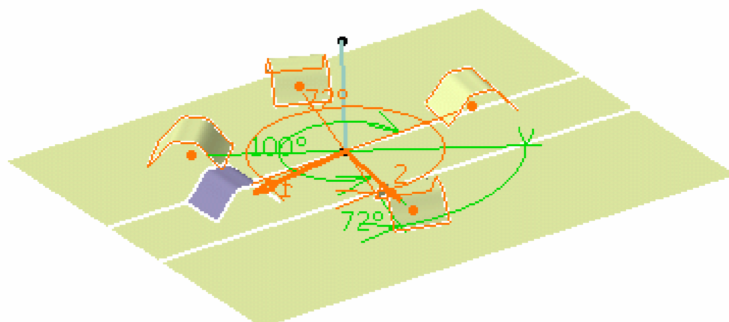


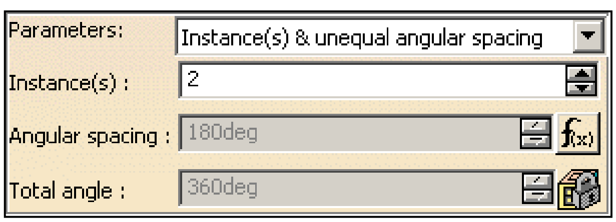


- **Instances & unequal angular spacing**: the number of patterns as specified in the instances field are created using a specific angular spacing between each instance.
  Angular spacing values are displayed between each instance.

To edit the values between each instance, you need to edit them individually. Select the angular spacing of interest, then choose one of the methods described hereafter: For instance, if you wish to change 72 degree for 100 degree for the angular spacing selected as shown in our picture, you can:

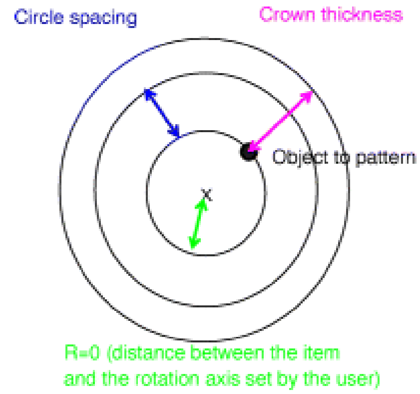
a. double-click the angle value in the 3D geometry. This displays the Parameter Definition dialog box in which you can enter the new value.
b. directly enter the new value in the Angular spacing field of the Circular Pattern Definition dialog box.





If you set **Instance(s) & total angle** or **Angular spacing & total angle** parameters, note that you cannot define the length by using formulas.

Defining a circular pattern



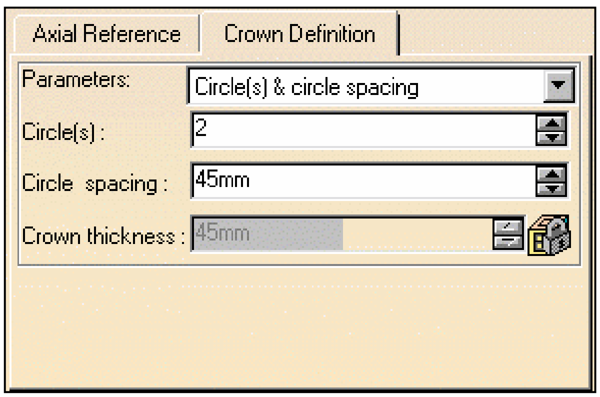
Now you are going to add a crown to this pattern.

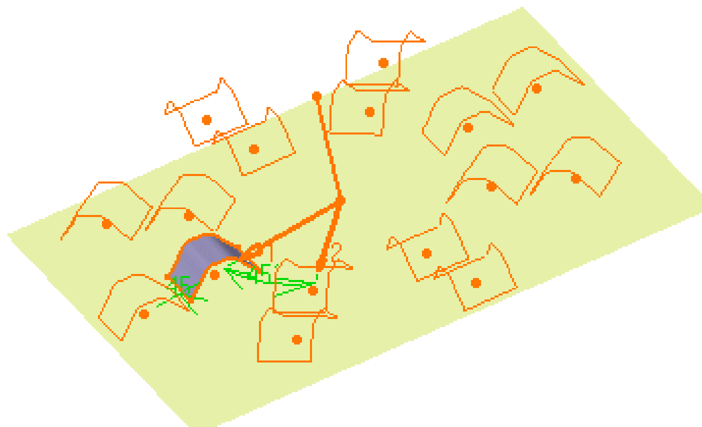
5. Click the **Crown Definition** tab, and choose which parameters you wish to define the crown.

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:

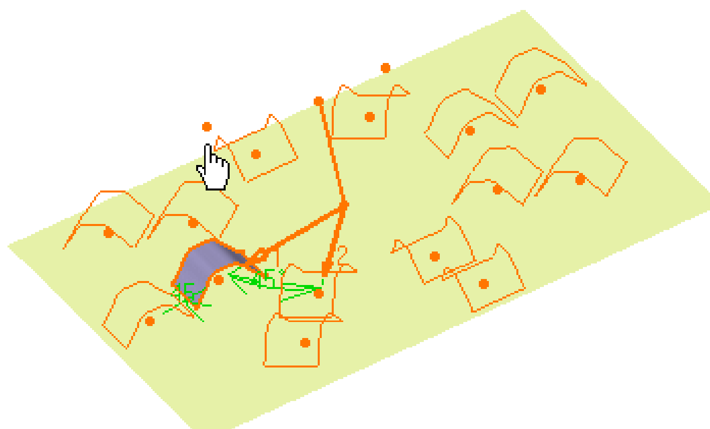




Note that a few patterns are created beyond the surface.

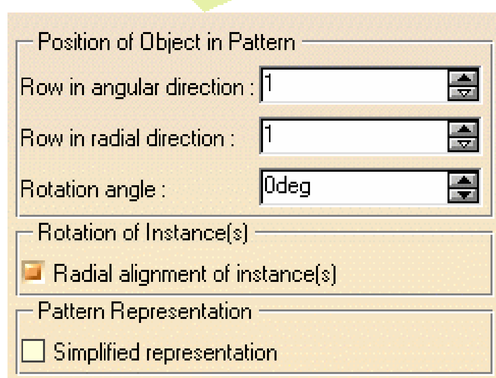
You can delete instances of your choice when creating or editing a pattern. To do so, just select the points materializing instances in the pattern preview.

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.

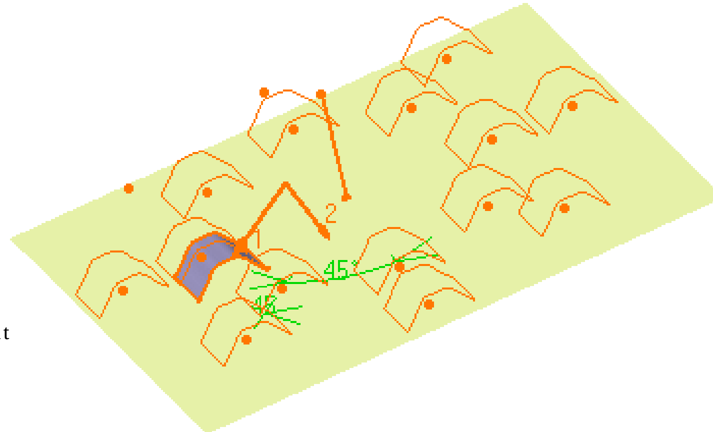


6. Click the More>> button to display further options:

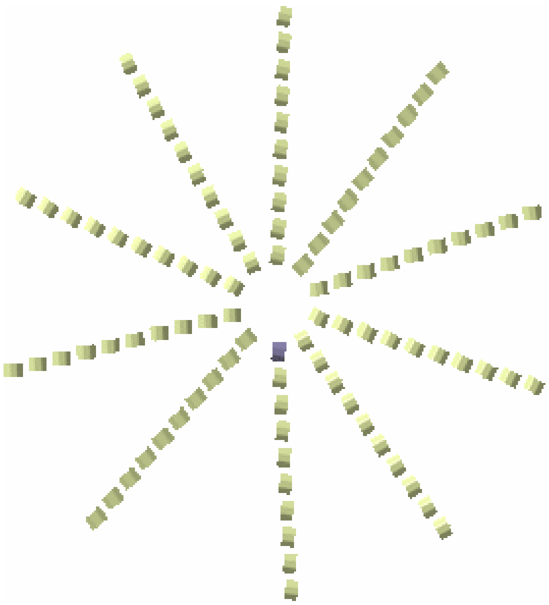
These options let you position the instances in relation to the first selected element.



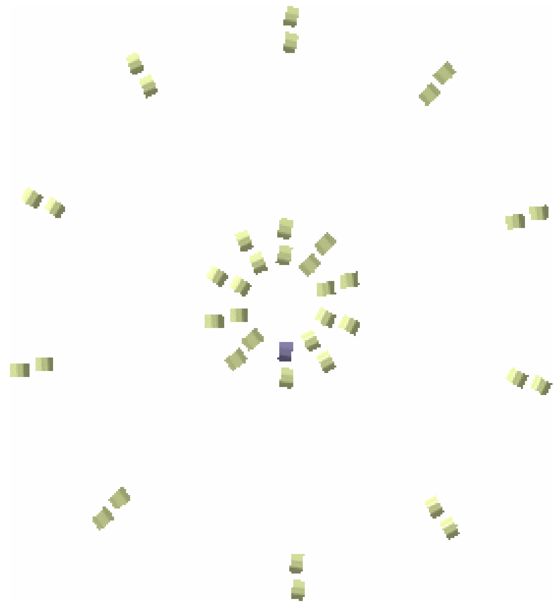
Using these options, you can change the position of the selected element within the crown. For example, if you set the **Rotation angle** parameter to 30° and you uncheck the **Radial alignment of instance(s)** option, this is what you obtain: the initially selected element has moved 30° from its initial location, based on the rotation direction, and all instances are normal to the lines tangent to the circle.



- The **Simplified representation** option lets you lighten the pattern geometry, when more than 15 instances are generated. What you need to do is just check the option, and click Preview. The system automatically simplifies the geometry:



Not simplified geometry



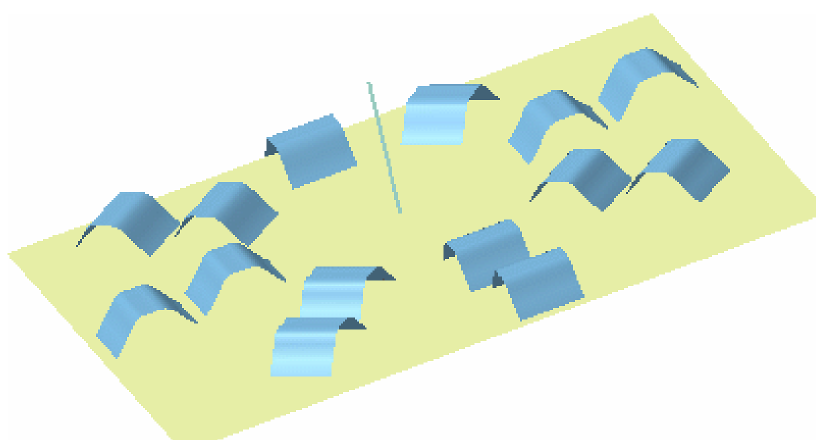
Simplified geometry

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.

   The pattern (identified as CircPattern.xxx) is added to the specification tree.

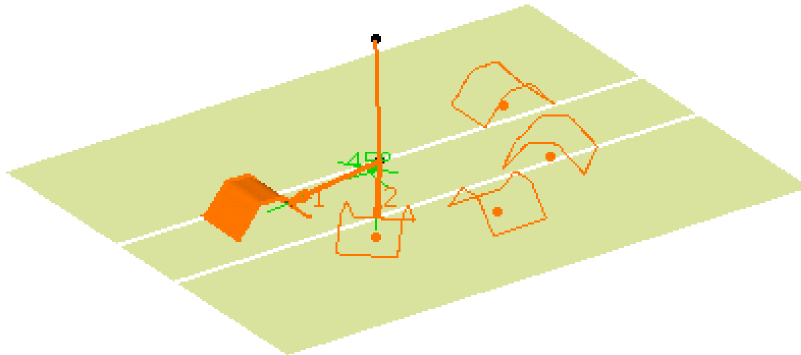


Patterning Volumes

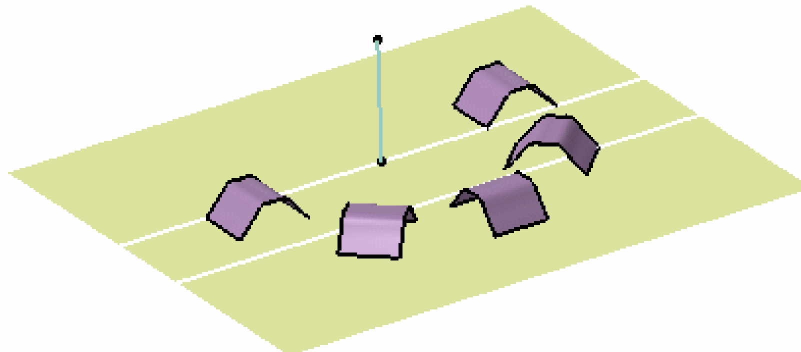
Open the PatterningVolumes2.CATPart document.

1. Click the **Circular Pattern** icon.
2. Select the element you wish to replicate as a pattern.

   The Circular Pattern Definition dialog box is displayed.



3. Click the **Reference element** field and select a direction to specify the first direction of creation.
4. Define the **Axial Reference** by choosing the **Parameters** type:



5. Click OK to create the pattern.

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.

Creating PowerCopies

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.

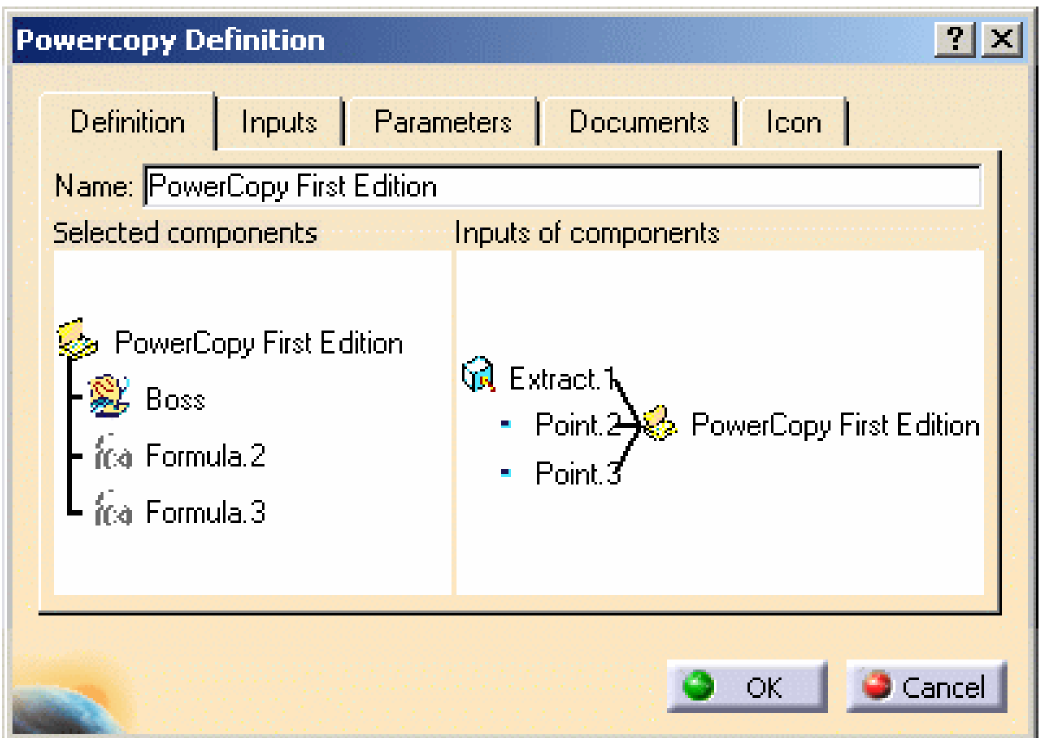
Open the PowerCopyStart1.CATPart document.

1. Click the **PowerCopy Creation** icon, or select the **Insert -> Knowledge Templates -> Power Copy...** menu item.

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

2. Select, from the specification tree, the elements to be included in the PowerCopy.

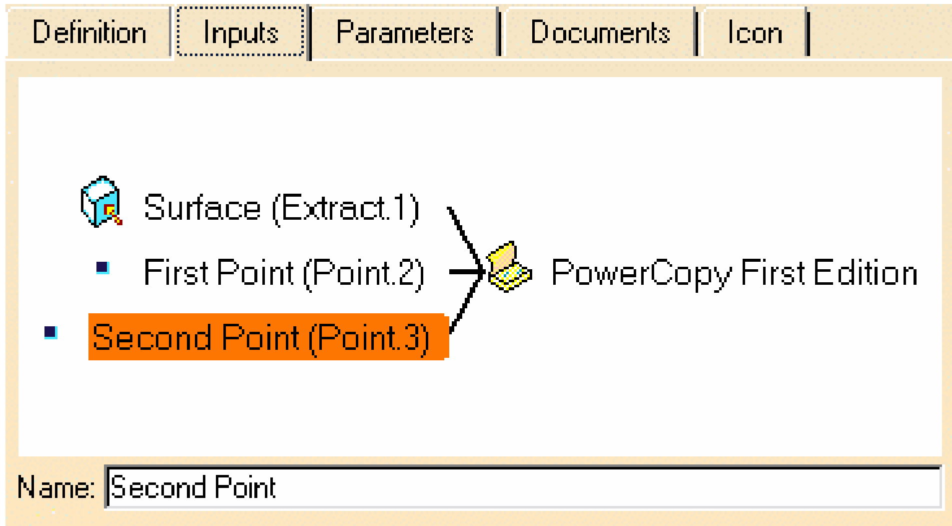
The **PowerCopy Definition** dialog box is automatically filled with information about the selected elements.



3. Define the PowerCopy as you wish to create it:

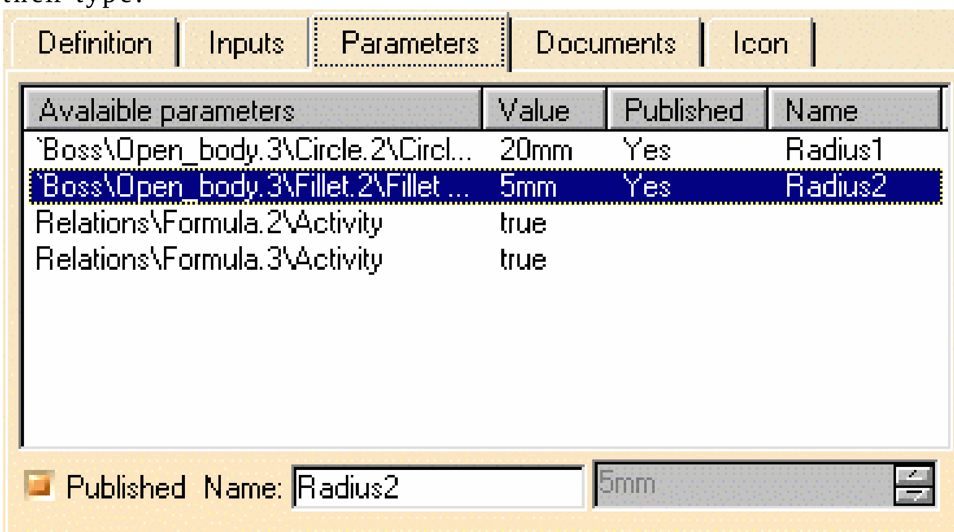
The **Definition** tab lets you assign a name to the PowerCopy and presents its components in the 3D viewer.

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.

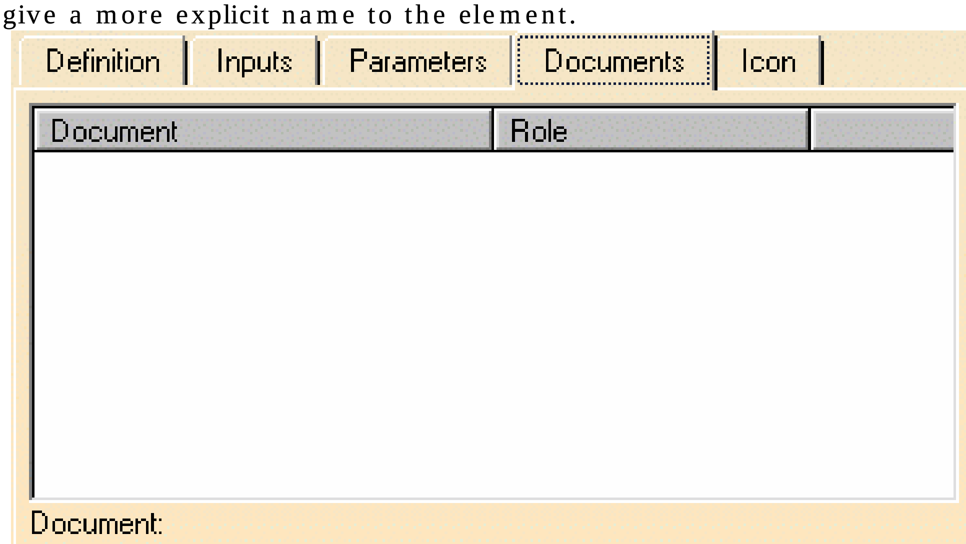
The **Parameters** tab lets you define which of the parameter values used in the PowerCopy you will be able to modify at instantiation time.



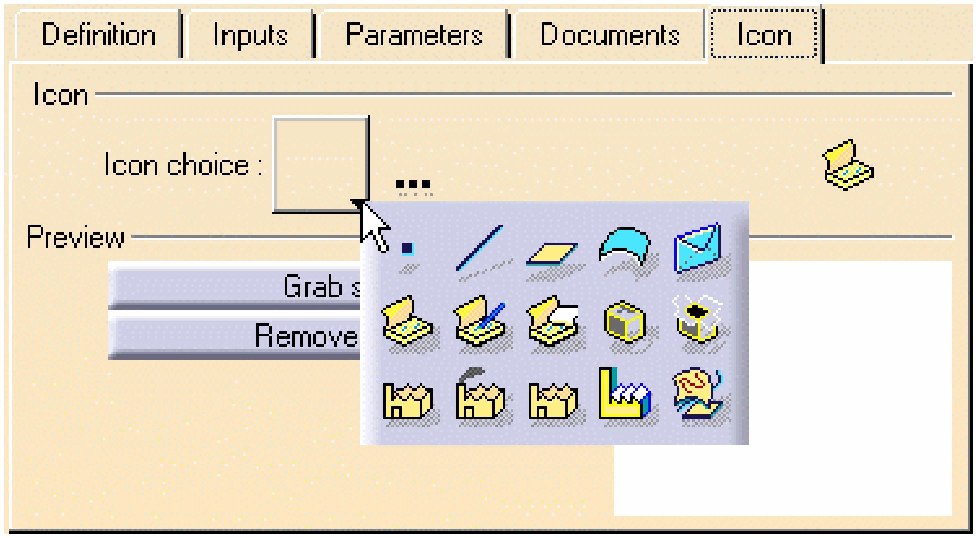
Simply check the Published button.

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.

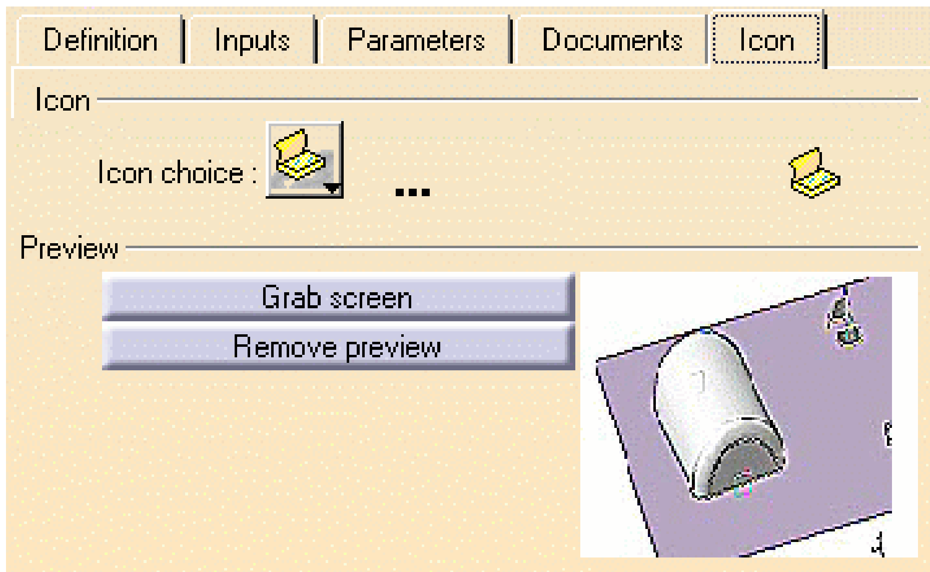


The **Icon** tab lets you modify the icon identifying the PowerCopy in the specifications tree.



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.
- Once your power copy is created, do not delete the referenced elements used to make up the PowerCopy.

Measure Tools cannot be used with PowerCopies.

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

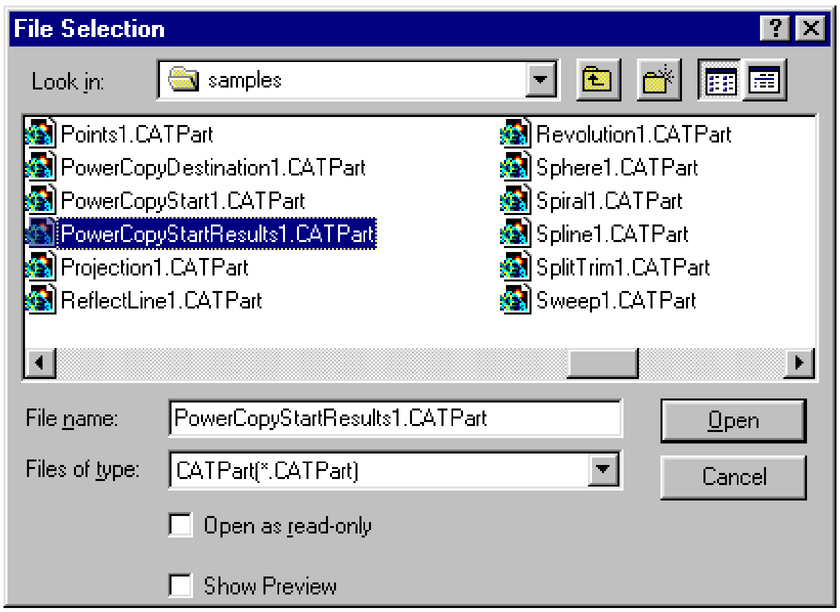
Furthermore, the use of the Replace viewer, regardless of the instantiation type, is detailed.

Open the PowerCopyDestination1.CATPart document.

Using the icon or menu item:

1. Click the **Instantiate From Document** 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

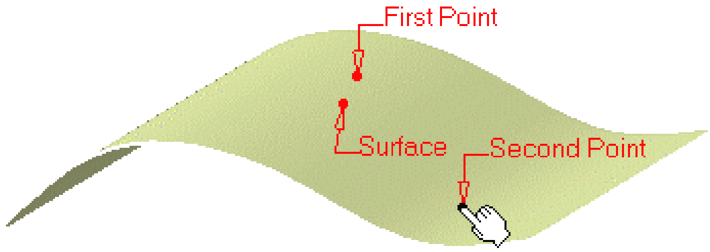
   document.

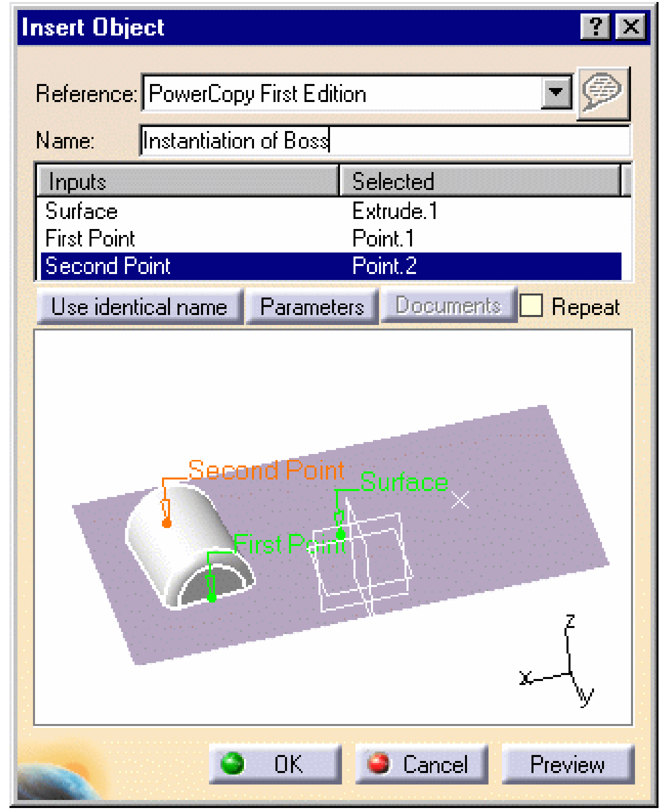
The Insert Object dialog box is displayed.

Use the **Reference** list to choose the correct PowerCopy
when several have been defined in the document and key in
a **Name** for the instantiated reference.

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.

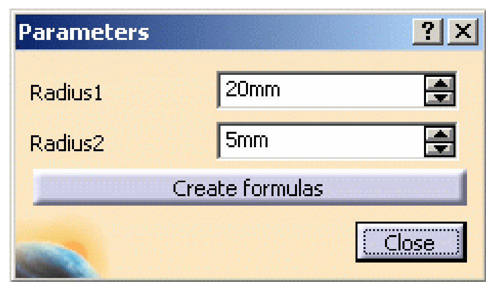
   This is especially useful when the input is the same one repeated several time.

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.

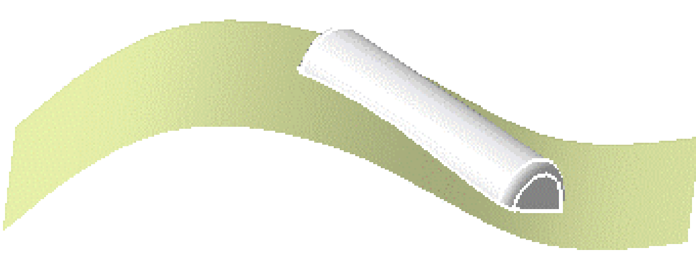
7. Click Close.



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.

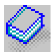
- The  icon is always grayed when instantiating Power Copies. It is available with User Features and allows you to create and modify URLs.

- Check the **Repeat** button 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.

Using the catalog:

You need to have a catalog available, created either:

- using the Catalog capability, see *Infrastructure User's Guide*
- using the **Insert -> Advanced Replication Tools -> PowerCopy Save In Catalog...** menu item.

1. Click the **Open catalog**  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 above starting on step 3.

Using the Replace Viewer

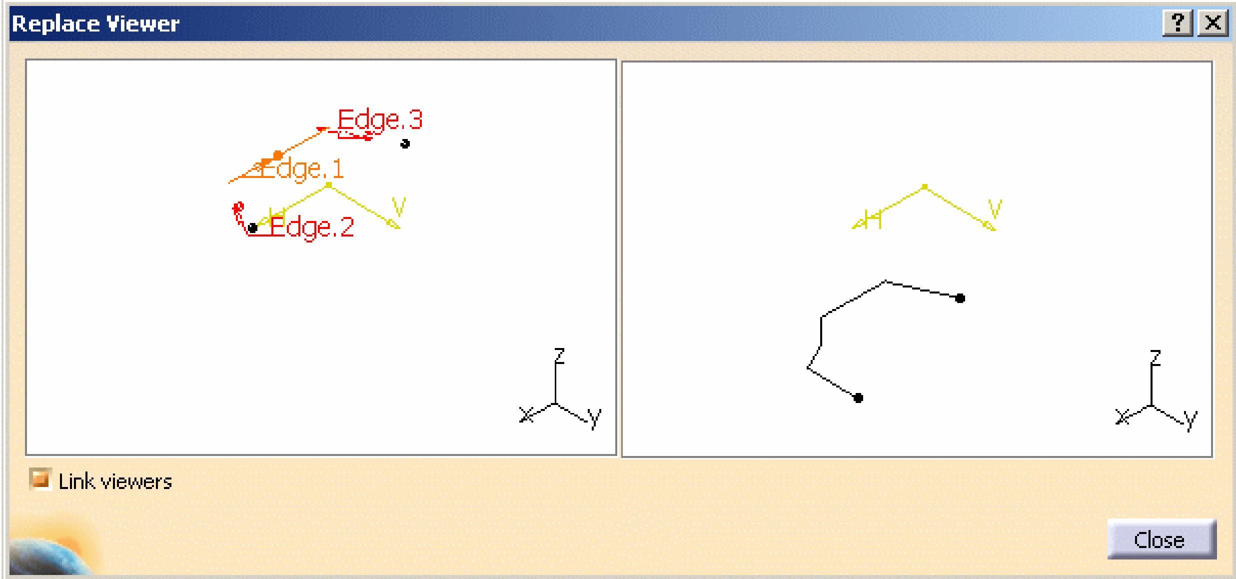
In some cases, when instantiating a powercopy, the replacing element does not present the same sub-elements as the replaced element. Therefore you need to clearly indicate in a specific dialog box, the Replace Viewer, how to rebuild the geometry from the replacing element. In the following example, the 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.

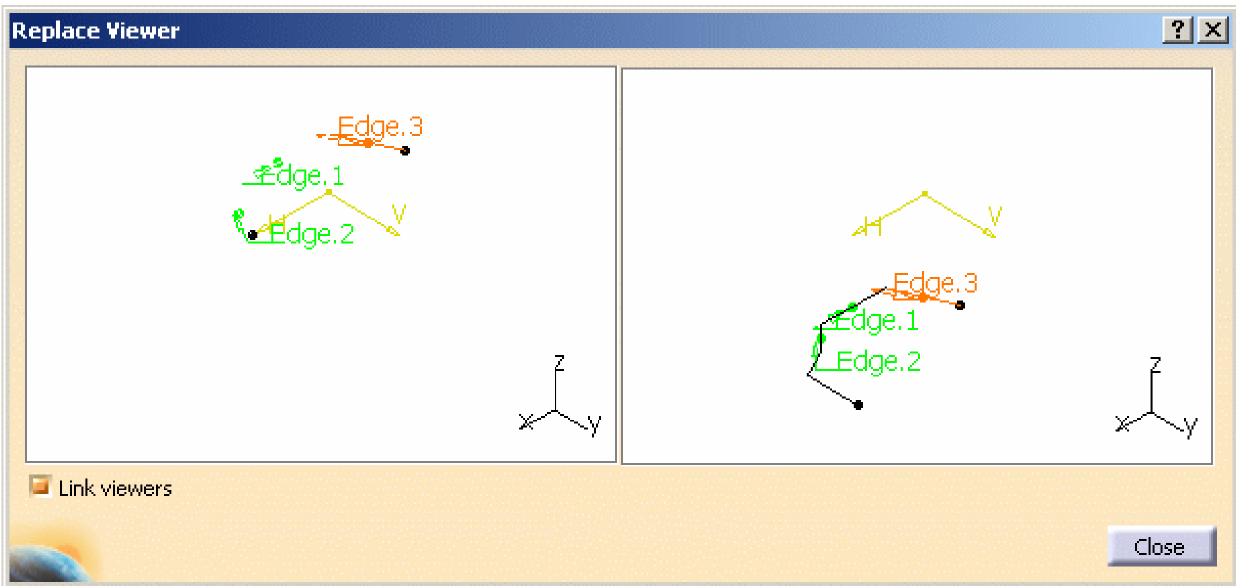
1. Expand the PowerCopy entry in the specification tree, right-click the PowerCopy.1 feature, and choose **PowerCopy.1 object -> Instantiate** command.

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.

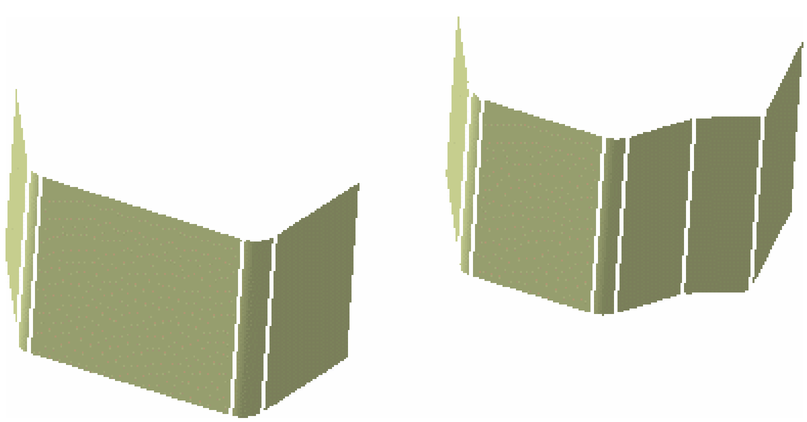


4. Click Close in the Replace Viewer.

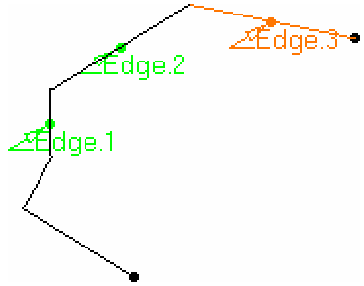
5. Select the XY plane, or click the **Use Identical Name** button to select it as the needed plane.

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.



The feature defined as the current object corresponds to the last instantiated component of the PowerCopy.

Working with the datum mode is independent from the instantiation type: indeed PowerCopies behave as a Copy as Specified and not as a Copy as Result.
For further information about the various formats available when pasting elements, please refer to the Using the Paste Special... Command in *CATIA Infrastructure User's Guide* documentation.

A panel allows you to select alternate document access methods.
See Opening Existing Documents Using the Browse Panel in *CATIA Infrastructure User Guide*.

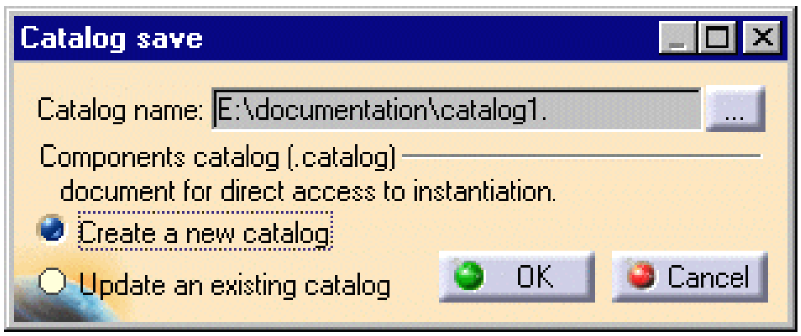
Saving PowerCopies into a Catalog

This task shows how to use store Power Copy elements into a catalog, for later use as described in Instantiating a PowerCopy.

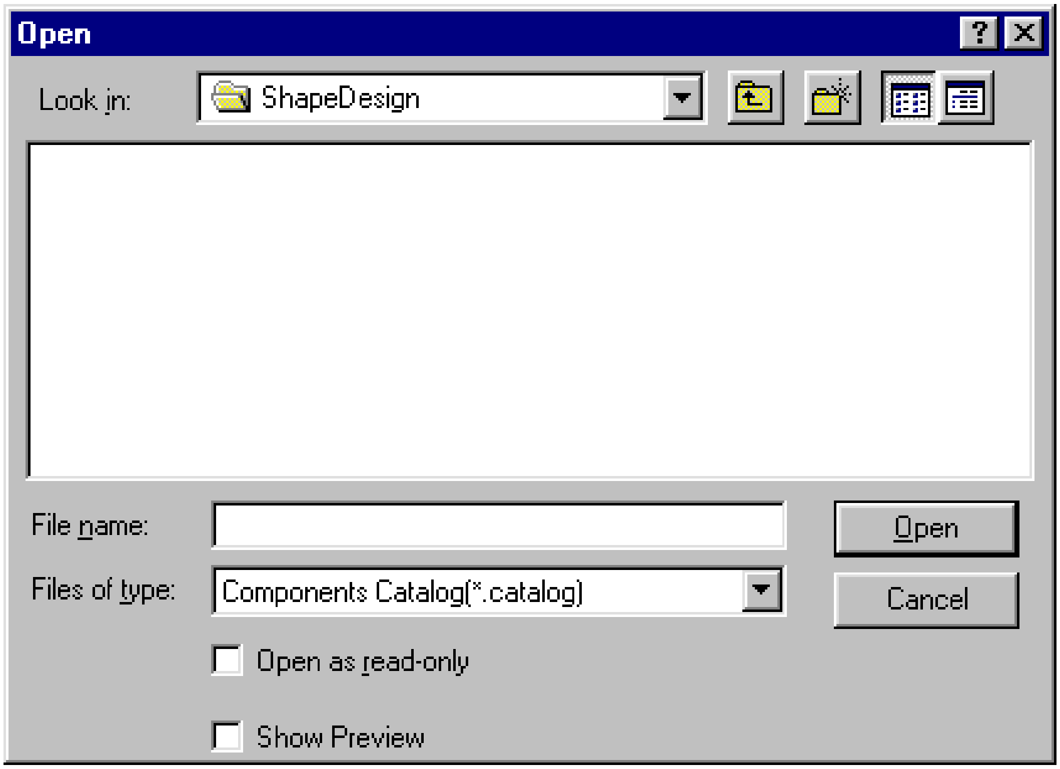
Open the PowerCopyStartResults1.CATPart document.

1. Select the PowerCopy from the specification tree for example.

2. Click the **PowerCopy Save In Catalog** icon or choose the **Insert -> Knowledge Templates -> 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.

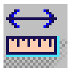
By default, the **Catalog Save** dialog box recalls the catalog accessed last.

3. Click OK.

The PowerCopy has been stored in the catalog.

Measure Tools

You can create a link between a measure and a parameter (length or angle) using the following methods:

 Measuring distances and angles: Click the Measure Between icon, set the measure type and mode in the Measure Between dialog box, then select two entities.

 Measuring properties: Click the Measure Item icon, then select an item.

 Measuring Inertia: Click the Measure Inertia icon, then select an item.

Measuring Distances between Geometrical Entities

The Measure Between command lets you measure distance between geometrical entities. You can measure:

- Minimum distance and, if applicable angles, between points, surfaces, edges, vertices and entire products
  Or,
- Maximum distance between two surfaces, two volumes or a surface and a volume.

This section deals with the following topics:

Measuring minimum distance and angles
Measuring maximum distance
Measuring distances in a local axis system
Customizing measure between
Editing measures
Creating geometry from measure results
Exact measures on CGRs and in visualization mode
Measuring angles
Updating measures
Using measures in knowledgeware
Measure cursors

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 cfysm/samples.

Restriction: Neither Visualization Mode nor cgr files permit selection of individual vertices.

Note: In the No Show space, the Measure Between command is not accessible.

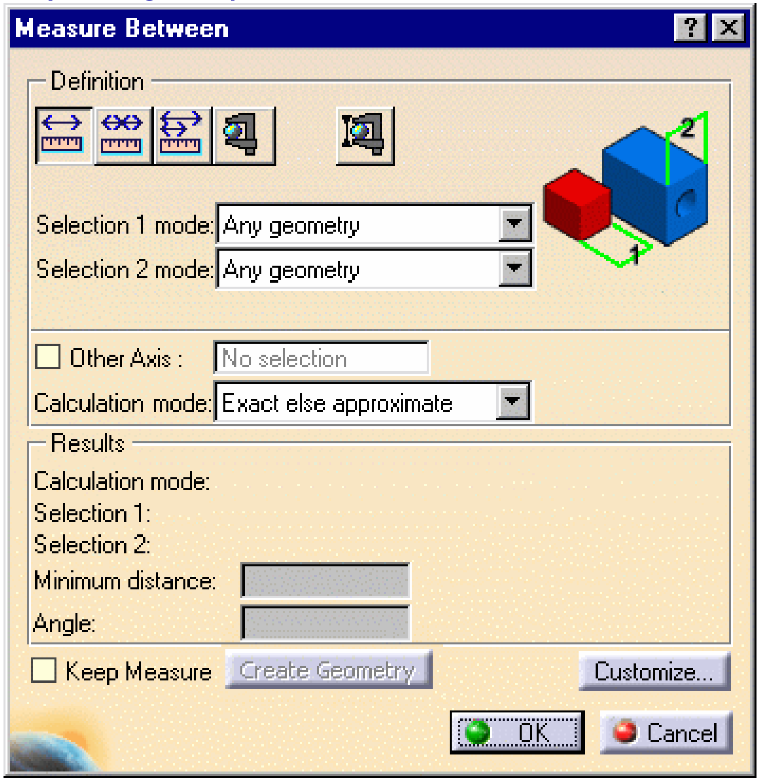
Measuring Minimum Distance and Angles

This task explains how to measure minimum and, if applicable, angles between geometrical entities (points, surfaces, edges, vertices and entire products).

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.

Dialog box options

- 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.

  Some measures kept as features are associativeand can be used to valuate parameters or in formulas.

In the Drafting and Advanced Meshing Tools workbenches, measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

- A Create Geometry option in the dialog box lets you create the points and line corresponding to the minimum distance result.
- A Customize... option opens the Measure Between Customization dialog box and lets you set the display of measure results.

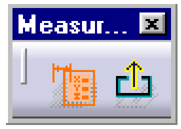
Accessing other measure commands

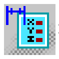
- The Measure Item command  is accessible from the Measure Between dialog box.
- In DMU, the Measure Thickness command is also accessible from the Measure Between dialog box. For more information, see the *DMU Space Analysis User's Guide*.

P1-Only Functionality

In P1, the Measure Tools toolbar appears.

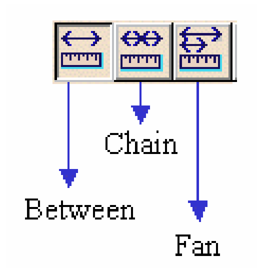
This toolbar has two icons:

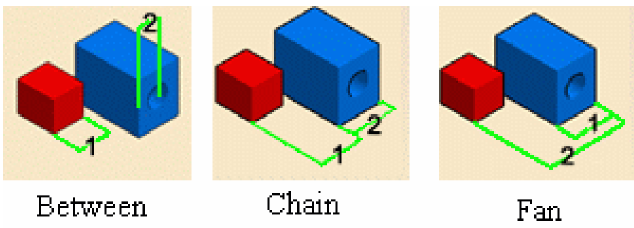


- Measure Dialogs : lets you show or hide the associated dialog box.
- Exit Measure : lets you exit the measure. This is useful when the dialog box is hidden.

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.

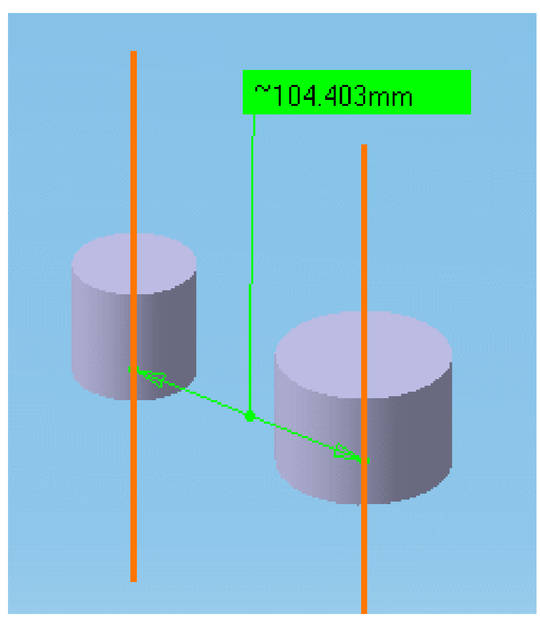
3. Set the desired mode in the Selection 1 and Selection 2 mode drop-down list boxes.

Defining Selection 1 & Selection 2 Modes

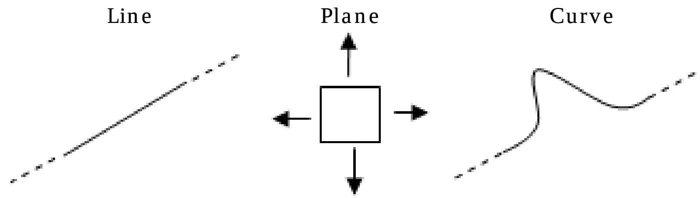
- Any geometry (default mode): measures distances and, if applicable, angles between defined geometrical entities (points, edges, surfaces, etc.).
  Note: The Arc center mode is activated in this selection mode.

This mode recognizes the axis of cylinders and lets you measure the distance between two cylinder axes for example.

Selecting an axis system in the specification tree makes the distance measure from the axis system origin.
You can select sub-entities of V5 axis systems in the geometry area only. For V4 axis systems, distances are always measured from the origin.

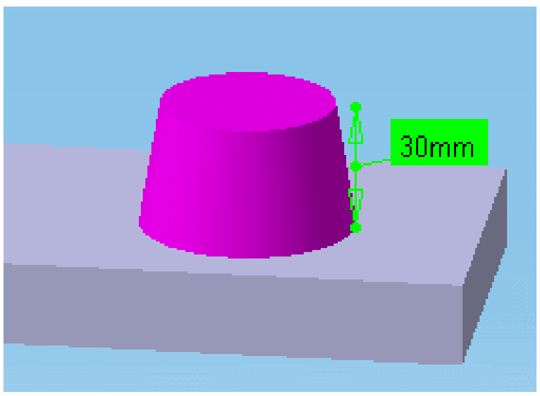


- Any geometry, infinite: measures distances and, if applicable, angles between the infinite geometry (plane, line or curve) on which the selected geometrical entities lie. Curves are extended by tangency at curve ends.

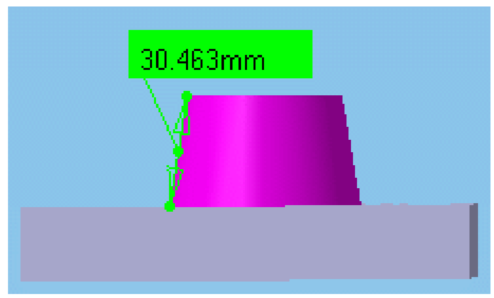


The Arc center mode is activated and this mode also recognizes cylinder axes. For all other selections, the measure mode is the same as any geometry.

Any geometry, infinite



Any geometry

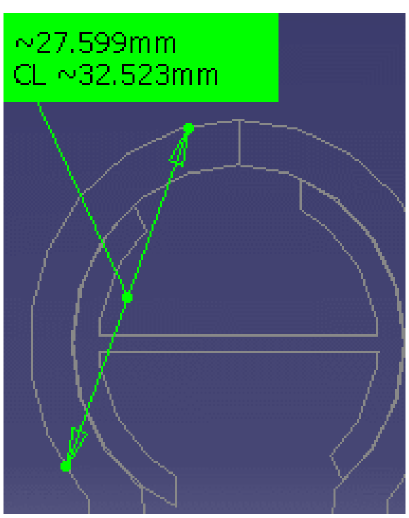


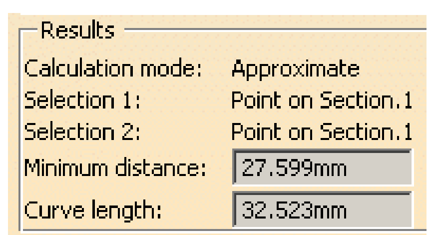
- Picking point: measures distances between points selected on defined geometrical entities. Always gives an approximate measure.

In the DMU section viewer, selecting two picking points on a curve gives the distance along the curve between points (curve length or CL) as well as the minimum distance between points.

Notes:

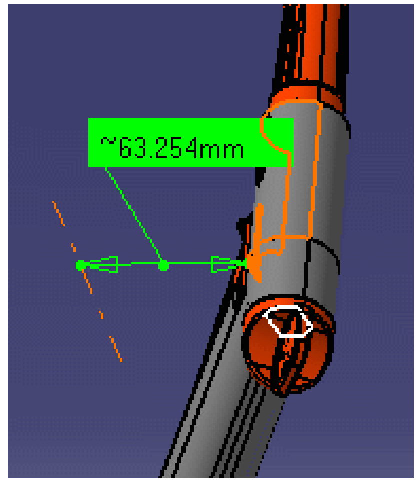
- Both points must be located on the same curve element.
- The minimum distance option must be set in the Measure Between Customization dialog box.





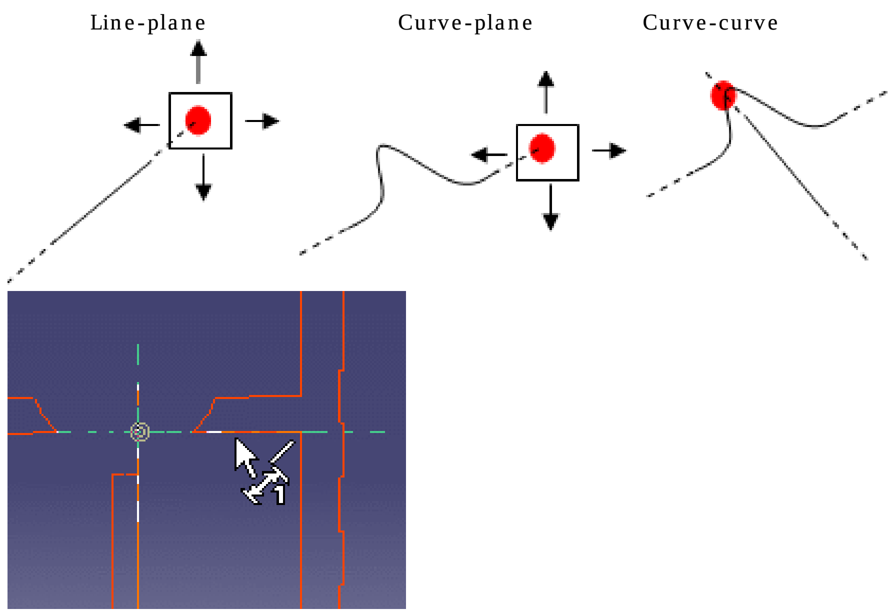
- Point only: measures distances between points. Dynamic highlighting is limited to points.
- Edge only, Surface only: measures distances and, if applicable, angles between edges and surfaces respectively. Dynamic highlighting is limited to edges or surfaces and is thus simplified compared to the Any geometry mode. All types of edge are supported.
- Product only: measures distances between products. Products can be specified by selecting product geometry, for example an edge or surface, in the geometry area or the specification tree.
- 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 points of intersection between two lines/curves/edges or a line/curve/edge and a surface. In this case, two selections are necessary to define selection 1 and selection 2 items.

Geometrical entities (planar surfaces, lines and curves) are extended to infinity to determine the point of intersection. Curves are extended by tangency at curve ends.

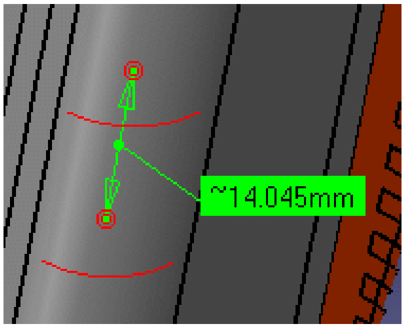


Note: Only intersections which result in points of intersection are managed.

- 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 and non associative.



- 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.
  After some geometric operations, vertices (and corresponding macropoints) may combine several representations on different supports (curves or surfaces). These representations are not all located in the same position in space which means that the exact position of the vertex cannot be determined. Only one vertex representation is visualized.
  Measure Between measurements are made with respect to the visualized representation. Measuring distance between two points therefore depends on the chosen representation. Any calculation errors are due to the fact that the exact position of the vertex cannot be determined.
- 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).

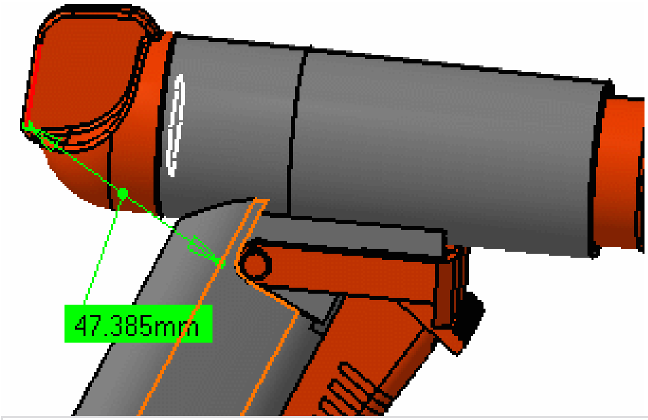
Notes:

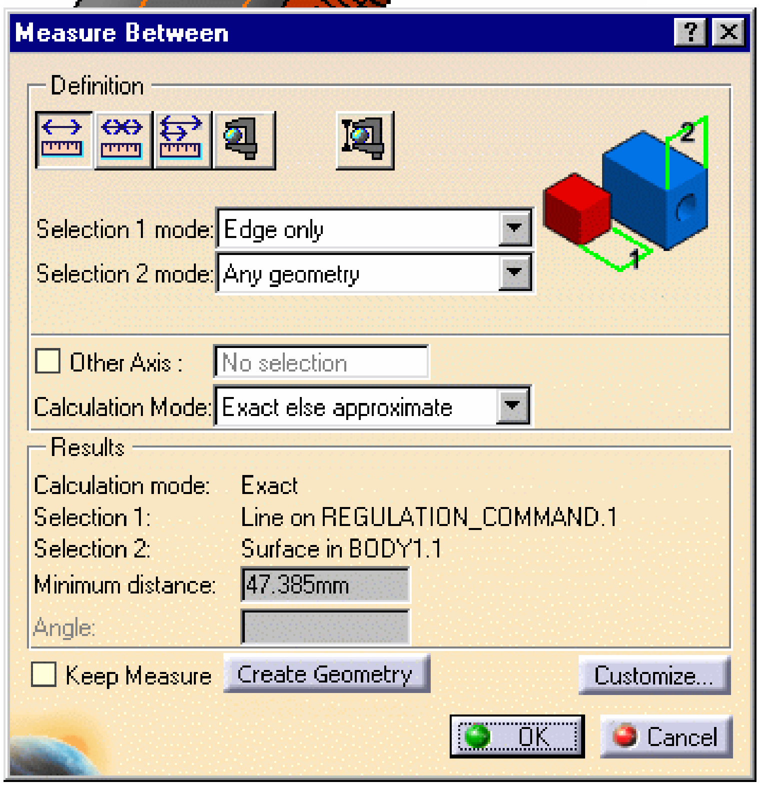
- The appearance of the cursor has changed to assist you.
- Dynamic highlighting of geometrical entities helps you locate items to click on.

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.

Note: For reasons of legibility, angles between lines and/or curves of less than 0.02 radians (1.146 degrees) are not displayed in the geometry area.

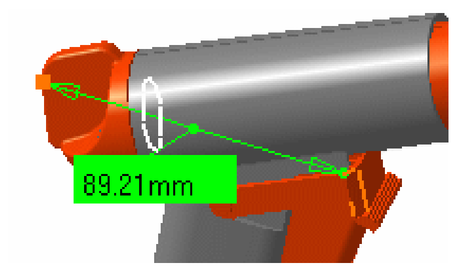




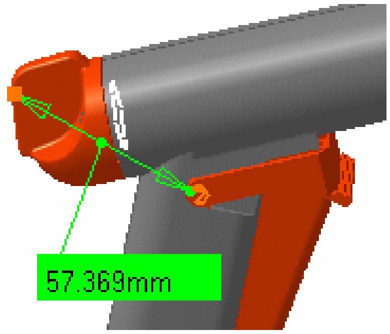
By default, the overall minimum distance and angle, if any, between the selected items are given in the Measure Between dialog box.

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.

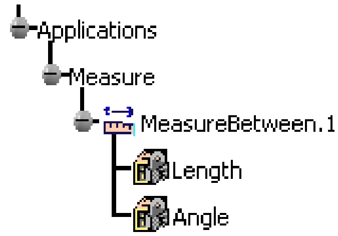


10. Select another item.



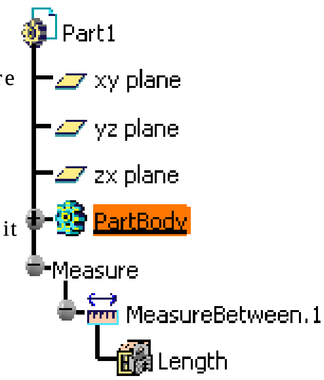
Using the Other Selection... command in the contextual menu, you can access the center of spheres.

11. 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.

Sectioning measure results

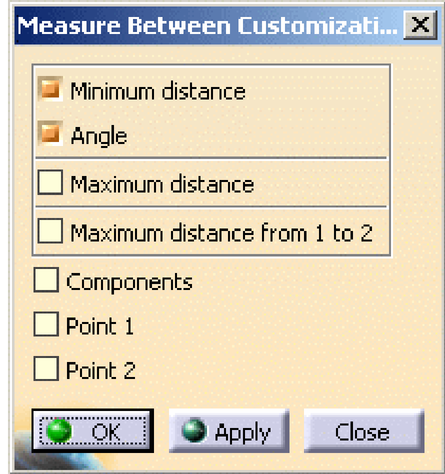
Having made and kept your measure, select it then click the Sectioning icon to section measure results. The plane is created parallel to the direction defined by the measure and sections entities selected for the measure only. All section plane manipulations are available.

Note: You may need an appropriate license to access the Sectioning command.

Customizing Measure Between

Customizing lets you choose what distance you want to measure:

- Minimum distance (and angle if applicable)
- Maximum distance
- Maximum distance from 1 to 2.



Note: These options are mutually exclusive. Each time you change option, you must make your measure again.

By default, minimum distances and if applicable, angles are measured.

You can also choose to display components and the coordinates of the two points (point 1 and point 2) between which the distance is measured.

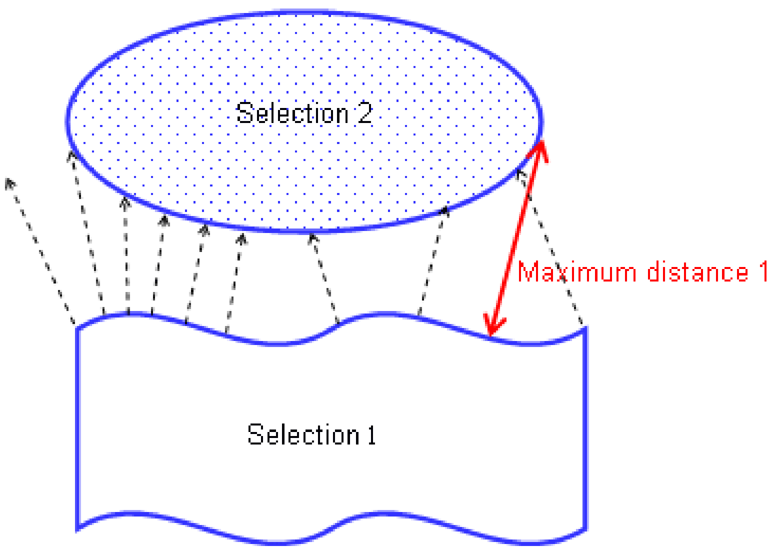
What you set in the dialog box determines the display of the results in both the geometry area and the dialog box.

Measuring Maximum Distance

You can measure the maximum distance between two surfaces, two volumes or a surface and a volume.

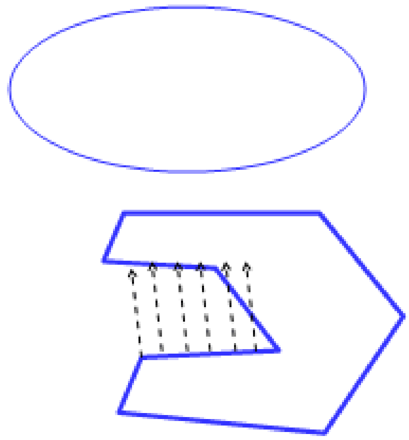
Distance is measured normal to the selection and is always approximate. Two choices are available:

- 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.

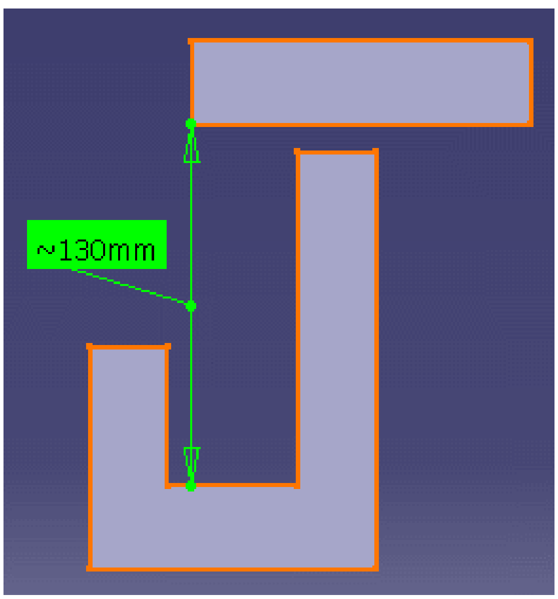


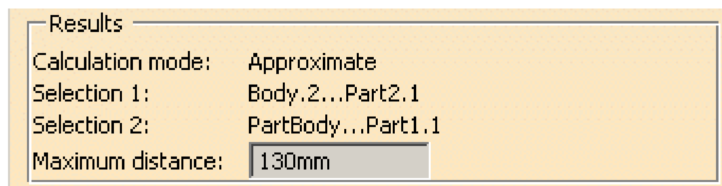
- Maximum distance: gives the highest maximum distance between the maximum distance measured from selection 1 and the maximum distance measured from selection 2.

Note: All selection 1 (or 2) normals intersecting selection 1 (or 2) are ignored.



1. Click Customize... and check the appropriate maximum distance option in the Measure Between Customization dialog box, then click **OK**.
2. Make your measure:
   - Select the desired measure type
   - Set the desired selection modes
   - Set the desired calculation mode
   - Click to select two surfaces, two volumes or a surface and a volume.





3. Click **OK** when done.

Measuring Distances in a Local Axis System

An Other Axis option in the dialog box lets you measure distance in a local axis system.

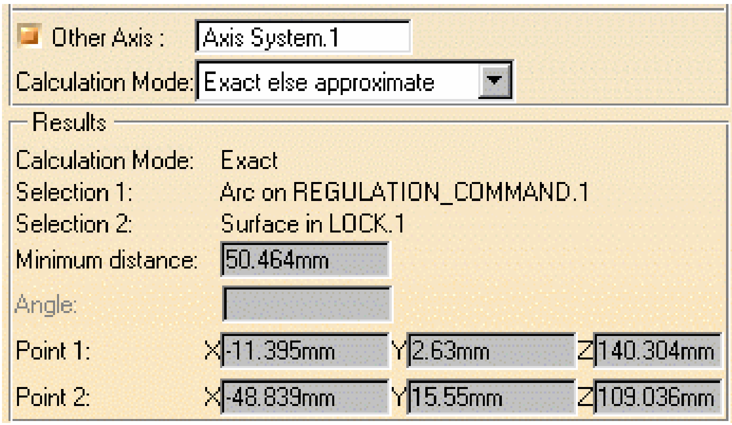
This type of measure is associative: if you move the axis system, the measure is impacted and can be updated.

You will need a V5 axis system.

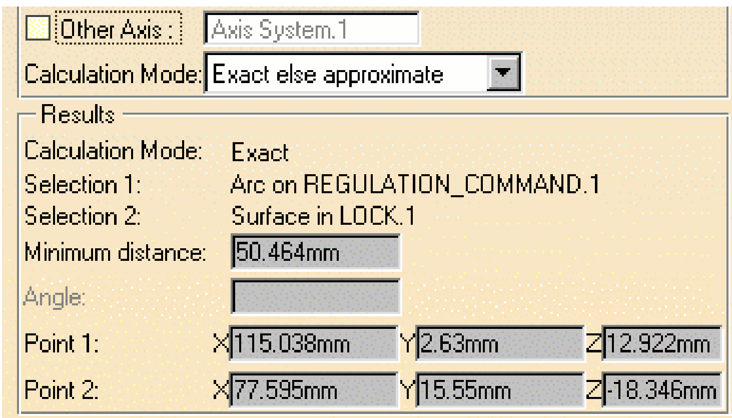
1. Select the Other Axis checkbox in the dialog box.
2. Select a V5 axis system in the specification tree or geometry area.

3. Make your measure.

In the examples below, the measure is a minimum distance measure and the coordinates of the two points between which the distance is measured are shown.



Same measure made with respect to absolute axis system:



Note: All subsequent measures are made with respect to the selected axis system.

4. To change the axis system, click the Other Axis field and select another axis system.
5. To return to the absolute axis system, click to clear the Other Axis checkbox.
6. Click **OK** when done.

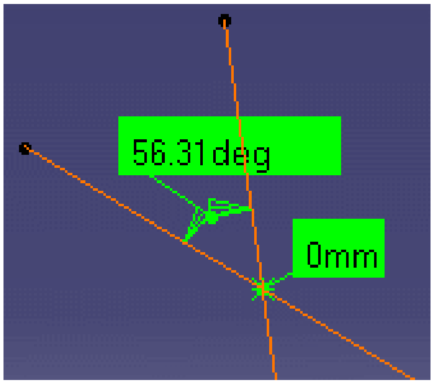
Measuring Angles

- Exact angles
- Complementary angles

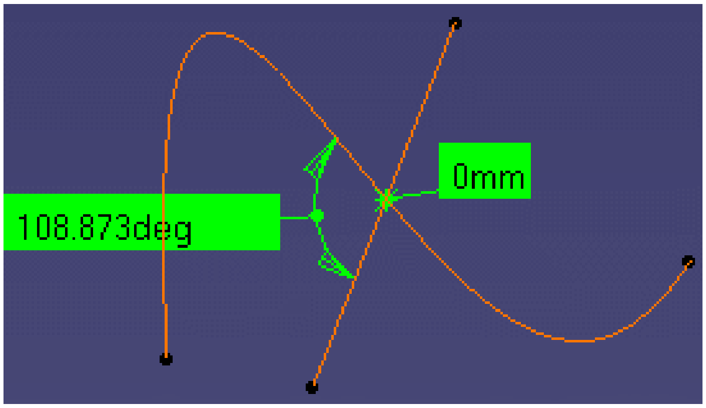
Exact Angles

The Measure Between command lets you measure exact angles between the following geometrical entities that have (at least) one common point.

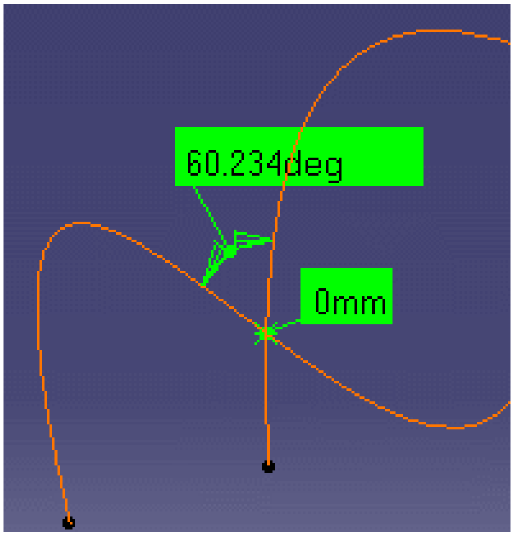
Two lines (even if not in the same plane):



A line and a curve:

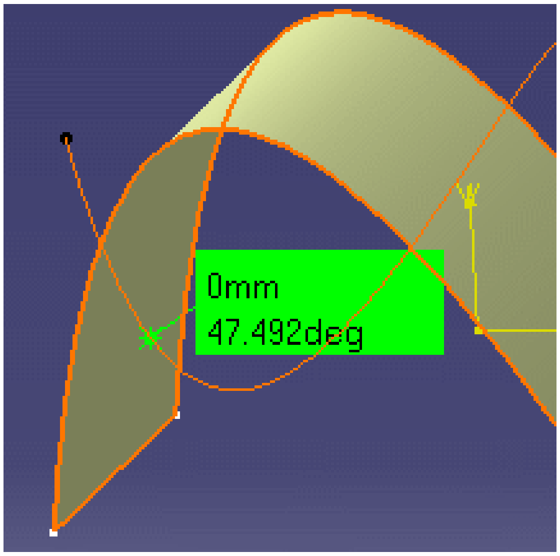


Two curves:

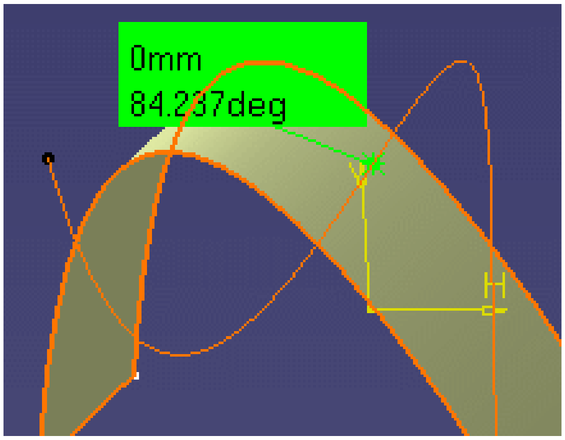


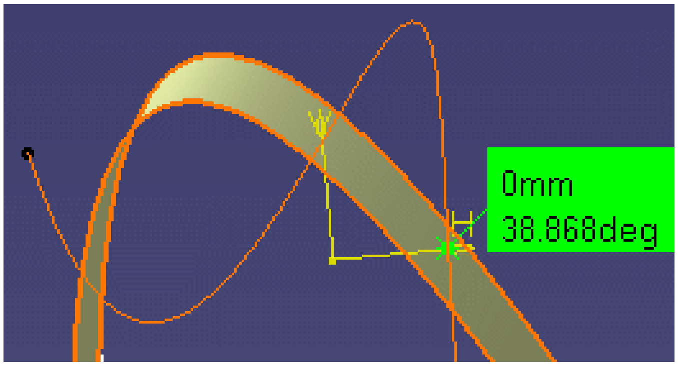
Note: In the above three cases, if entities intersect more than once, the measure is made at the point of intersection nearest the point at which selection 1 is made.

A curve and a surface:

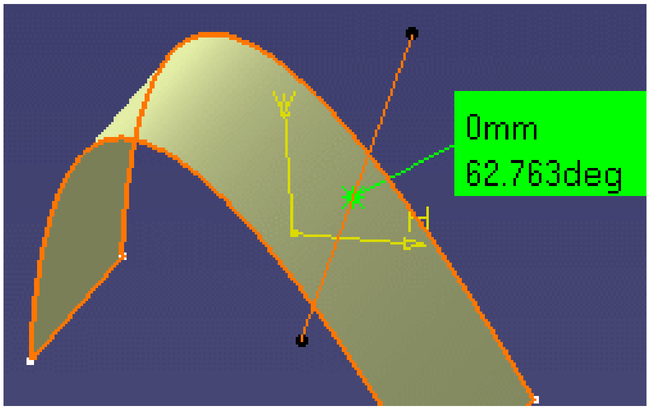


Note: If the curve and surface intersect more than once, the measure is made at the point of intersection nearest the point of the selection on the curve.

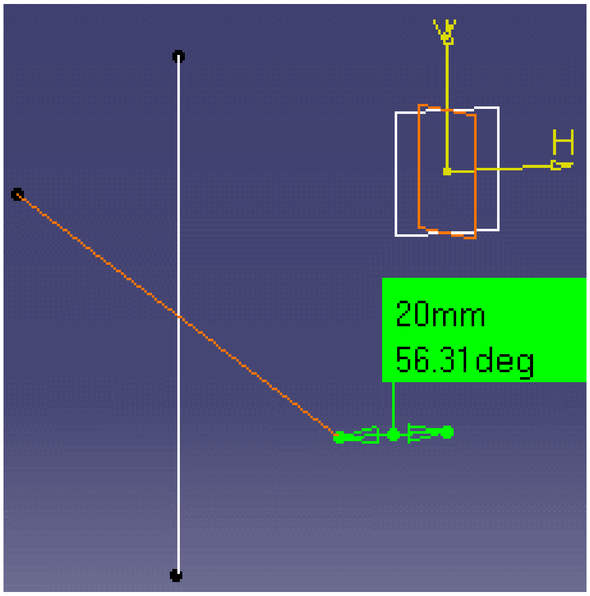




A line and a surface:



A line and a plane:

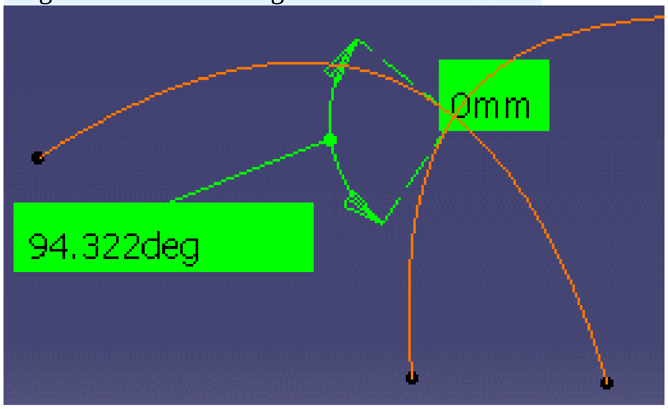


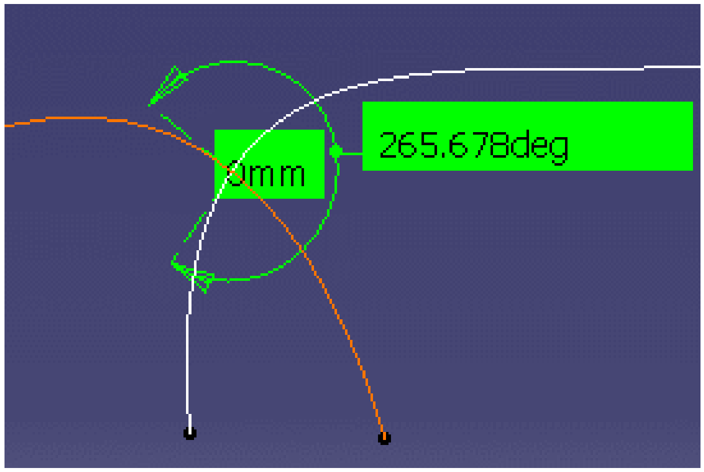
Two surfaces: You can also measure the angle between two surfaces provided both surfaces are planar.

Complementary Angles

You can obtain the complementary angle (360° - the initial angle measured) when measuring between two curves: drag the angle line to show the complementary angle.

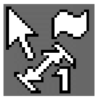
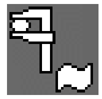
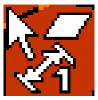
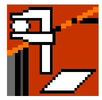
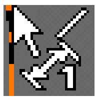
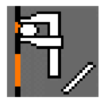
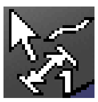
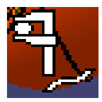
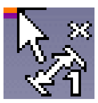
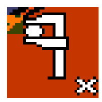
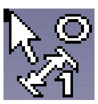
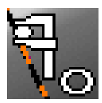
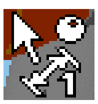
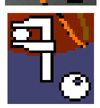
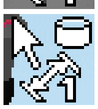
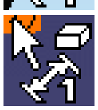
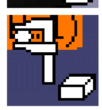
Note: The dialog box and knowledge parameters are refreshed. The value of the complementary angle is stored along with the measure.

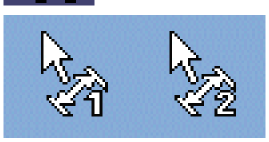




Measure Cursors

The appearance of the Measure Between and Measure Item cursor changes as you move it over items to reflect the measure command you are in and to help you identify the selection. Dynamic highlighting of surfaces, points, and vertices, etc. also helps you locate items to click on.

Measure Between	Measure Item	Geometry
		Surface
		Planar surface
		Line
		Curve
		Point
		Circle
		Sphere
		Cylinder
		Volume



In Measure Between, a number (1 for selection 1 and 2 for selection 2) identifies where you are in your measure.

Measuring Properties

The Measure Item command lets you measure the properties associated to a selected item (points, edges, surfaces and entire products).

This section deals with the following topics:

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 cfysm/samples.

Restriction: Neither Visualization Mode nor cgr files permit selection of individual vertices.

Note: In the No Show space, this command is not accessible.

Measuring Properties

This task explains how to measure the properties associated to a selected item.

1. Switch to Design Mode (**Edit ->Representations ->Design Mode**).

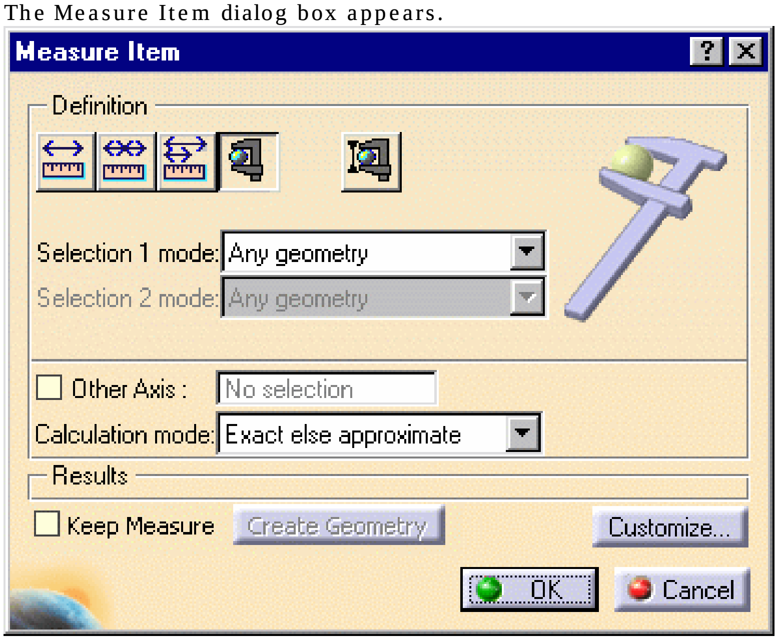
2. Set **View -> Render Style** to Shading with Edges.

   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.

Dialog box options

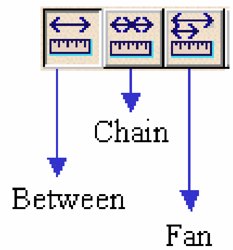
- You can also measure properties with respect to a local V5 axis system.
- The Keep Measure option lets you keep current and subsequent measures as features. This is useful if you want to keep measures as annotations for example.

  Some measures kept as features are associativeand can be used to valuate parameters or in formulas.

In the Drafting and Advanced Meshing Tools workbenches, measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

- A Create Geometry option in the dialog box lets you create the center of gravity from measure results.
- A Customize... option lets you customize the display of measure results.

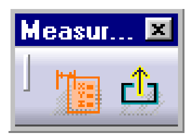
Accessing other measure commands

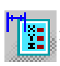


- The Measure Between command is accessible from the Measure Item dialog box. Simply click one of the Measure Between icons in the Definition box to switch commands.
- In DMU, the Measure Thickness command is also accessible from the Measure Item dialog box. For more information, see the appropriate 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.

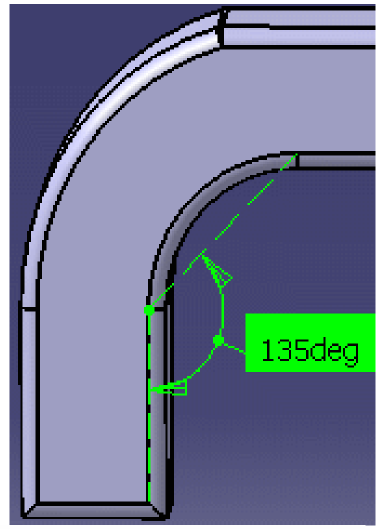
Defining the Selection 1 Mode

- 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.

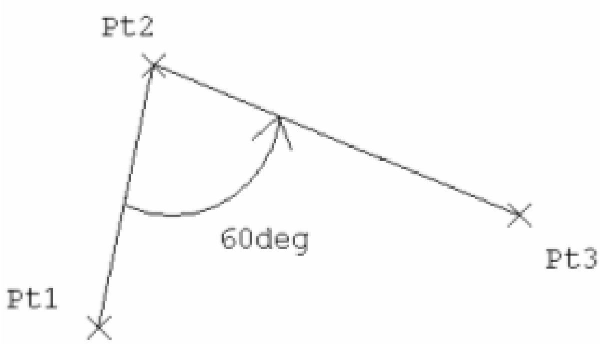
  In the last three modes, dynamic highlighting is limited to points, edges or surfaces depending on the mode selected, and is thus simplified compared to the Any geometry mode.

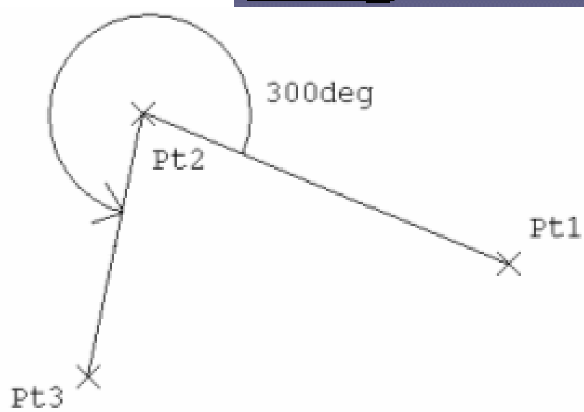
- Product only: measures distances between products.
  Products can be specified by selecting product geometry, for example an edge or surface, in the geometry area or the specification tree.
- Angle by 3 points: measures the angle between two lines themselves defined by three points.

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).

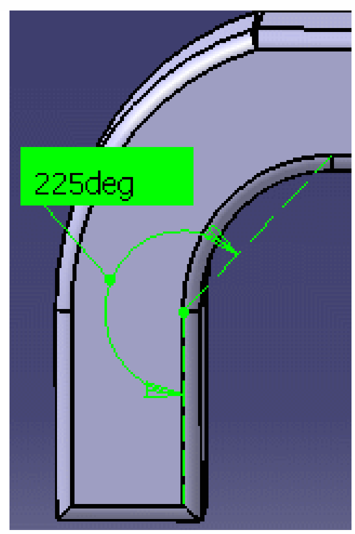


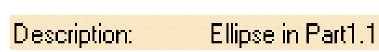


You can drag the angle line to show the complementary angle (360° - the initial angle measured).

You can also obtain the complementary angle when measuring the angle on arcs.

Note: The dialog box and knowledge parameters are refreshed. The value of the complementary angle is stored along with the measure.



- Thickness (DMU only): measures the thickness of an item. For more information, see the appropriate task in the *DMU Space Analysis User's Guide*.
- 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. 

- 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.

5. Set the desired calculation mode in the Calculation mode drop-down list box.

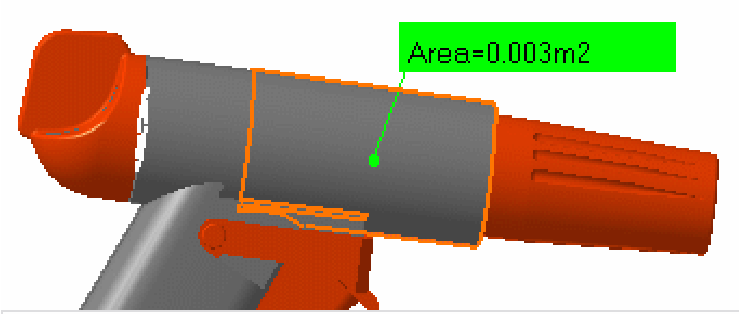
Defining the Calculation Mode

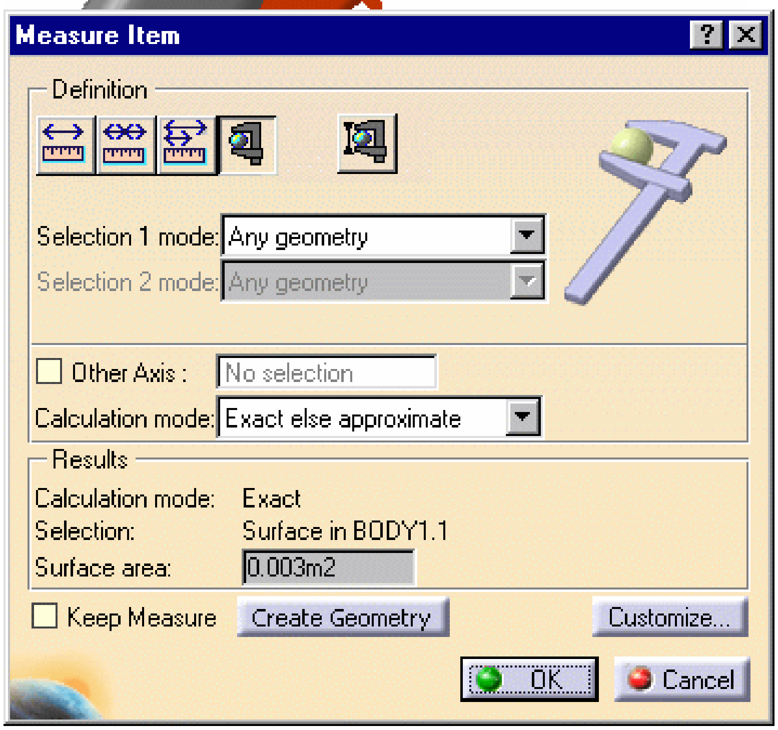
- Exact else approximate (default mode): measures access exact data and wherever possible true values are given. If exact values cannot be measured, approximate values are given (identified by a ~ sign).
- Exact: measures access exact data and true values are given. Note that you can only select exact 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.

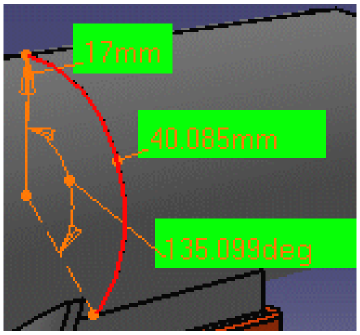


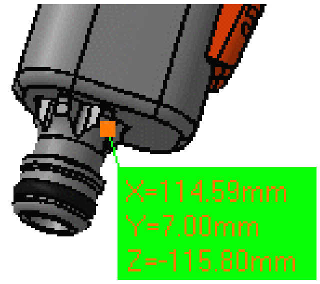


The dialog box gives information about the selected item, in our case a surface and indicates whether the result is an exact or approximate value. The surface area is also displayed in the geometry area.
The number of decimal places, the display of trailing zeros and limits for exponential notation is 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. Try selecting other items to measure associated properties.

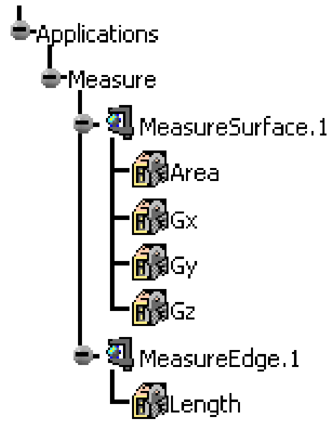
Note: For reasons of legibility, angles measured by Angle by 3 points or on an arc of circle of less than 0.02 radians (1.146 degrees) are not displayed in the geometry area.



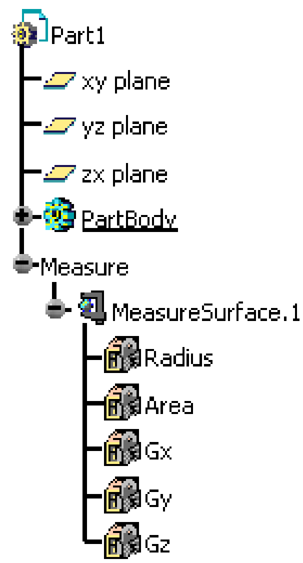


8. Click **OK** when done.

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.

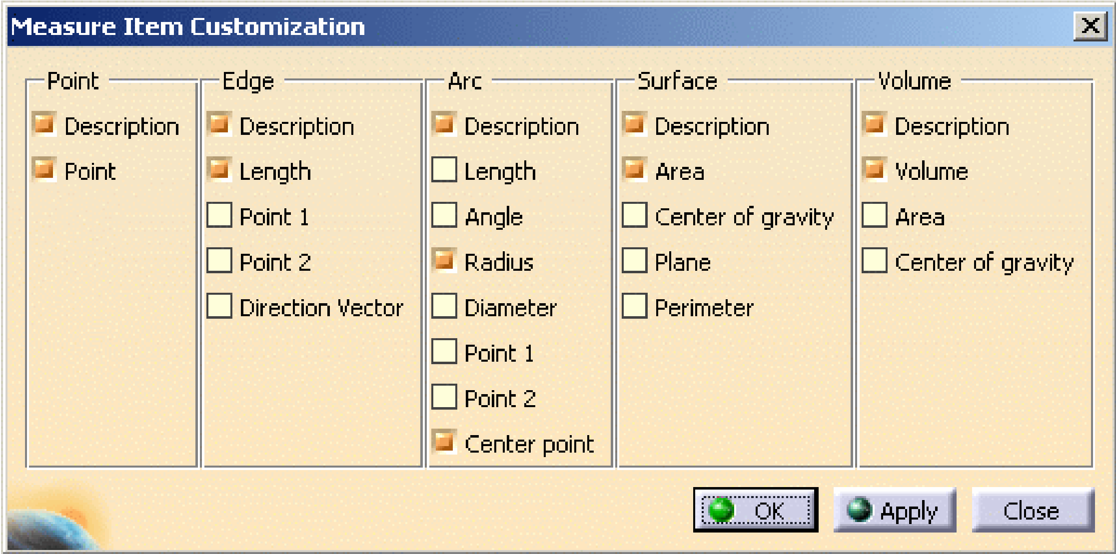
Some measures kept as features are associative. In Design Mode, if you modify a part or move a part in a product structure context and the measure is impacted, it will be identified as not up-to-date in the specification tree. You can then update it locally have it updated automatically.

When measures are used to valuate parameters, an associative link between the measure and parameter is created. Measures can also be used in formulas.

Customizing the Display

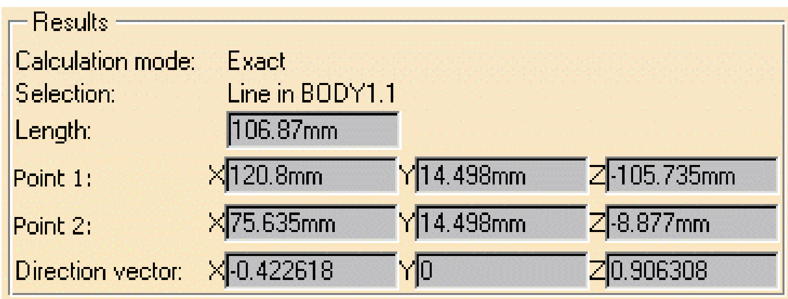
Customizing lets you choose the properties you want to see displayed in both the geometry area and the dialog box.

1. 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:



Edges

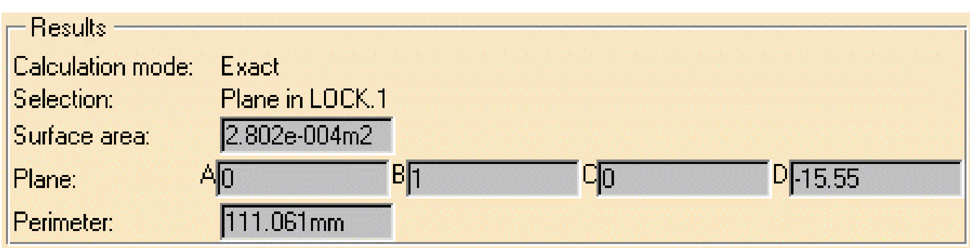
The system detects whether the edge is a line, curve or arc, taking model accuracy into account and displays the properties as set in the Measure Item Customization dialog box.



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.

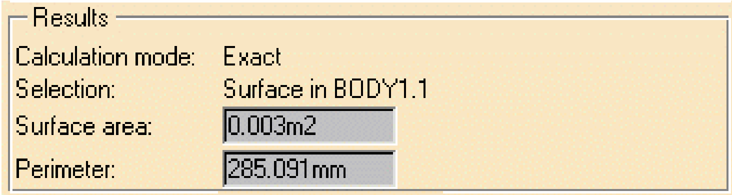
Surfaces

- **Center of gravity**: 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.
- **Plane**: gives the equation of a planar face. The equation of a plane is: **Ax + By + Cz + D=0**.



Note that there is an infinite number of equations possible (and an infinite number of solutions for values ABC and D). The result given by Measure Item does not necessarily correspond to that in the feature specification. This is because the measure is based on topology and does not know the feature specification associated with the measured item.

- **Perimeter**: Visualization mode does not permit the measure of surface perimeter.



2. Set the properties you want the system to detect, then click **Apply** or **Close**.
   The Measure Item dialog box is updated if you request more properties of the item you have just selected.
3. Select other items to measure associated properties.

Measuring Properties in a Local Axis System

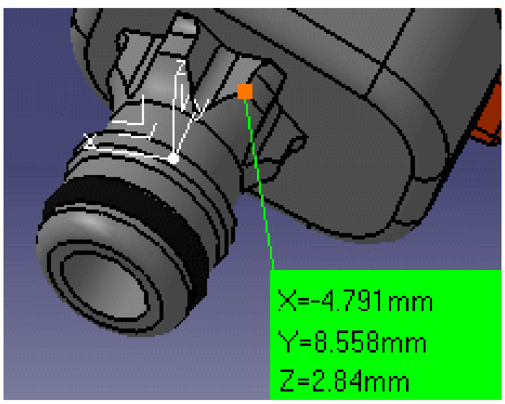
An Other Axis option in the dialog box lets you measure properties in a local axis system.

This type of measure is associative: if you move the axis system, the measure is impacted and can be updated.

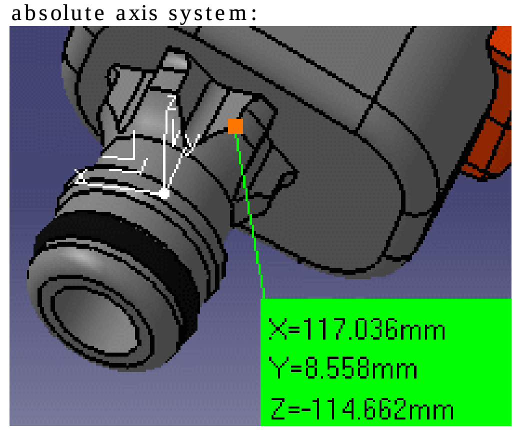
You will need a V5 axis system.

1. Select the Other Axis checkbox in the Measure Item dialog box.
2. Select a V5 axis system in the specification tree or geometry area.
3. Make your measure.

Measure made with respect to local axis system:



Same measure made with respect to absolute axis system:



Note: All subsequent measures are made with respect to the selected axis system.

4. To change the axis system, click the Other Axis field and select another axis system.
5. To return to the main axis system, click to clear the Other Axis checkbox.
6. Click **OK** when done.

Measuring Inertia

The Measure Inertia command lets you measure:

- 3D inertia properties of surfaces and volumes (explained below)
- 2D inertia properties of plane surfaces.

Note: In the No Show space, this command is not accessible.

This section deals with the following topics:

Measuring 3D Inertia

This task explains how to measure the 3D inertia properties of an object.

You can measure the 3D inertia properties of both surfaces and volumes, as well as retrieve the density or surface density if valuated from V4 model type documents. You can also retrieve inertia equivalents set in Knowledgeware formulas.

The area, density, mass and volume (volumes only) of the object are also calculated.

Note: You cannot measure inertia properties of either wireframe or infinite elements.

For examples showing 3D inertia properties measured on surfaces. To find out more about notations used.

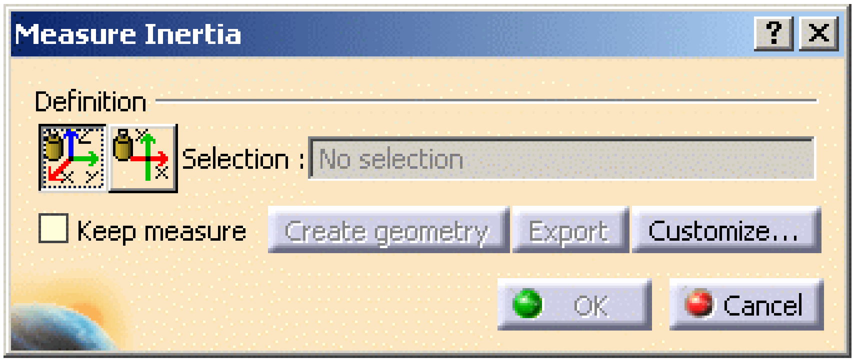
Insert the Valve.cgr document from the samples folder. It is to be found in the online documentation filetree in the common functionalities sample folder cfysa/samples.

1. Click the Measure Inertia icon.

In DMU, you can also select **Analyze -> Measure Inertia** from the menu bar.

The Measure Inertia dialog box appears. By default 3D inertia properties are measured.

The Measure 2D Inertia icon lets you measure 2D inertia properties of plane surfaces.

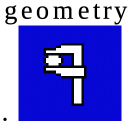


Dialog box options

- A Keep Measure option in the dialog box lets you keep current and subsequent measures as features in the specification tree. Some measures kept as features are associative and can be used as parameters.

In the Drafting workbench, the Keep Measure option is not available. Measures are done on-the-fly. They are not persistent. This means that they are not associative and cannot be used as parameters.

- A Create Geometry option lets you create the center of gravity and the axis system for principal axes in a part from inertia results.
- An Export option lets you write results to a text file.
- A Customize... option lets you define what will be computed and displayed in the dialog box.

Note: When you move the cursor over the geometry or specification tree, its appearance changes to reflect the measure command you are in. 

2. Click to select the desired item in the specification tree, for example Valve.

Selecting Items

- In the geometry area, you can select individual faces and edges on cgr files and in Visualization mode.

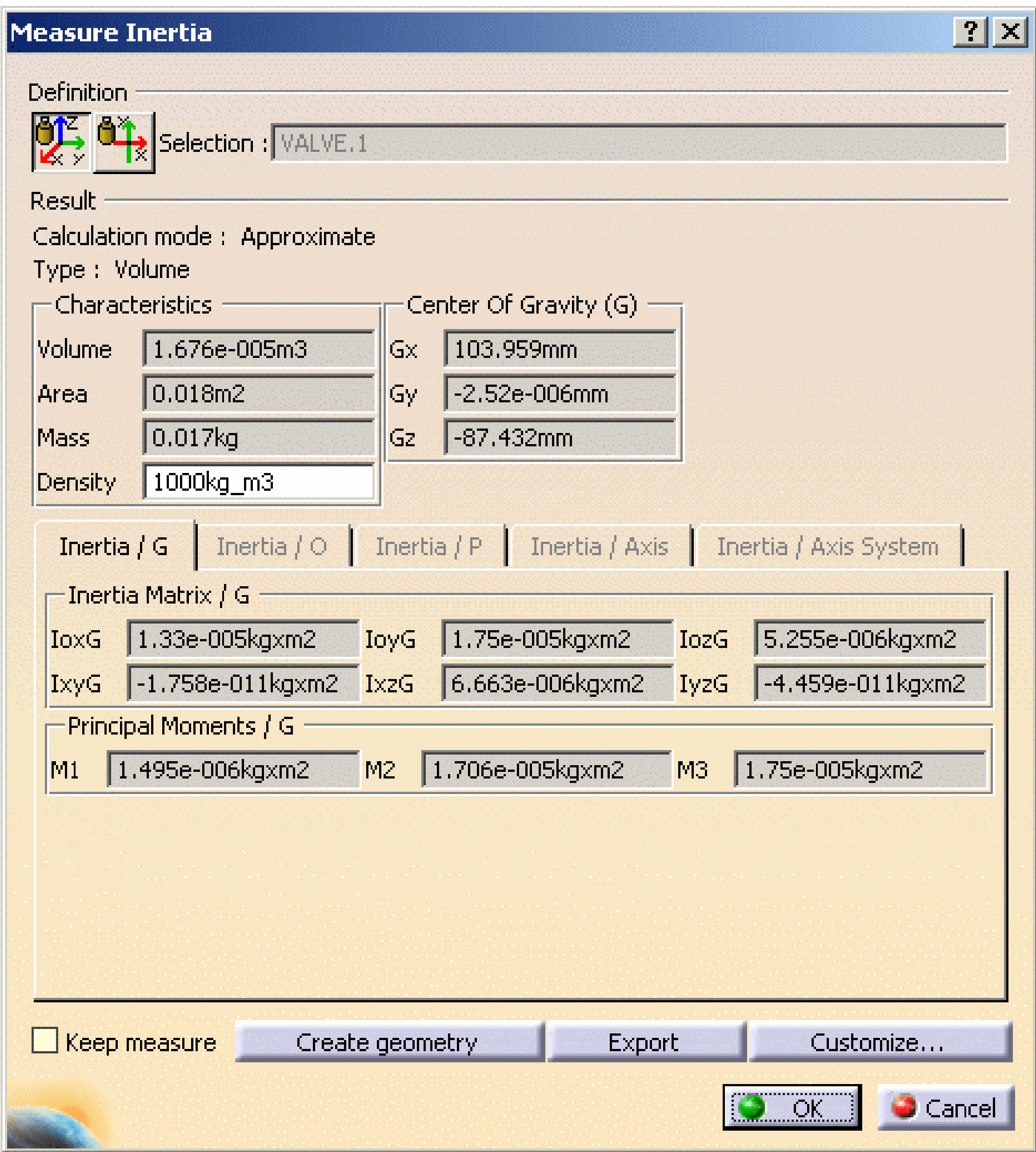
- Ctrl-click in the geometry area or the specification tree to add other items to the initial selection.
- Shift-click in the specification tree to make a multiple selection.
- Drag (using the left mouse button) to select items using the bounding outline.
- (P2 only) Use the Group command to make your multiple selection.

Notes:

- Only items of the same type can be included in a multiple selection or a bounding outline; you cannot mix volumes and surfaces.
- Inertia measures made on a multiple selection of items are not associative.

The **Dialog Box** expands to display the results for the selected item.

The measure is made on the selection, geometry, assembly or part. To measure the inertia of individual sub-products making up an assembly and see the results in the document window, you must select the desired sub-product.
In our example, the item selected has no sub-products.



The dialog box identifies the selected item and indicates whether the calculation is exact or approximate:

- In Design mode, measures access exact data and wherever possible true values are given. Note that it is possible to obtain an exact measure for most items in design mode.
- In Visualization mode, measures are made on tessellated items and approximate values are given.

In addition to the center of gravity G, the principal moments of inertia M and the matrix of inertia calculated with respect to the center of gravity, the dialog box also gives the area, volume (volumes only), density and mass of the selected item.

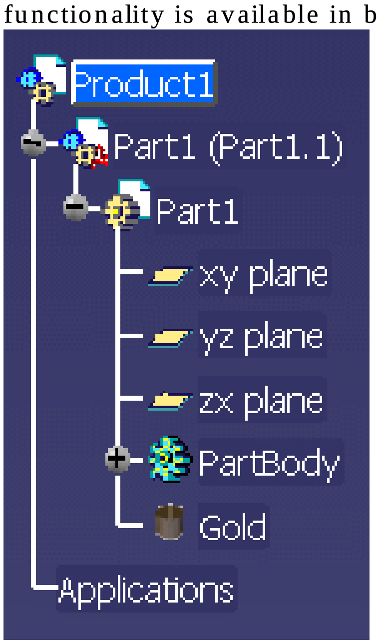
You can also compute and display the principal axes A. To do so, you must first activate the appropriate option in the Measure Inertia Customization dialog box.

The density is that of the material, if any, applied to a product, part or part body:

- If no density is found, a default value is displayed. You can, if desired, edit this value. If you do so, all the other inertia values are re-calculated. The default value is 1000 kg/m3 for volumes and 10 kg/m2 for surfaces.
- If sub-products or part bodies have different densities, the wording Not uniform is displayed.

Notes:

- You can access the density of parts saved as CGR files and opened in visualization mode. This functionality is available in both a part and a product context.



To do so:

  - Select the Save density in cgr option in the Meaure Tools tab (**Tools ->Options ->General ->Parameters and Measure**).
  - Open a part to which material has been applied and save as CGR type.
    The density is stored in the CGR file.

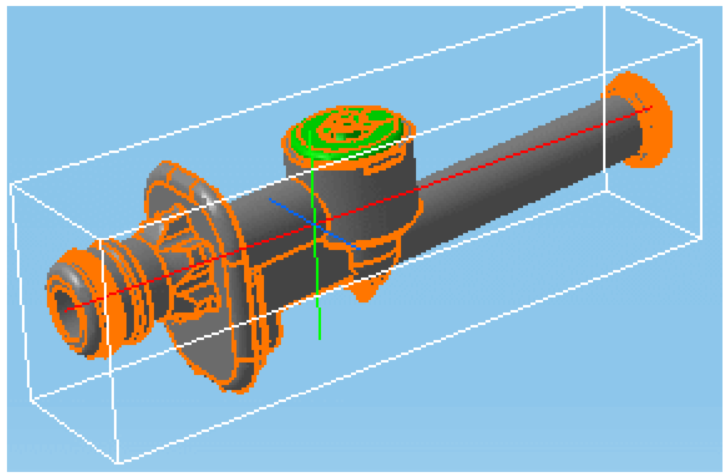
Important: The material must be applied to the part node. If materials are applied to part bodies, no density is saved.

- Close the Part document.
- Open the CGR file or switch to DMU Space Analysis and insert the part saved as CGR, then measure the inertia.

- You must be in design mode to access the density of part bodies to which materials have been applied.
- Unless specified otherwise, material inheritance is taken into account.
- Density is a measure of an item's mass per unit volume expressed in kg/m3; surface density is a measure of an item's mass per unit area expressed in kg/m2.

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**).

In the Geometry Area, axes of inertia are highlighted and a bounding box parallel to the axes and bounding the selected item also appears.



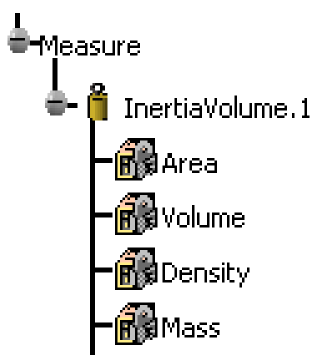
Color coding of axes:

- Red: axis corresponding to the first moment M1
- Green: axis corresponding to the second moment M2
- Blue: axis corresponding to third moment M3.

3. Click Customize... to customize the inertia computation and define what will be exported to the text file.

4. Click **OK** when done.

If you checked the Keep Measure option in the Measure Inertia dialog box, your measures are kept as features and your specification tree will look something like this.

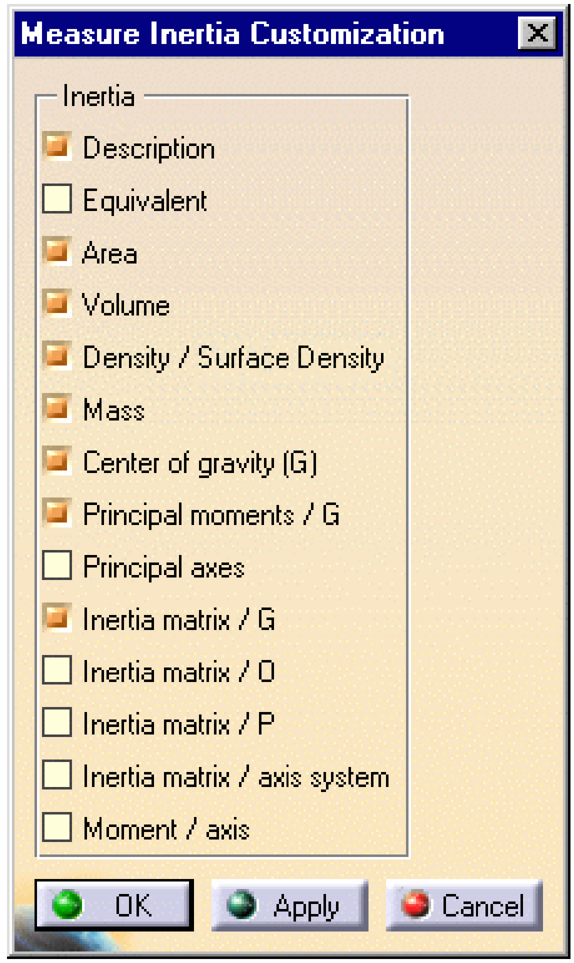


Some measures kept as features are associative and can be used as parameters.

You can write a macro script to automate your task. See *Space Analysis* on the Automation Documentation Home Page.

Customizing Your Measure

You can, at any time, define what will be computed and displayed in the Measure Inertia dialog box.



1. Click Customize... in the Measure Inertia dialog box.

   The Measure Inertia Customization dialog box opens.

   Note: The inertia properties checked here are also the properties exported to a text file.
2. Click the appropriate options to compute and display in appropriate tabs of the Measure Inertia dialog box the:
   - Inertia equivalents
   - Principal axes
   - Inertia matrix with respect to the origin O
   - Inertia matrix with respect to a point P
   - Inertia matrix with respect to an axis system
   - Moment of inertia about an axis
3. Click **Apply** or **OK** in the Measure Inertia Customization dialog box when done.

Measuring 2D Inertia

This task explains how to measure the inertia properties of plane 2D surfaces.

You can measure the area, center of gravity, principal moments, inertia matrix as well as the principal axes.
You can measure the inertia properties of plane surfaces including DMU sections. The area of the surface is also calculated.

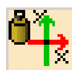
Note: You cannot measure inertia properties of either wireframe or infinite elements.
To find out more about notations used.

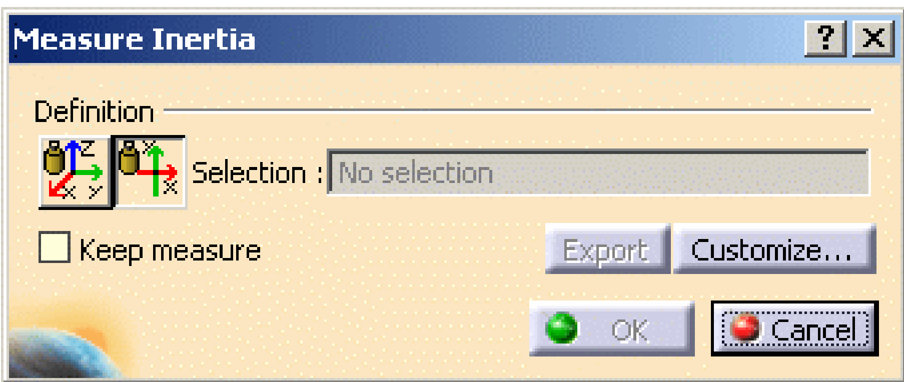
No sample document provided.

1. Click the Measure Inertia  icon.

In DMU, you can also select **Analyze -> Measure Inertia** from the menu bar.

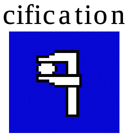
The Measure Inertia dialog box appears.

2. Click the Measure 2D Inertia  icon.



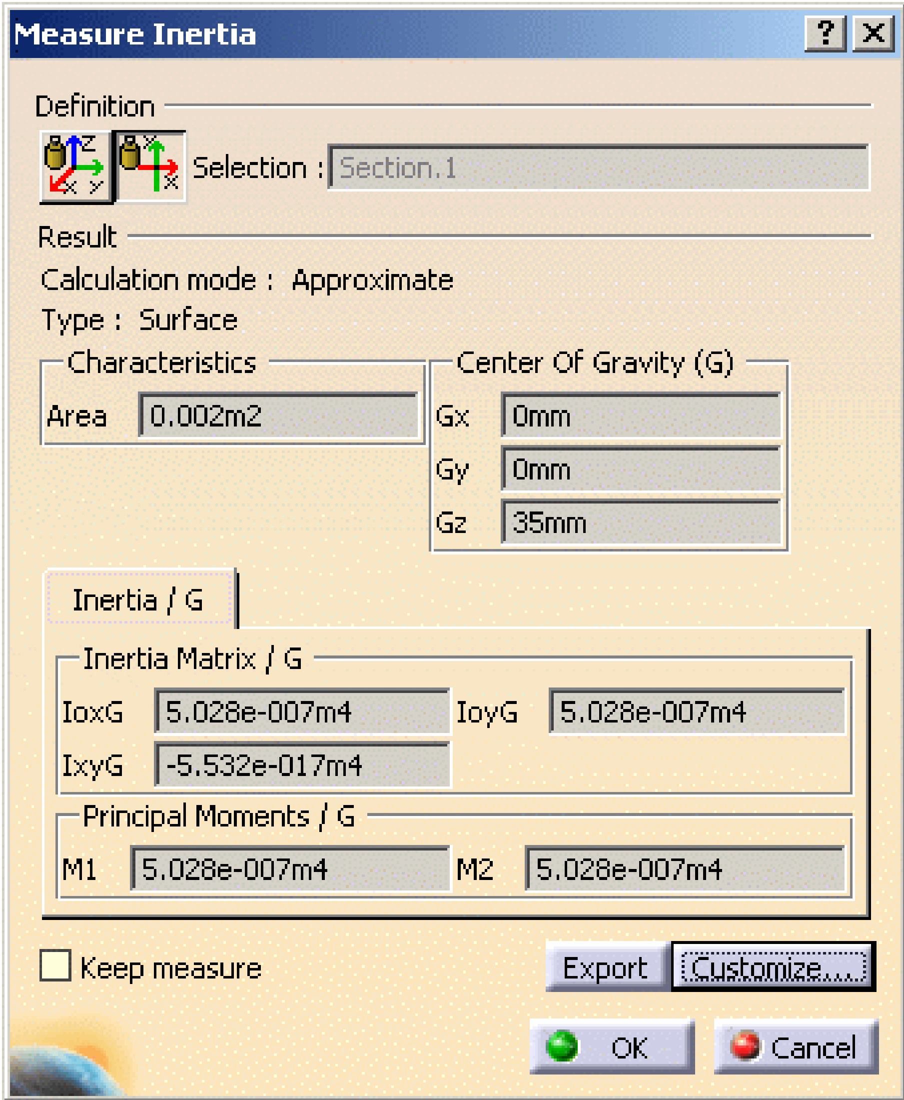
Dialog box options

- A Keep Measure option in the dialog box lets you keep current and subsequent measures as features. Some measures kept as features are associative and can be used as parameters.
  Note: This option is not available in the Drafting workbench.
- An Export option lets you write results to a text file.
- A Customize... option lets you define what will be computed and displayed in the dialog box.

When you move the cursor over the geometry or specification tree, its appearance changes to reflect the measure command you are in. 

3. Click to select a plane 2D surface in the geometry area or the specification tree.

The **Dialog Box** expands to display the results for the selected item.



The dialog box identifies the selected item, in our case a DMU section, and indicates whether the calculation is exact or approximate:

- In Design mode, measures access exact data and wherever possible true values are given. Note that it is possible to obtain an exact measure for most items in design mode.
- In Visualization mode, measures are made on tessellated items and approximate values are given.

In addition to the center of gravity G, the principal moments of inertia M and the matrix of inertia, the dialog box also gives the area of the selected item.

The center of gravity G is computed with respect to the document axis system. The matrix of inertia is expressed in an axis system whose origin is the center of gravity and whose vectors are the axes of inertia.

Note: The matrix of inertia and the principal moments do not take density into account.

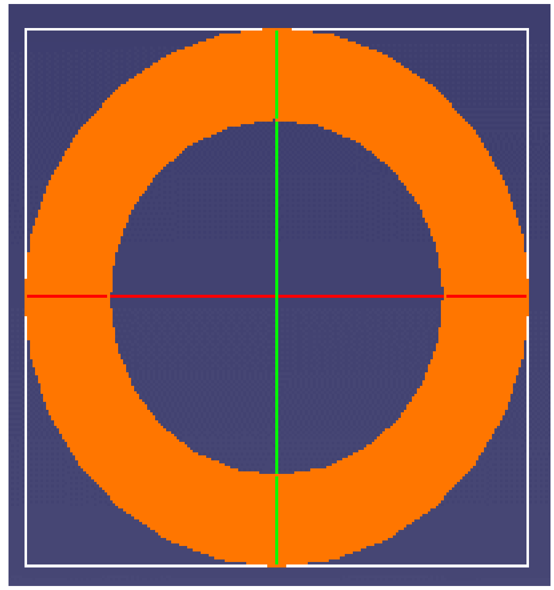
You can also compute and display the principal axes A. To do so, you must first activate the appropriate option in the Measure Inertia Customization dialog box.

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)**.

In the Geometry Area, the axes of inertia are highlighted and a bounding box parallel to the axes and bounding the selected item also appears.

Color coding of axes:

- Red: axis corresponding to the first moment M1
- Green: axis corresponding to the second moment M2



4. Click **OK** in the Measure Inertia dialog box.

If you checked the Keep Measure option in the Measure Inertia dialog box, your measures are kept as features.

Customizing Your Measure

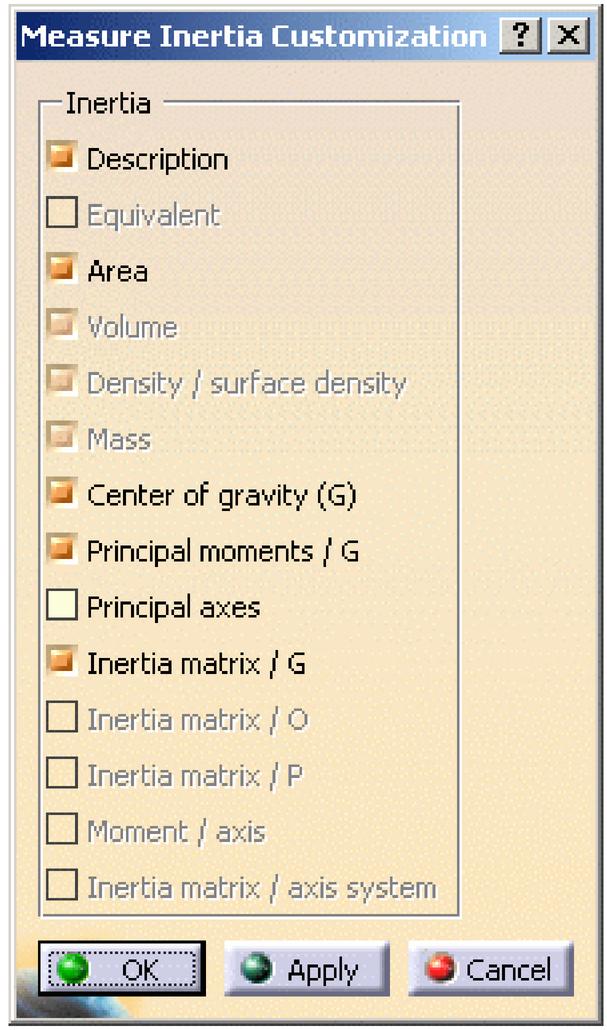
You can, at any time, define what will be computed and displayed in the tabs of the Measure Inertia dialog box.

When measuring 2D plane surfaces, in addition to the properties computed by default, you can compute and display the principal axes.

1. Click Customize... in the Measure Inertia dialog box.

   The Measure Inertia Customization dialog box opens.

   Note: The inertia properties checked here are also the properties exported to a text file.



2. Click the appropriate options:
   - Principal axes

3. Click **Apply** or **OK** in the Measure Inertia Customization dialog box when done.

Exporting Measure Inertia Results

This task shows you how to export both 3D and 2D inertia results to a text file.

Insert the Body1.cgr and the Body2.cgr documents from the common functionalities samples folder.

1. Select the root product and click the Measure Inertia icon.

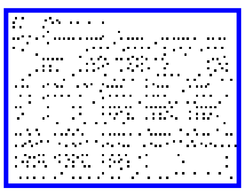
   The dialog box expands to display the results for the selected item.
2. Click Export to write the results to a text (*.txt) file.

   Important: Results shown in the Measure Inertia dialog box only are exported. Exported results are given in current units.

3. Identify the file name and location in the Export Results dialog box that appears, then click **Save**.

   Note: The examples given below concern 3D inertia results.

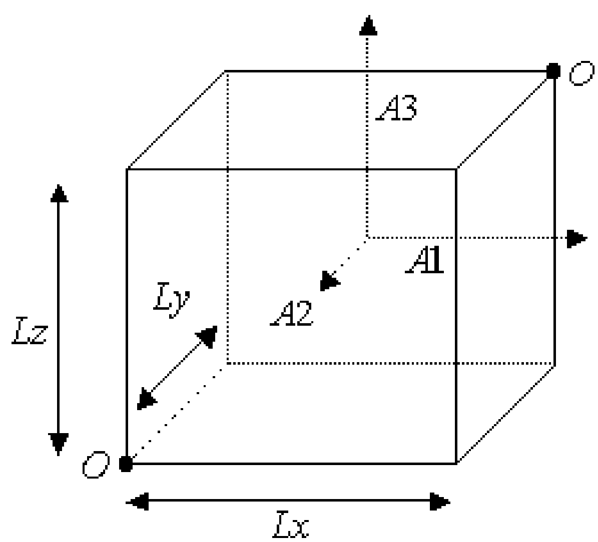
Note: If an assembly comprises sub-products or a part comprises part bodies, individual results for all sub-products or part bodies are also exported and written to the text file.



If the principal axes A are exported, bounding box values are also exported.

BBOx[mm]	BBOy[mm]	BBOz[mm]
0.143784	-0.0144969	-0.123277
0.16293	-0.0170551	-0.0783764
0.161982	-0.00353007	-0.0793372
0.169575	-0.0167384	-0.107493

BBLx[mm]	BBLy[mm]	BBLz[mm]
0.130313	0.0383596	0.0289959
0.166176	0.089425	0.0207404
0.166088	0.0894922	0.0206928
0.188821	0.0908866	0.0337512



$$O' = O + \vec{A1}.Lx + \vec{A2}.Ly + \vec{A3}.Lz$$

where BBOx,y,z defines the origin and BBLx,y,z the length along the corresponding axis.

Note: When importing the text file into an Excel spreadsheet, do not forget to identify the pipe character (|) used as separator in the Text Import Wizard dialog box.

Notations Used for Inertia Matrices

This section will help you read the information given in the Measure Inertia dialog box for Inertia Matrix / G, Inertia Matrix / O, Inertia Matrix / P and Inertia Matrix / Axis System A.

Moments and Products of 3D Inertia

Iox	Moment of inertia of the object about the ox axis:	$Iox = \int_M (y^2 + z^2) dM$
Ioy	Moment of inertia of the object about the oy axis:	$Ioy = \int_M (x^2 + z^2) dM$
Ioz	Moment of inertia of the object about the oz axis:	$Ioz = \int_M (x^2 + y^2) dM$
Pxy	Product of inertia of the object about axes ox and oy:	$Pxy = \int_M (x.y) dM$
Pxz	Product of inertia of the object about axes ox and oz:	$Pxz = \int_M (x.z) dM$
Pyz	Product of inertia of the object about axes oy and oz:	$Pyz = \int_M (y.z) dM$

(where M is the mass of the object; units: $kg.m^2$)

Moments and Products of 2D Inertia

Iox	Moment of inertia of the surface about the ox axis:	$IoX = \int_A (y^2) dA$
Ioy	Moment of inertia of the surface about the oy axis:	$IoY = \int_A (x^2) dA$
Pxy	Product of inertia of the surface about axes ox and oy:	$Pxy = \int_A (x.y) dA$

(where A is the surface; units: m^4)

Matrix of Inertia

3D Inertia: 2D Inertia:

$$I = \begin{bmatrix} Iox & -Pxy & -Pxz \\ -Pxy & Ioy & -Pyz \\ -Pxz & -Pyz & Ioz \end{bmatrix} \qquad I = \begin{bmatrix} IoX & -Pxy \\ -Pxy & IoY \end{bmatrix}$$

where I is the matrix of inertia of the object with respect to orthonormal basis Oxyz

Expression in Any Axis System:

I is the matrix of inertia with respect to orthonormal basis Oxyz.

Huygen's theorem is used to transform the matrix of inertia: $Oxyz \rightarrow Pxyz$ (parallel axis theorem).

Let I' be the matrix of inertia with respect to orthonormal basis Pxyz

where $V = \overline{PO}$

$$I' = I + m.\begin{bmatrix} V_y^2 + V_z^2 & -V_x.V_y & -V_x.V_z \\ -V_x.V_y & V_x^2 + V_z^2 & -V_y.V_z \\ -V_x.V_z & -V_y.V_z & V_x^2 + V_y^2 \end{bmatrix}$$

M = {u,v,w}: transformation matrix from basis (Pxyz) to basis (Puvw)
TM is the transposed matrix of matrix M.

J is the matrix of inertia with respect to an orthonormal basis Puvw:

J = TM.I'.M

Additional Notation used in Measure Inertia command

Ixy = (-Pxy)

Ixz = (-Pxz)

Iyz = (-Pyz)

Note: Since entries for the opposite of the product are symmetrical, they are given only once in the dialog box.

IoxG	Moment of inertia of the object about the ox axis with respect to the system Gxyz, where G is the center of gravity.

IoxO	Moment of inertia of the object about the ox axis with respect to the system Oxyz, where O is the origin of the document.
IoxP	Moment of inertia of the object about the ox axis with respect to the system Pxyz, where P is a selected point.
IoxA	Moment of inertia of the object about the ox axis with respect to the system Axyz, where A is a selected axis system.

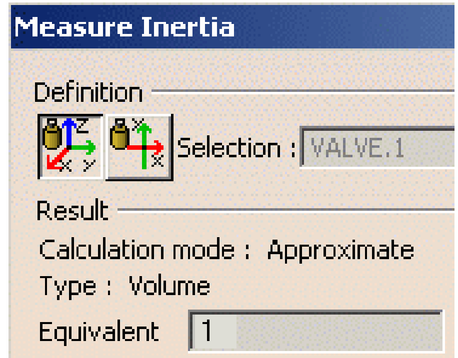
etc.

Inertia Equivalents

If your document contains inertia equivalents set using Knowledgeware capabilities, then the Inertia command will not calculate the inertia properties of the selected geometry but return the equivalent values.

The Equivalent box of the Measure Inertia dialog box indicates whether or not equivalents have been used:

- 0: the measure is made on the selection, geometry or assembly
- 1 or more: One or more inertia equivalents are taken into account.



To display inertia equivalents in the Measure Inertia dialog box:

1. Click Customize... in the Measure Inertia dialog box.
   The Measure Inertia Customization dialog box appears.
2. Check Equivalent in the Measure Inertia Customization dialog box.
3. Click **Apply**.

Equivalents are user parameters set using the Knowledgeware formula command under parts or products and imported from text (*txt) or Excel (*xls) files. Sets of equivalent parameters must be valid to be taken into account. To be valid, all the properties shown in the example below must be listed.

An example of a text file follows. In text files, the name of the property and the value are separated by a tab stop.

```
Equivalent_IsSurface	false
Equivalent_IsVolume	true
Equivalent_Area	6m2
Equivalent_Volume	1m3
Equivalent_Mass	1000kg
Equivalent_COGx	75mm
Equivalent_COGy	-10mm
Equivalent_COGz	-25mm

Equivalent_MatGxx	50000gxmm2
Equivalent_MatGyy	50000gxmm2
Equivalent_MatGzz	50000gxmm2
Equivalent_MatGxy	0gxmm2
Equivalent_MatGxz	0gxmm2
Equivalent_MatGyz	0gxmm2
```

In Excel files, simply list property names and values in two separate columns.

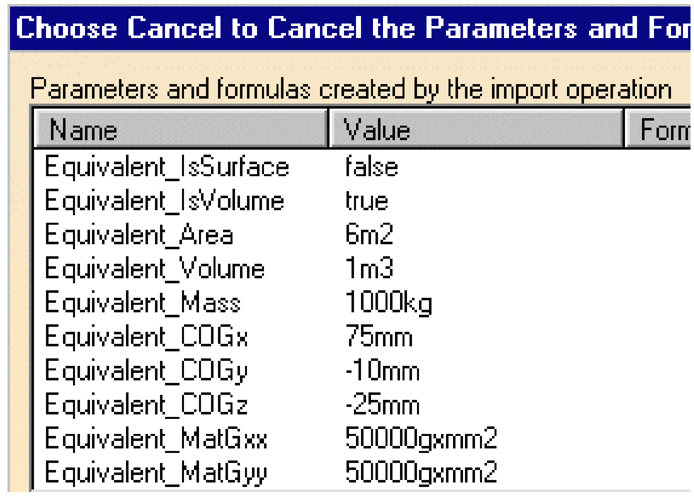
Importing Inertia Equivalents

1. Select the product to which you want to associate inertia equivalents.
2. Click the formula icon.

3. Click Import... in the Formulas dialog box.

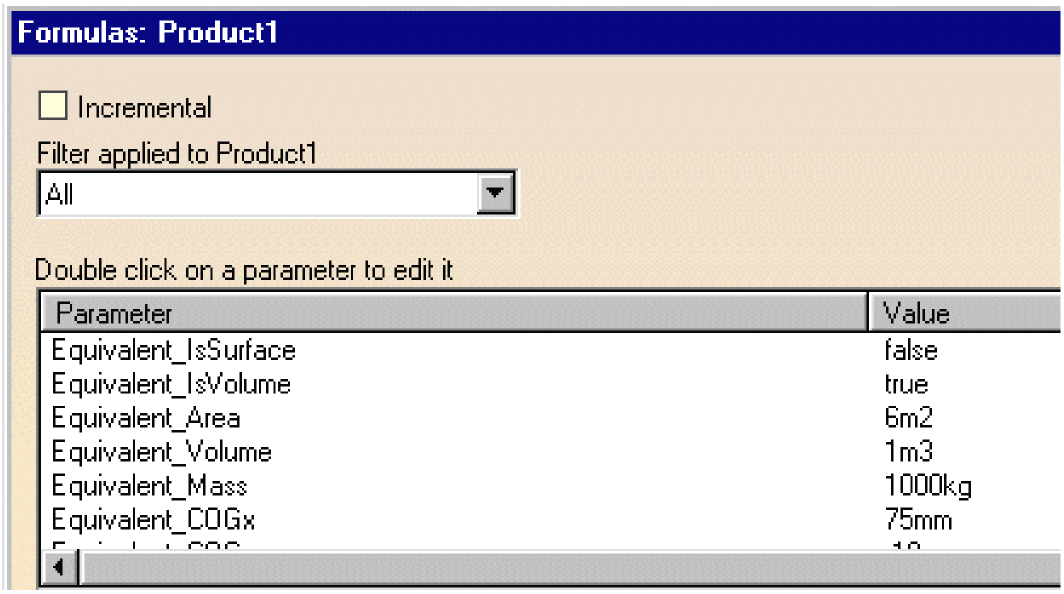
4. Select the text or Excel file containing the inertia equivalents in the file selection dialog box, then click Open.

   Parameters to be imported are listed.



5. Click **OK** to import all the parameters listed into the document.

   Imported parameters are now displayed in the Formulas dialog box.



6. Click **OK** in the Formulas dialog box.

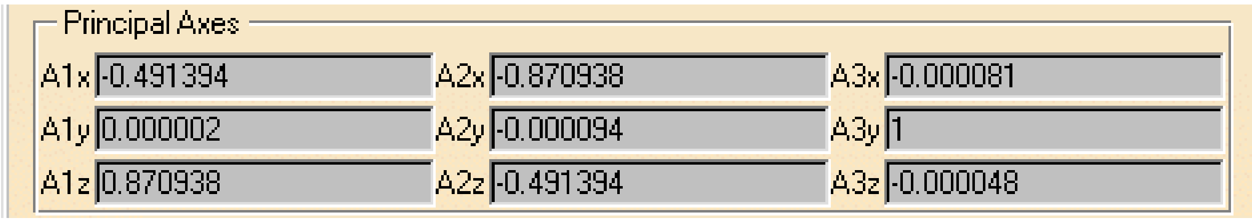
   You are now ready to run your inertia calculation.

- Having imported inertia equivalents, you no longer need the representations of the product or sub-products and you can de-activate them (**Edit ->Representations**). De-activated representations are unloaded. This frees the geometry area and improves system response time.
- To display parameters in the specification tree, select the Parameters checkbox below Display in Specification Tree in the Display tab of the Options dialog box (**Tools-> Options-> Infrastructure-> Part Infrastructure**).

Measuring the Principal Axes A about which Inertia is Calculated

1. In the Measure Inertia Customization dialog box, click Principal axes.
2. Click **Apply**.

   The Inertia / G tab in the Measure Inertia dialog box becomes available.
3. Click the Inertia / G tab to display the principal axes about which inertia is calculated.



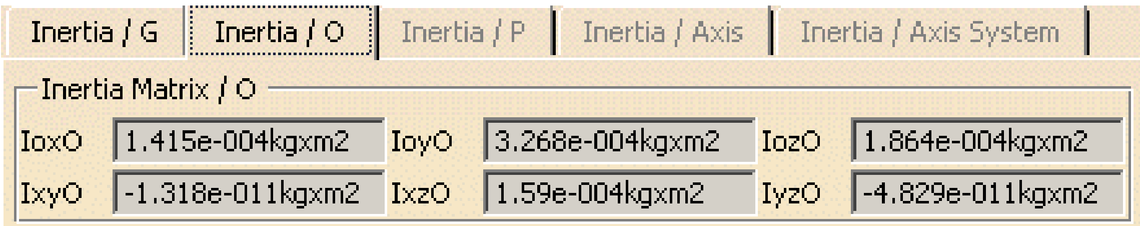
Note: If you checked the Keep Measure option, bounding box values are also displayed in the specification tree.

You can create the axis system corresponding to the principal axes.

Measuring the Inertia Matrix with respect to the Origin O of the Document

1. In the Measure Inertia Customization dialog box, click Inertia matrix / O.
2. Click **Apply**.

   The Inertia / O tab in the Measure Inertia dialog box becomes available. Entries for the inertia matrix appear in the specification tree.
3. Click the Inertia / O tab to display the inertia matrix of selected items with respect to the origin O of the document.



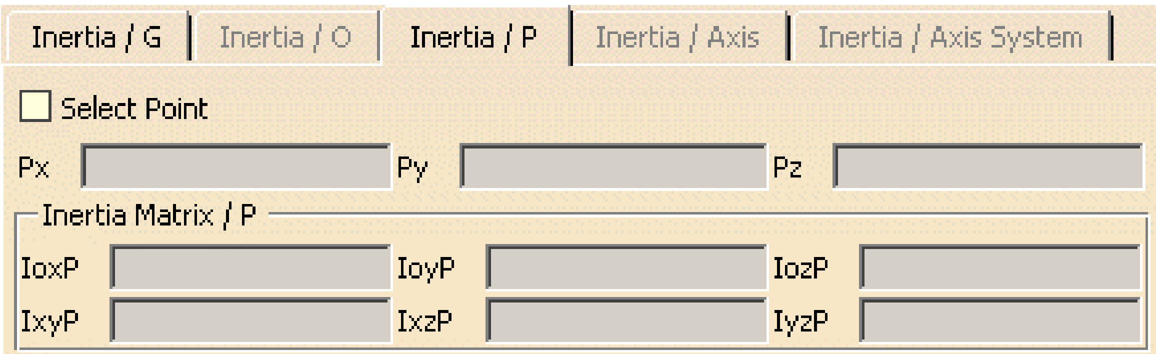
Measuring the Inertia Matrix with respect to a Point P

Insert or open the InertiaVolume.CATPart from the common functionalities sample folder **cfysm/samples**.

1. In the Measure Inertia Customization dialog box, click Inertia matrix / P.

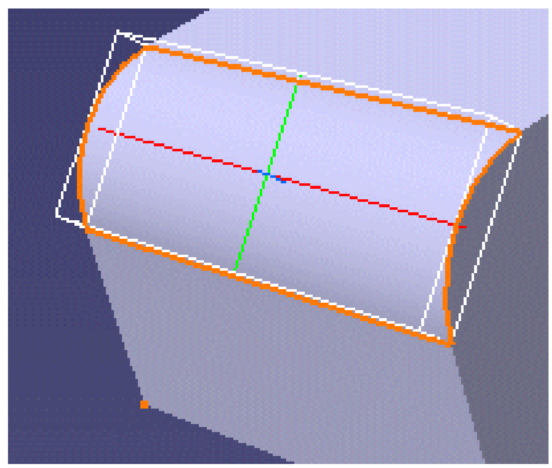
   Note: Only points created in the Part Design workbench are valid.
2. Click **Apply**.

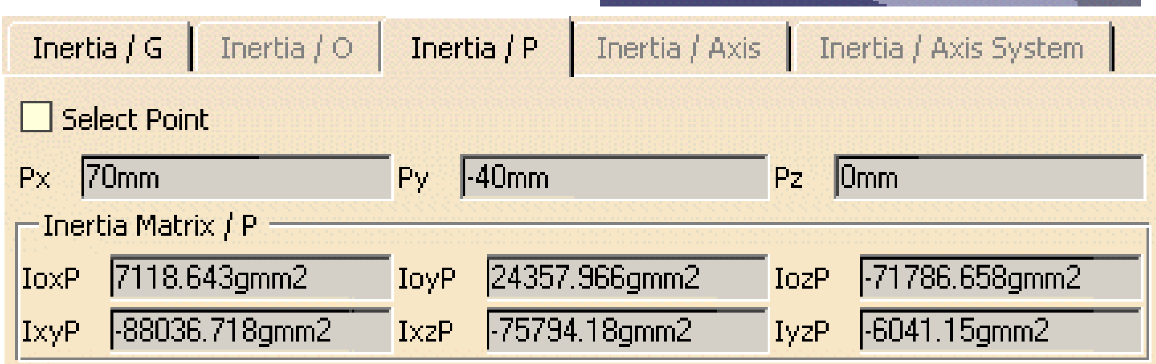
   The Inertia / P tab in the Measure Inertia dialog box becomes available.
3. Click the Inertia / P tab.



4. Select the Select point checkbox.
5. Select a point in the geometry area:

   The coordinates of the point and the inertia matrix are given in the dialog box.





Selecting another item calculates the inertia matrix of the selected item with respect to the same point. To change point, click the Select point checkbox again, then select another point.

Measuring the Matrix of Inertia with respect to an Axis System

Insert or open the InertiaVolume.CATPart from the common functionalities sample folder **cfysm/samples**.

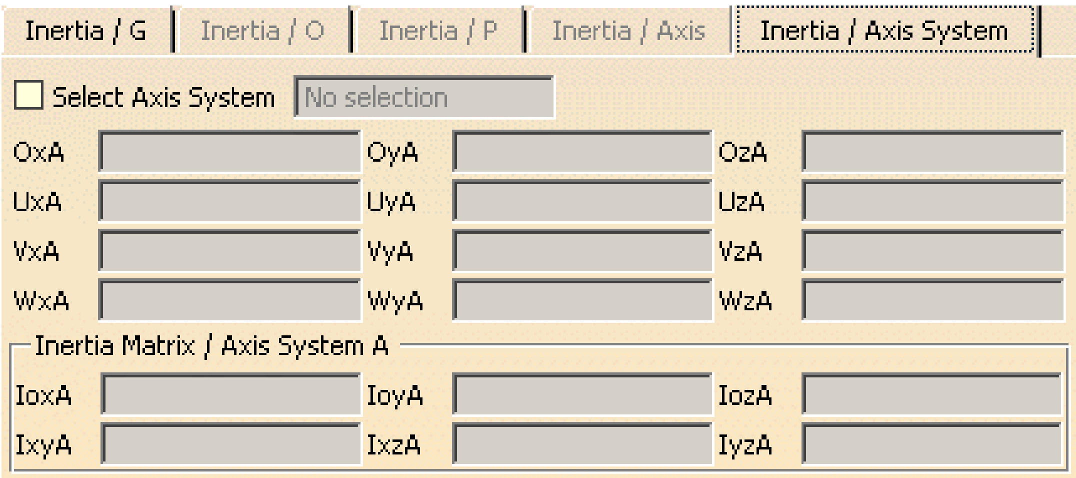
1. In the Measure Inertia Customization dialog box, click Inertia matrix / axis system.

   Note: Only axis systems created in the Part Design workbench (Axis System command) are valid.

2. Click **Apply**.

   The Inertia / Axis System tab in the Measure Inertia dialog box becomes available.

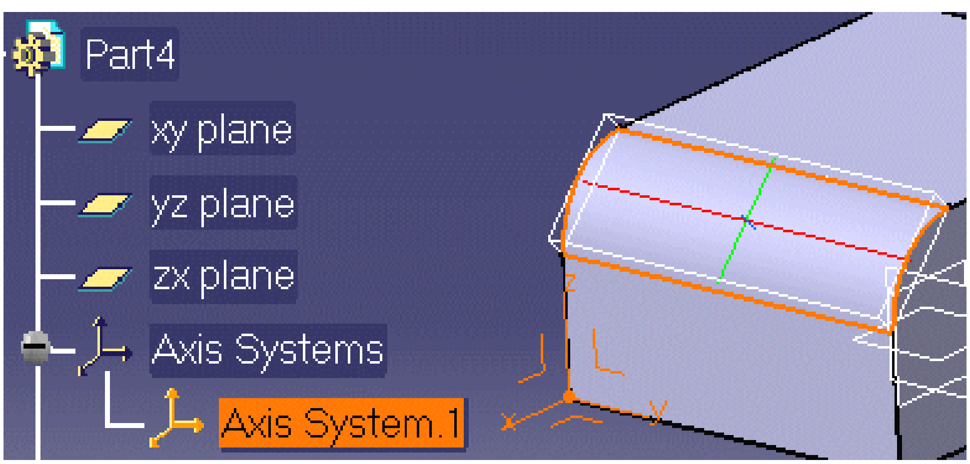
3. Click the Inertia / Axis System tab.
4. Select the Select axis system checkbox.

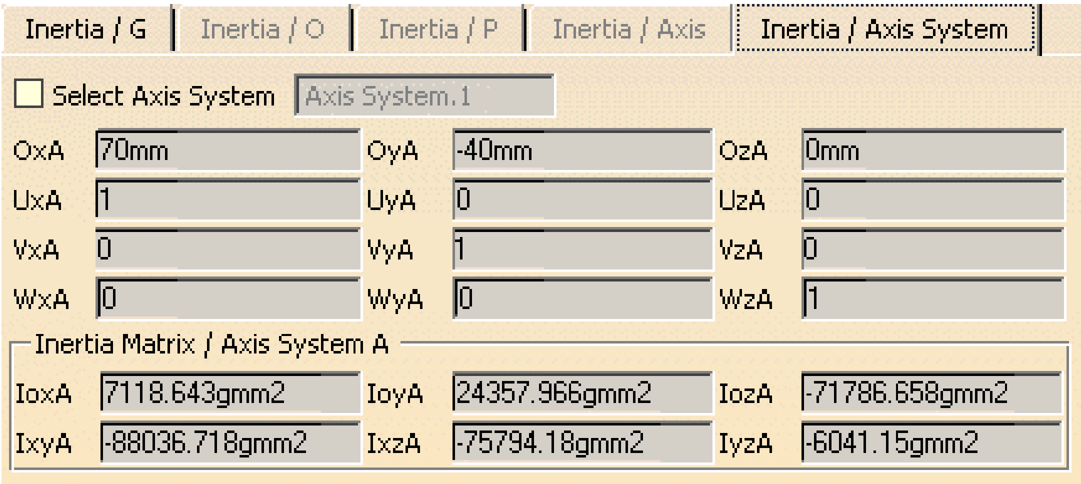


5. Select an axis system in the specification tree:

   Note: You must select the axis system in the specification tree.

   The name of the axis system as well as the origin O, (U, V, W) -vectors and the matrix of inertia with respect to the axis system are given in the dialog box. Entries for the matrix of inertia appear in the specification tree.





Selecting another item measure inertia properties of the selected item with respect to the same axis system. To change axis system, click the Select axis system checkbox again, then select another axis system.

If you checked the Keep Measure option in the Measure Inertia dialog box, your matrix of inertia measures are kept as features and, if made with respect to a V5 axis system, are associative.

Measuring the Moment of Inertia about an Axis

Insert or open the InertiaVolume.CATPart from the common functionalities sample folder **cfysm/samples**.

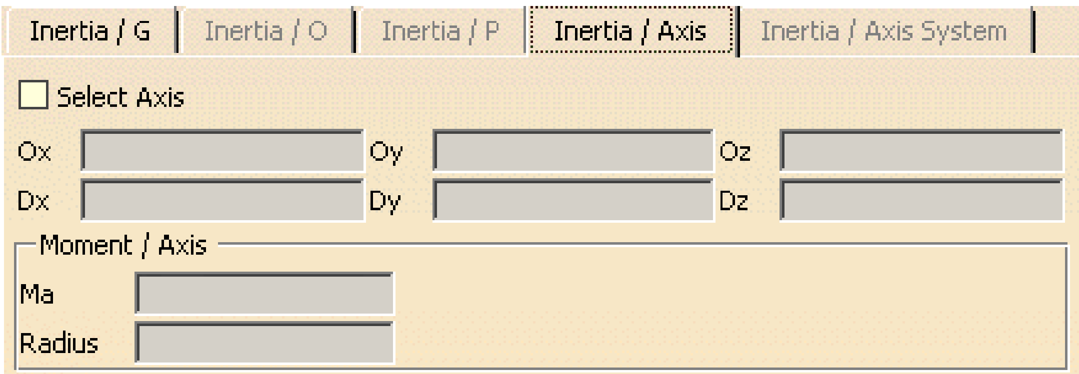
1. In the Measure Inertia Customization dialog box, click Moment / axis to measure inertia with respect to an axis.

   Note: Only axes created in the Part Design workbench are valid.

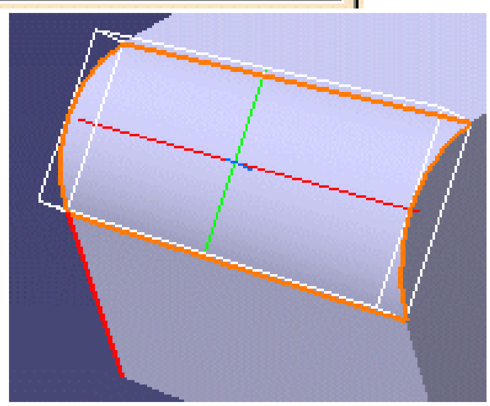
2. Click **Apply**.

   The Inertia / Axis tab in the Measure Inertia dialog box becomes available.

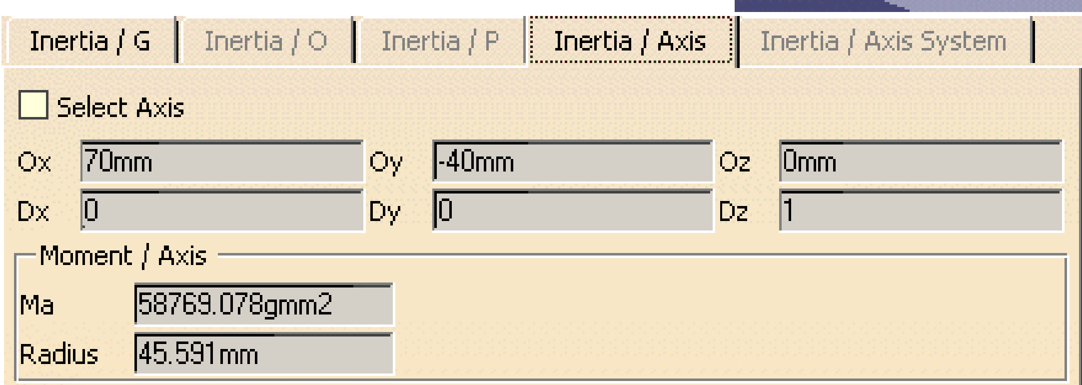
3. Click the Inertia / Axis tab.
4. Select the Select axis checkbox.



5. Select an axis in the geometry area:



   The equation and direction vector of the axis as well as the moment of inertia Ma about the axis and the radius of gyration are given in the dialog box.

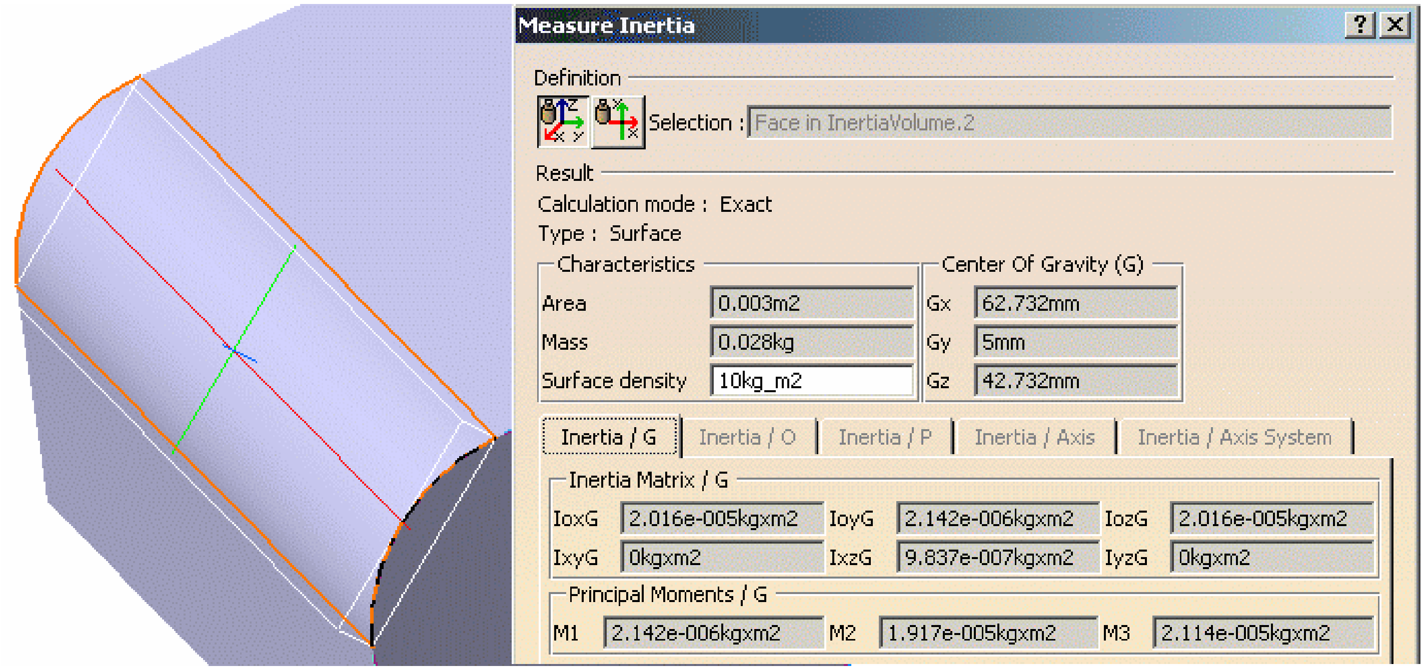


Selecting another item measures the inertia of the selected item about the same axis. To change axis, click the Select axis checkbox again, then select another axis.

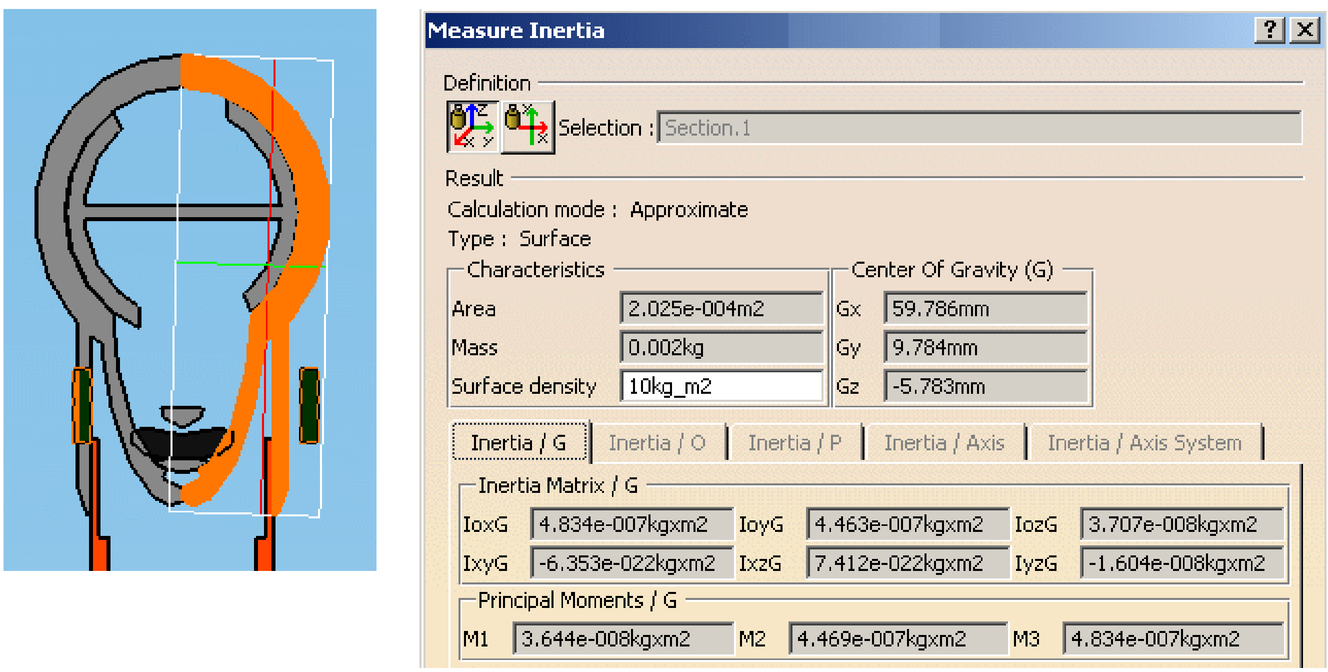
3D Inertia Properties of a Surface

You can measure 3D inertia properties on exact and tessellated surfaces. Examples showing a surface and a DMU section are given below.

Insert or open the InertiaVolume.CATPart from the common functionalities sample folder **cfysm/samples**.



The DMU section is a tessellated surface.

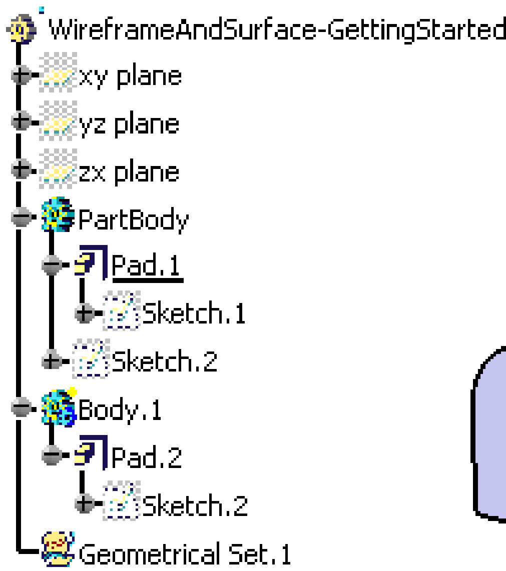


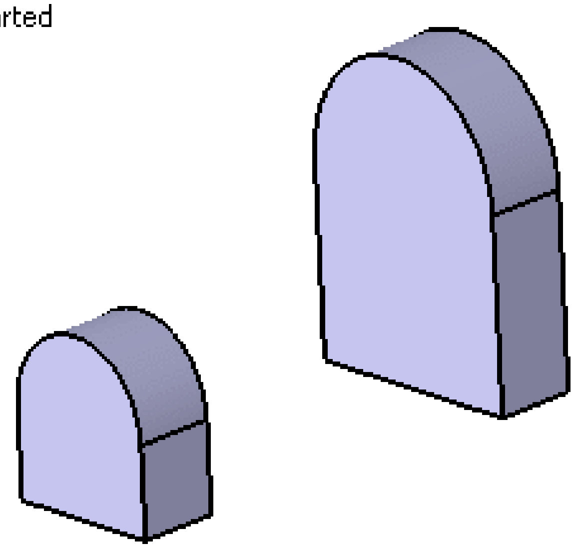
Using Hybrid Parts

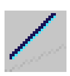
This task shows how to create a hybrid part comprising wireframe, surface and solid geometry.

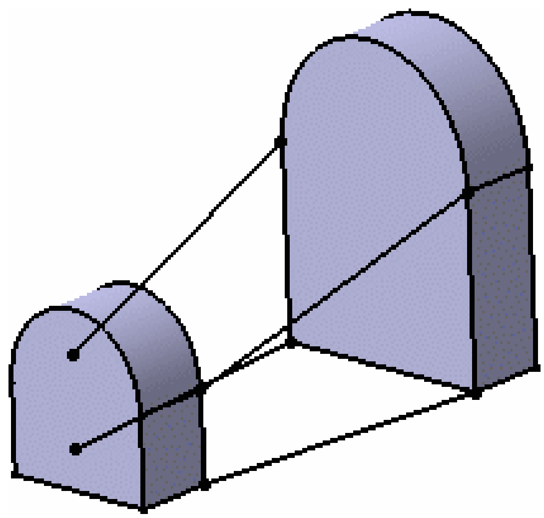
You must have access to the Part Design product.
Open the Hybrid1.CATPart document.

1. In the Generative Shape Design workbench, open a document comprising solid entities.



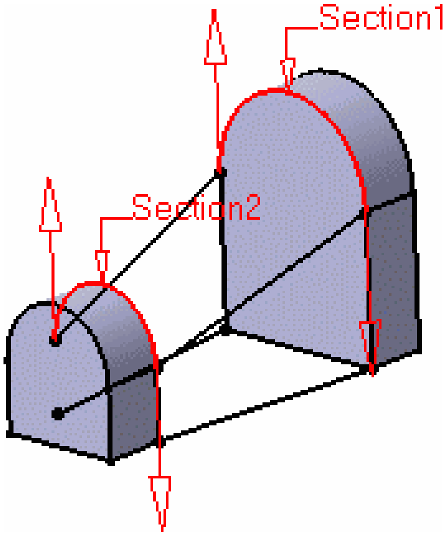


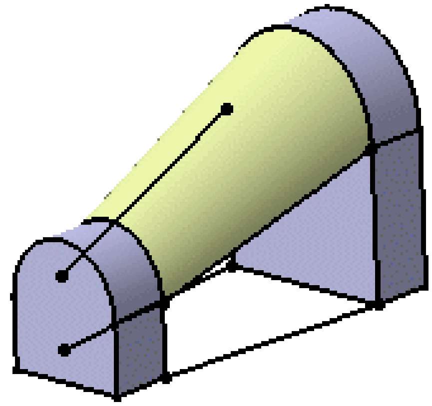
2. Click the **Line** icon  then create construction point-point lines between the opposite vertices of the two pads.



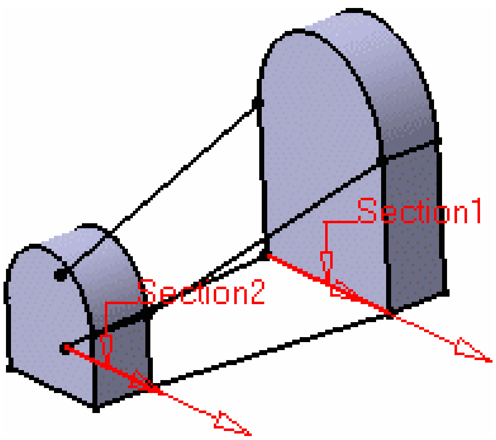
These lines are created in a Geometrical Set entity.

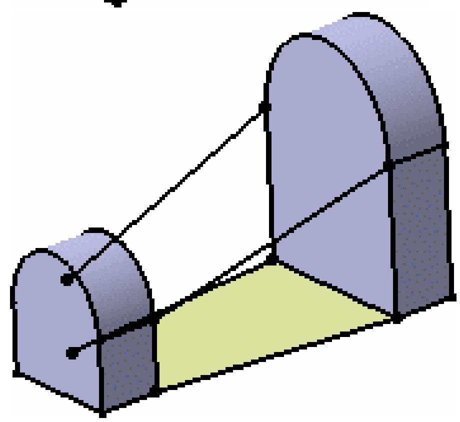
3. Click the **Multi-sections surface** icon  and create a lofted surface between the curved edges of the two pads.



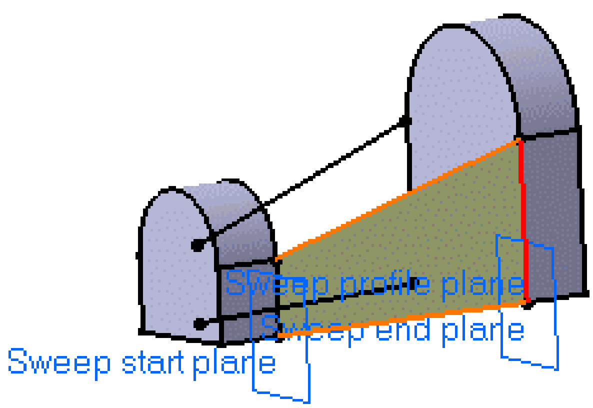


4. Create another lofted surface between the bottom edges of the two pads.

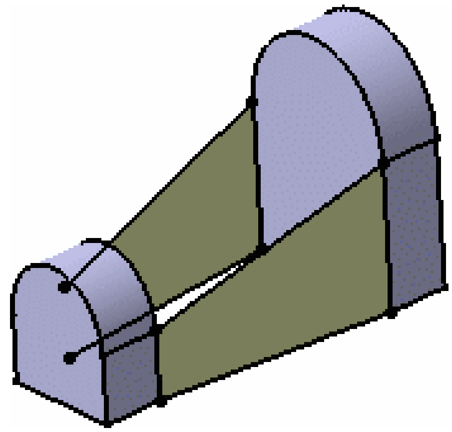




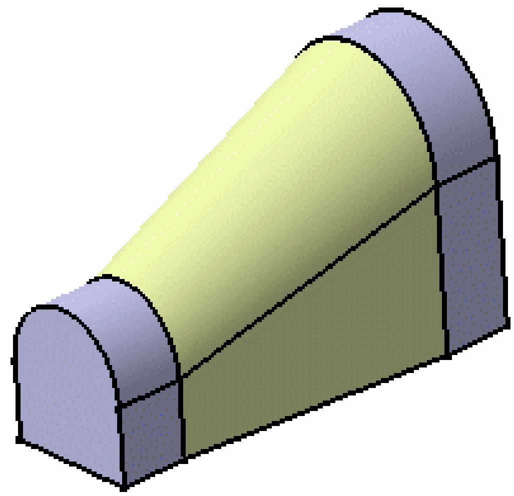
5. Click the **Sweep** icon  and create a swept surface between two opposite vertical edges of the two pads.

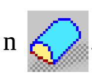


6. Create another swept surface on the other side of the side of the two pads.

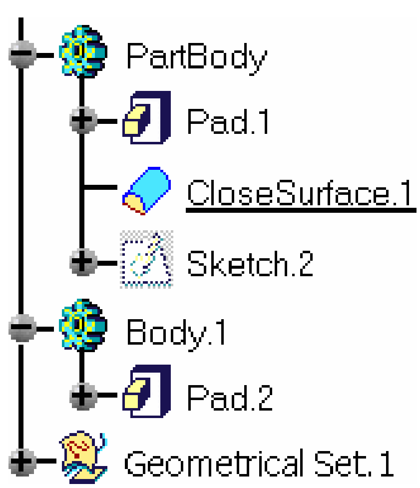


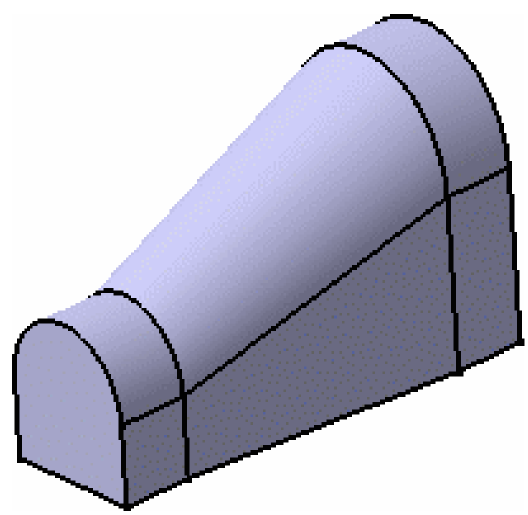
7. Click the Join icon  then select the four surfaces to create a single joined surface.



8. Open the Part Design workbench and select the Closed Surface icon .

9. Select the joined surface in order to close it.





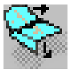
The model and specification is updated with the Close Surface feature.

Working with the Generative Shape Optimizer Workbench

 Create bumped surfaces: select a surface, a limit curve, the deformation center, direction, and value.

 Deform surfaces based on curve wrapping: select the surface to be deformed then matching pairs or reference and target curves.

 Deform surfaces based on surface wrapping: select the surface to be deformed, the reference surface then the target surface.

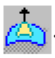
 Deform surfaces based on shape morphing: select the surface to be deformed and deformation elements

Creating Bumped Surfaces

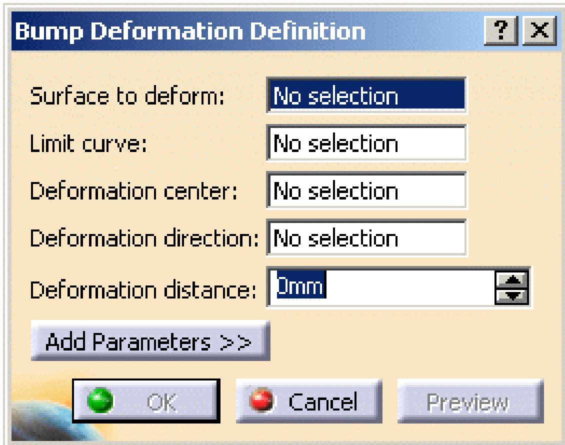
This command is only available with the Generative Shape Optimizer product.

This task shows how to create bumped surfaces, by deformation of an initial surface.

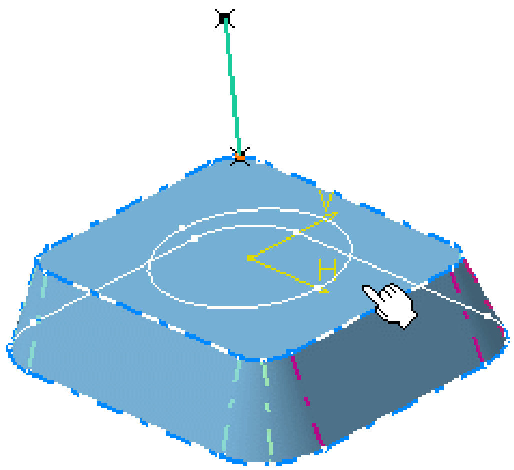
Open the Bump1.CATPart document.

1. Click the **Bump** icon .

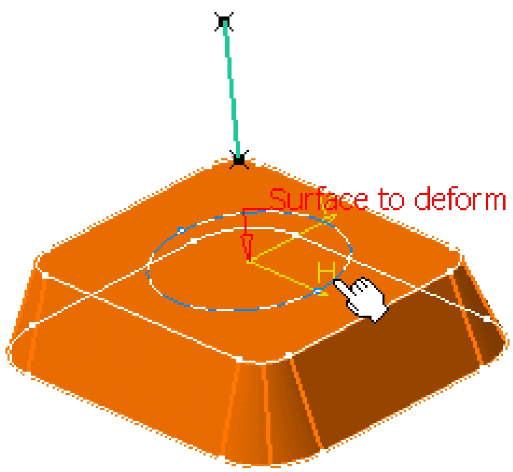
   The Bump Deformation Definition dialog box is displayed.



2. Select the surface to be deformed.

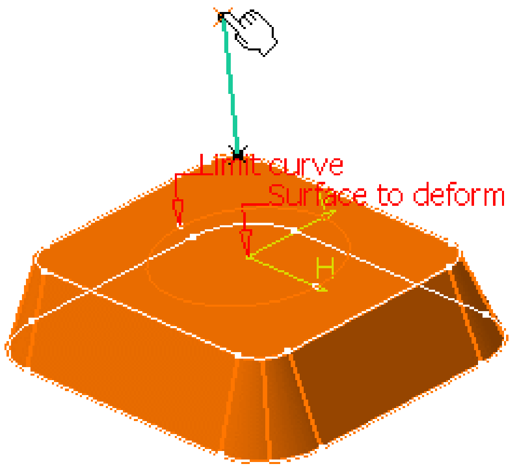


3. Select the **Limit curve**, that is the curve delimiting the deformation area.



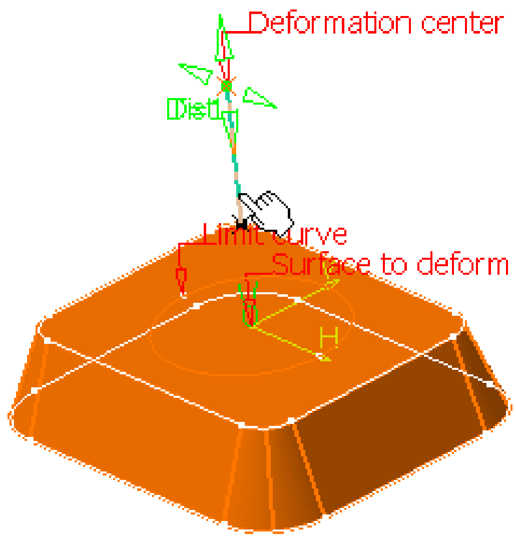
The limit curve needs to be lying on the surface to be deformed. If not, use the **Create Projection** contextual menu on the **Limit curve** field to project the limit curve on the surface.

4. Select the **Deformation center**, that is the point representing the center of the deformation.



The deviation will be at its maximum at this point, and evolve towards the limit curve, where it should reach 0.

5. Select the curve indicating the **Deformation direction**.



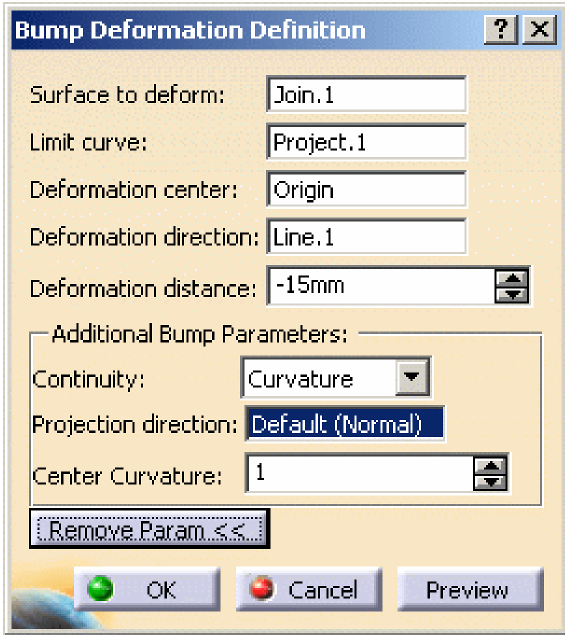
The deformation is propagated along this direction. By default, the **Deformation direction** is normal to the deformed element.

6. Set the Deformation distance, that is the maximum distance, along the **Deformation direction**, from the deformed surface towards the **Deformation Center**.

   We keyed in 20mm.

7. Click Preview to preview the bumped surface.

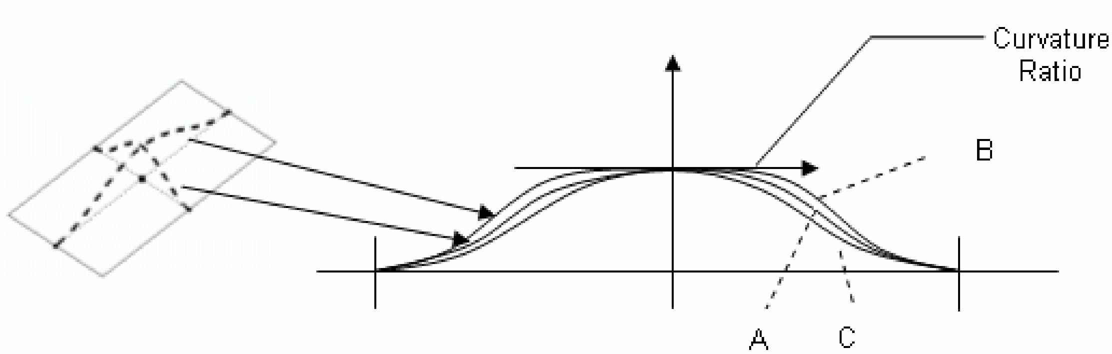
8. Click the **Add Param>>** button to display further options.



You can:

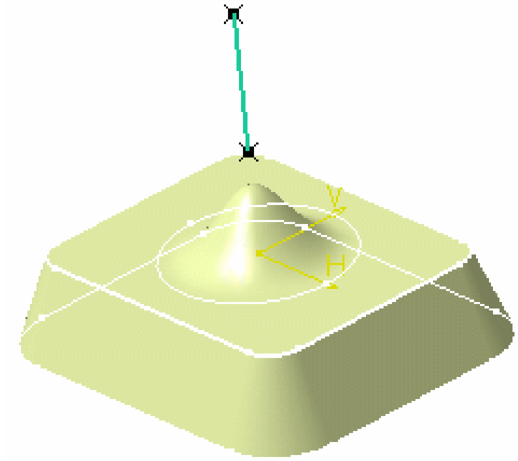
- define the continuity to be kept between the deformed area and the surface outside the deformation area (point, tangent, or curvature continuity)
- specify a projection direction if the **Deformation Center** does not lie within the selected surface to be deformed, so that it is projected onto it.
- define a center curvature value to control the shape of the bump deformation.
  If the value is:
  - equal to 1 (case A), the shape is the default one (as if no value was defined)
  - smaller than 1 (case B), the shape is flatter
  - bigger than 1 (case C), the shape is steeper

  Note: all values are allowed (positive, null, and negative values), however it is advised to define a value comprised between -1 and 5.



In case of a curvature continuity, the original continuity between the deformed area and the surface outside the deformation area will be at best kept but may be approximate in certain cases.

9. Click **OK** to validate the surface deformation.



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

You can edit the bump's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

Deforming Surfaces According to Curve Wrapping

This command is only available with the Generative Shape Optimizer product.

This task shows how to deform surfaces basing the deformation on curve wrapping, that is matching each reference curve onto a target curve. The deformation is then defined by the transformation of the reference curves into target curves.
The curves used for the deformation do not necessarily lie on the initial surface.
Several cases are presented here, from the simplest one to cases using various options. Note that whatever information is given in the first example also applies to the following examples.

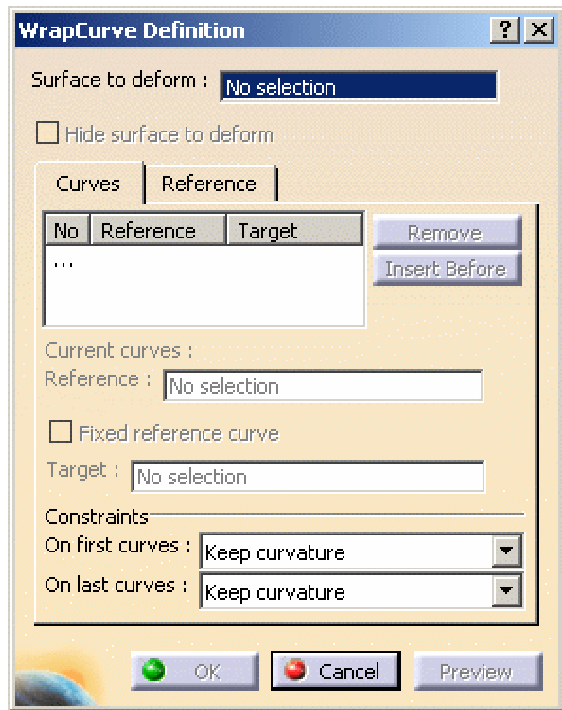
- Basic curve wrapping deformation
- Curve wrapping deformation with a fixed reference
- Editing a deformed surface
- How is the deformation computed ?

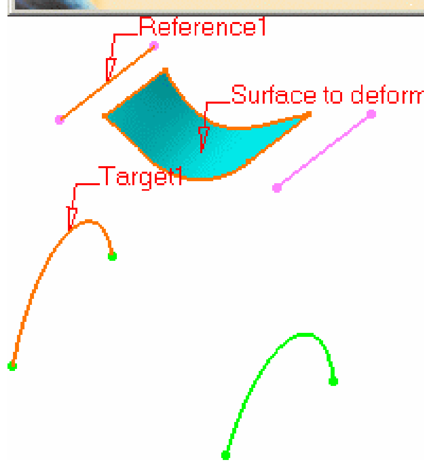
Open the WrapCurve1.CATPart document.

Basic Curve Wrapping Deformation

1. Click the **Wrap Curve** icon .

The WrapCurve Definition dialog box is displayed.

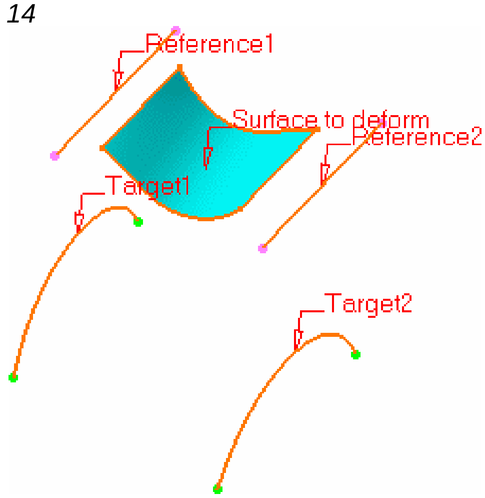




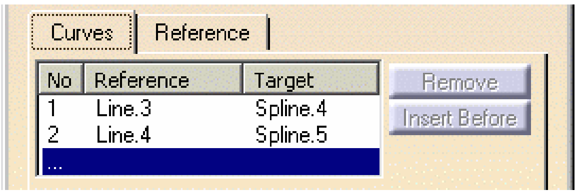
2. Select the surface to be deformed.

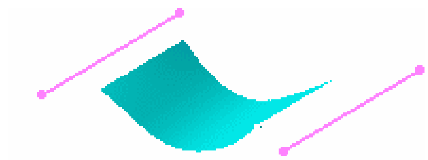
3. Successively select the first reference curve and the first target curve.

4. Repeat this operation by selecting the second reference curve then the second target curve.



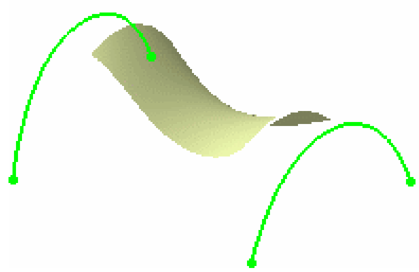
As you select pairs of reference/target curves, the curves list in dialog box is updated accordingly.





5. Click **OK** to create the deformed surface.

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



- You must always select successively a reference then a target curve to define a pair. You cannot select all reference curves, then all target curves for example.
- You need to select only one pair of curves (reference and target) to be able to define the deformation by clicking Preview.
- When several pairs of curves are selected, they must be ordered, not randomly selected
- Reference curves should not intersect each other, nor should the target curves should intersect each other

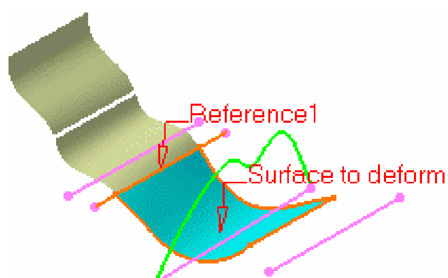
Reference and target curves can be multi-cells. Joined, blended, or matched curves, for example, can be used as reference or target curves.

Curve Wrapping Deformation with a Fixed Reference

Some times you need to create a deformed surface in relation to another element, when you want to match two surfaces for example. The curve wrapping capability lets you fix an element that can be used by another one, thus allowing you to retain a connection between elements while deforming the initial surface.

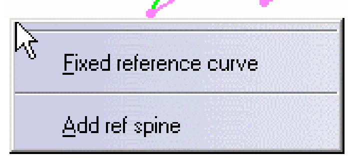
1. Click the **Wrap Curve** icon .

The Wrap Curve dialog box is displayed.



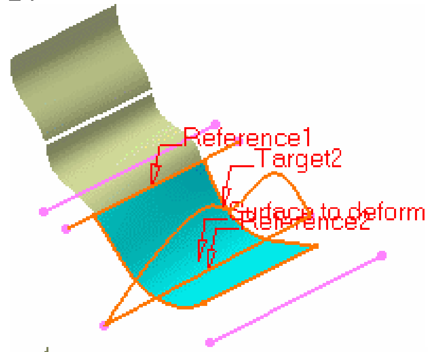
2. Successively select the surface to be deformed, and the first reference curve.

3. Right-click in space to display the contextual menu, and choose the Fixed reference curve.



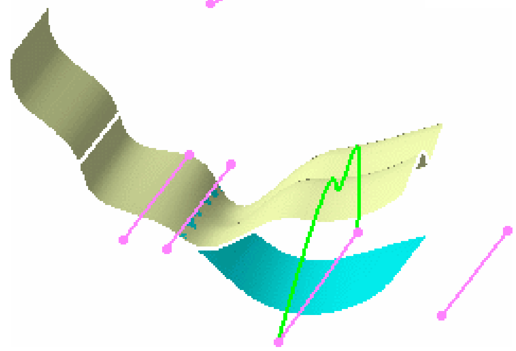
The reference curve you selected previously now is fixed, i.e. you do not need a target curve, this curve being used to create the deformation.

In the target area of the list, no element is displayed.



4. Select another pair of reference and target curves and click Preview.

A new surface is created based on the first reference curve and the second target curve.



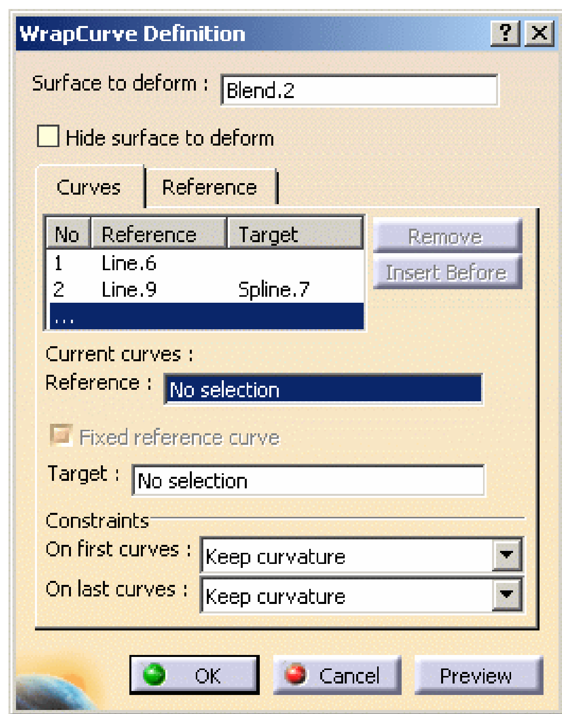
5. Click **OK** to create the deformed surface.

Because the first reference curve is an element used by the blended surface the connection between the two surfaces is retained.

Editing a Deformed Surface

Using the deformed surface you just created, this section shows you how to modify it by:

- inserting curve pairs
- remove curve pairs
- fix reference curves
- add constraints onto the first and last curve pairs

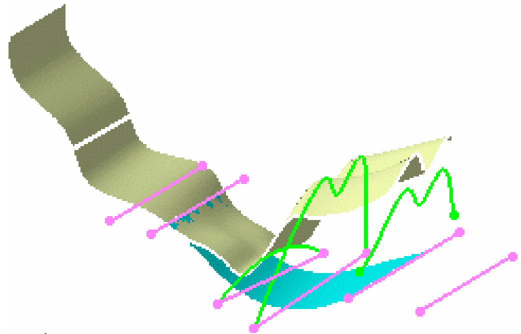


1. Double-click on the wrap curve surface you just created.

The dialog box is displayed containing the creation information.

2. Within the list, select the second line (Reference: Line.9, Target: Spline.7) and click the **Insert Before** button.

The **Reference** field of the **Current curves** area gets active.

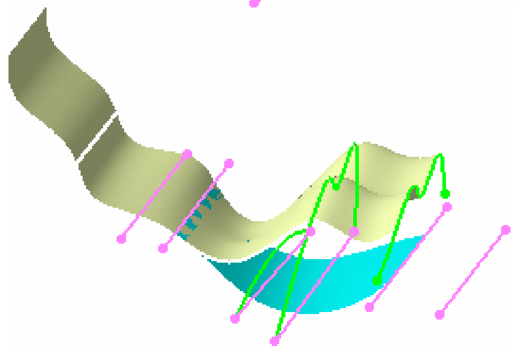


3. Select a new reference curve (Line.8) and a new target curve (Spline.6), and click Preview.

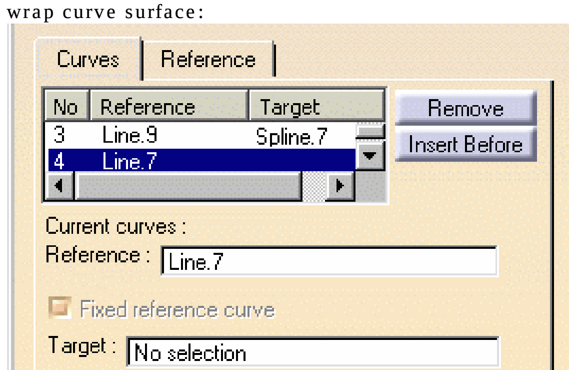
The deformed surface now takes into account the new pair of curves.

To add a pair of curves as the last entry in the list, you need to select the ... line, and directly select the reference and target curves.

With our example, we selected the ... line, then selected Line.7 and Spline.8 as reference and target curves respectively.

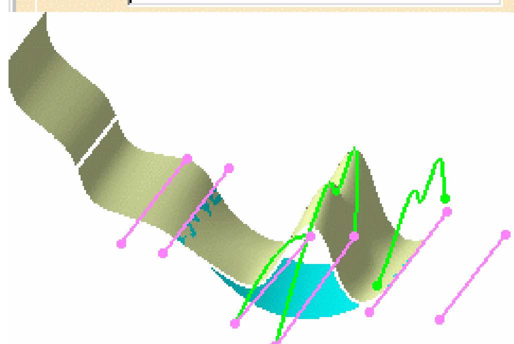


Just like you fixed a reference curve at creation time, you can do it when editing a wrap curve surface:



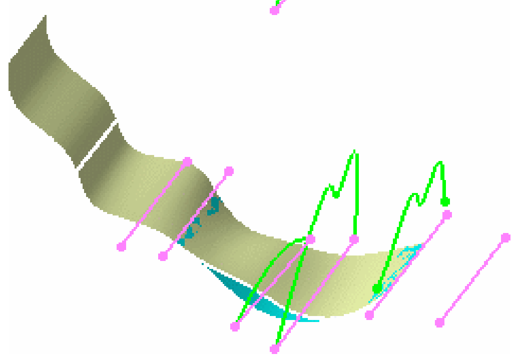
4. Select the fourth line from the list in the dialog box, and check the **Fix reference curve** option.

The target curve is automatically removed from the **Target** column and field.



5. Click Preview.

The resulting surface looks like this:

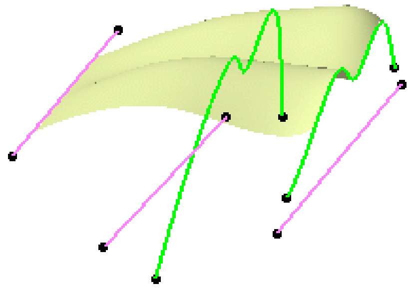


6. Select the third line from the list in the dialog box, and click the Remove button, and click OK.

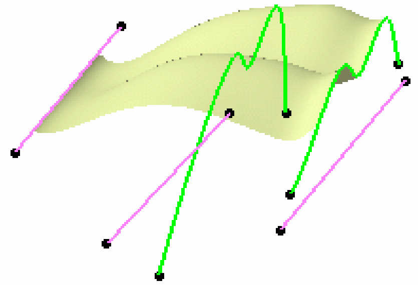
The selected pair of curves no longer being used to compute the resulting surface, the latter looks like this:

You can define further constraints on the deformed surface by means of the **Constraints** fields. You can choose to retain the initial surface's curvature or tangency constraint on the first, and/or last; pair of curves.

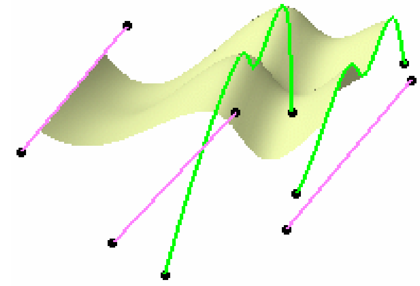
In case of a curvature or tangency continuity, the original continuity between the deformed area and the surface outside the deformation area will be at best kept but may be approximate in certain cases.



Not keeping initial surface's constraint on last reference/target curves



Keeping initial surface's tangency constraint on last reference/target curves

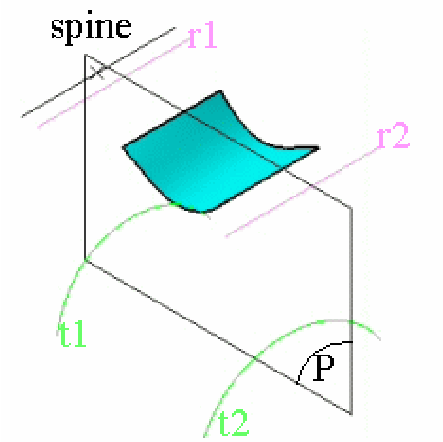


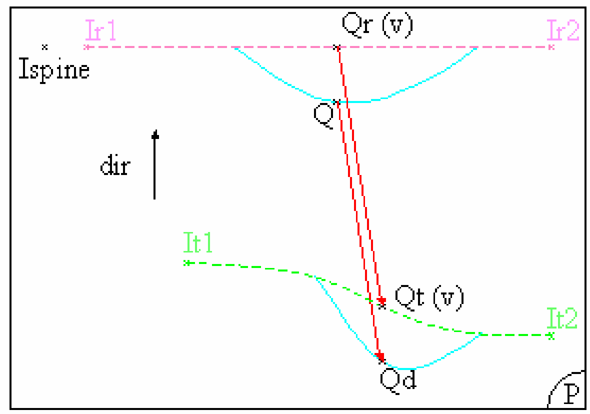
Keeping initial surface's curvature continuity on last reference/target curves

When the spine or the first reference/target curve (default spine) is too short in relation to the corresponding surface's bounding box, the curve is extrapolated according to this bounding box. Then other reference/target curves are extrapolated as well, in relation to this extrapolated spine.

How Is the Deformation Computed ?

The following diagrams will help you understand how the deformation is computed in relation to the entered data, i.e. reference/target curves and possible spine.





3D view, where:

- r1, r2 are the reference curves
- t1, t2 are the target curves
- P is a plane normal to the spine

Planar view, where:

- Ir1: is the intersection between P and r1
- Ir2: is the intersection between P and r2
- It1: is the intersection between P and t1
- It2: is the intersection between P and t2

The deformation is computed in each plane *P*, normal to the spine. By default the spine is the first reference curve, but you can select a new spine using the **Spine** field in the **Reference** tab.

In each plane *P*, the system computes the intersection between the plane and each curve.
A curve (*Cr*) is created between the first intersection point (*Ir1*) and the last intersection point (*Irn*) on reference curves, passing through all the intersection points between these two.
Similarly, a curve (*Ct*) is created passing through all the intersections points between the first (*It1*) and the last intersection point (*Itn*) on target curves.

Then, for each point *Q*, resulting from the intersection of the surface to be deformed with the plane, *Q* is projected onto the curve *Cr* according to the projection direction (*dir*). This projection direction is the vectorial product of: vector(*Ispine*, *Ir2*) ^ vector normal to *P*.
The result of the projection of point *Q* is the point *Qr*, which parameter on *Cr* is *v*.
Similarly, a point *Qt* is created on the curve *Ct*, with the same v parameter as point *Qr* on curve *Cr* .

Then *Qd*, that is the transformation of point *Q* according to the wrap curve deformation, is obtained by adding: *Q+vector(Qr,Qt)*

Deforming Surfaces According to Surface Wrapping

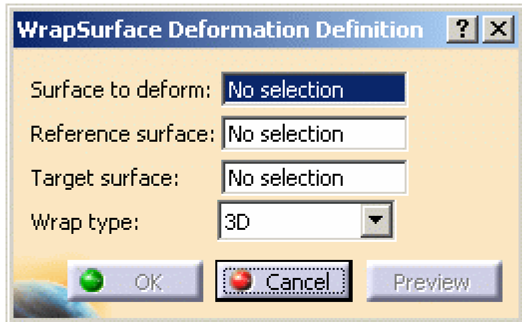
This command is only available with the Generative Shape Optimizer product.

This task shows how to deform surfaces basing the deformation on the projection of the element to be deformed onto two definition surfaces.

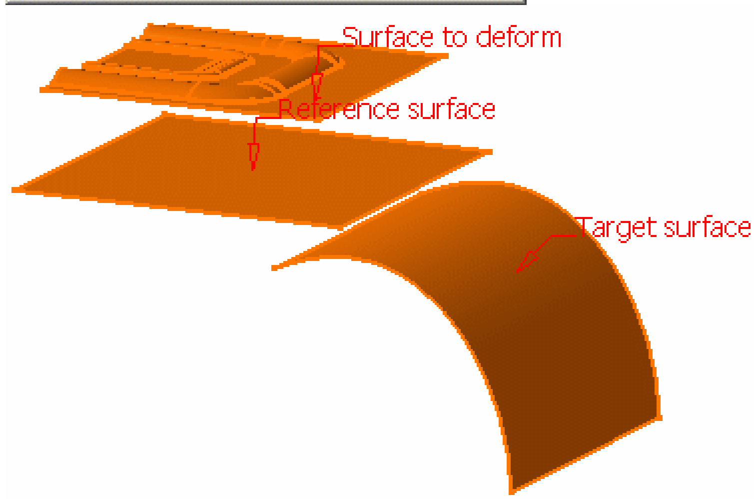
Open the WrapSurface1.CATPart document.

1. Click the **Wrap Surface** icon.

   The WrapSurface Deformation Definition dialog box is displayed.

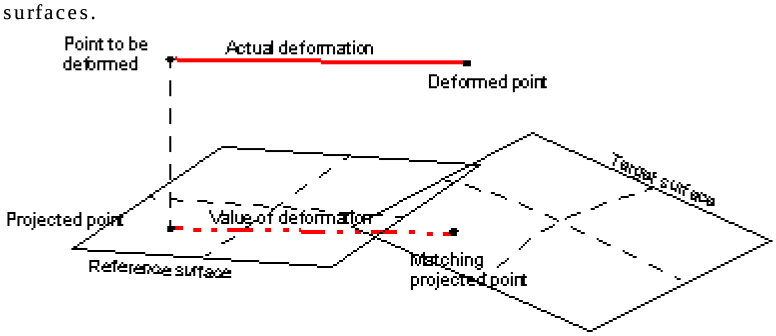


2. Select Surface 1. as the surface to be deformed.
3. Select Surface 2. as the reference surface.
4. Select Surface 3. as the target surface.
5. Select the Wrap type:
   - 3D
   - Normal



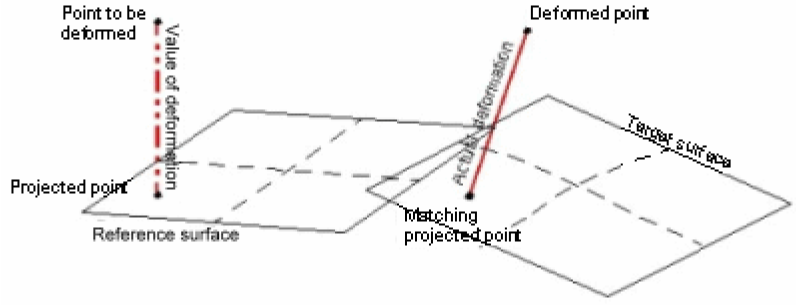
3D Wrap

The following diagram will help you understand how the deformation is computed in relation to the entered data, i.e. reference/target surfaces.



Normal Wrap

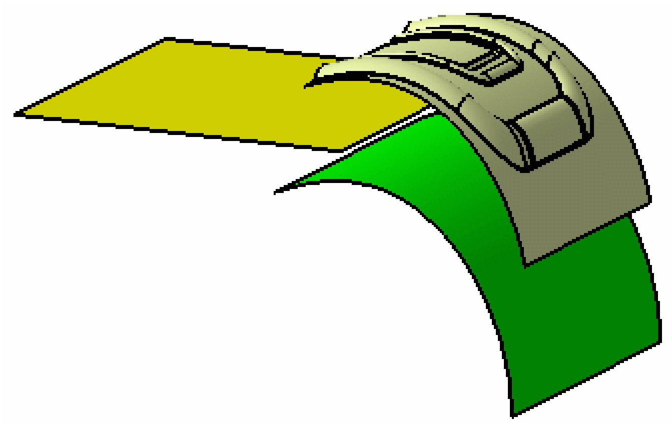
The following diagram will help you understand how the deformation is computed in relation to the entered data, i.e. reference/target surfaces.



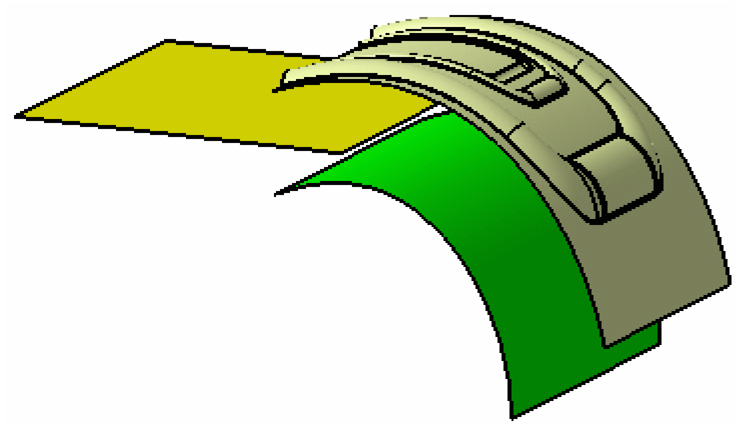
6. Click OK to create the deformed surface.

   The surface to deform is put in no show.

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



3D Surface Wrapping



Normal Surface Wrapping

When the definition surfaces (reference or target surface) are too short to allow the projection of the surface to deform, these surfaces are automatically extrapolated.

- The size of the resulting element may not be identical to that of the initial element, if the reference and the target surfaces do not have the same size.
- Reference and target surfaces must be mono-cell surfacic elements.

Deforming Surfaces According to Shape Morphing

This command is only available with the Generative Shape Optimizer product.

This task shows how to deform surfaces basing the deformation on shape morphing, that is matching each reference curve or point (reference elements) onto a target curve or point (target elements)
The deformation is then defined by the transformation of the reference curves or points into target curves or points.
The elements used for the deformation do not necessarily lie on the initial surface.

Several cases are presented here, from the simplest one to cases using various options. Note that whatever information is given in the first example also applies to the following examples.

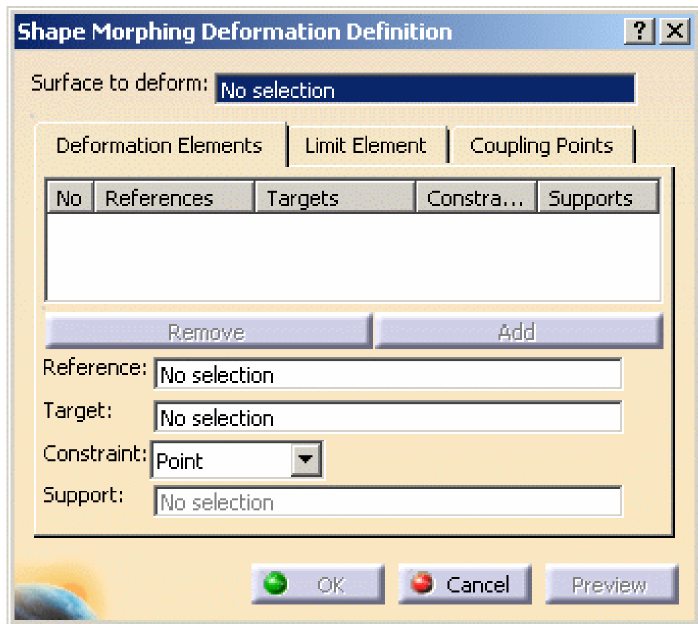
- Basic shape morphing deformation
- Defining a limit element
- Coupling points
- Shape morphing with a fixed element

Open the ShapeMorphing1.CATPart document.

Basic shape morphing deformation

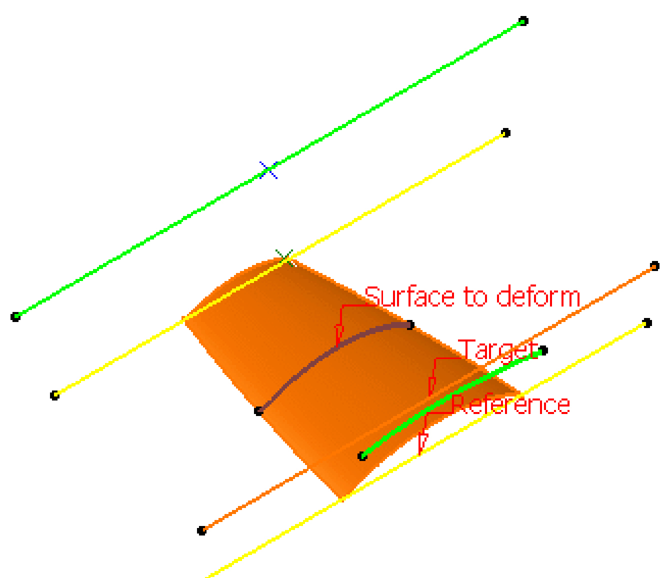
1. Click the **Shape Morphing** icon .

The Shape Morphing Deformation Definition dialog box is displayed.

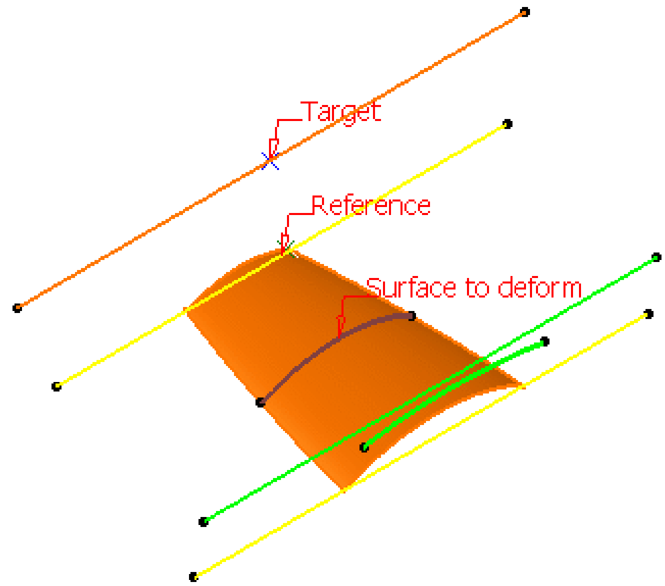


2. Select the surface to be deformed.

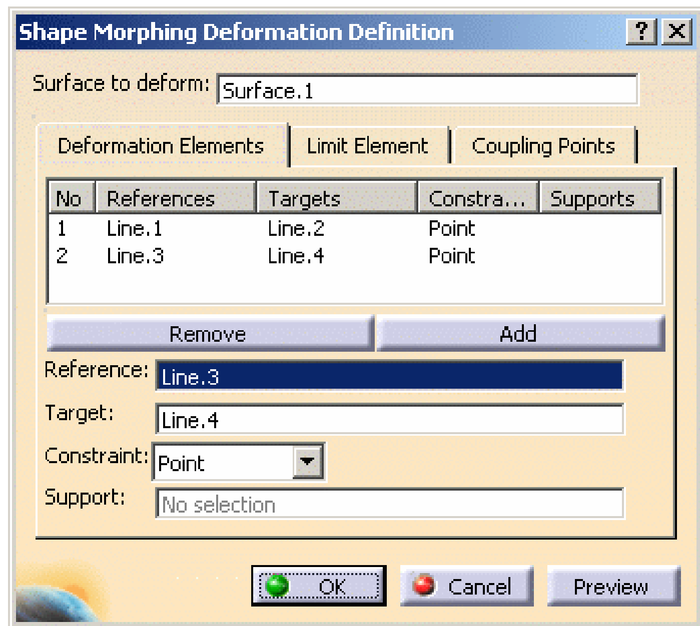
3. Successively select the first reference element and the first target element.



4. Repeat this operation by selecting the second reference element then the second target element.



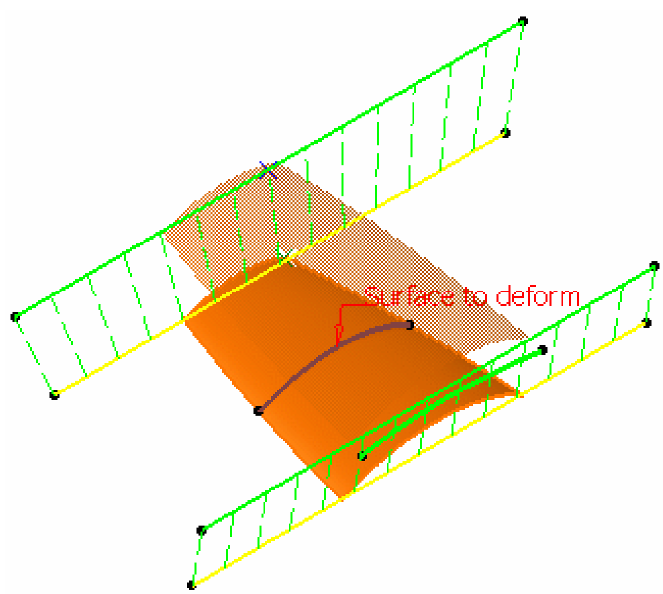
As you select pairs of reference/target elements, the list in the Deformation Elements tab is updated accordingly.

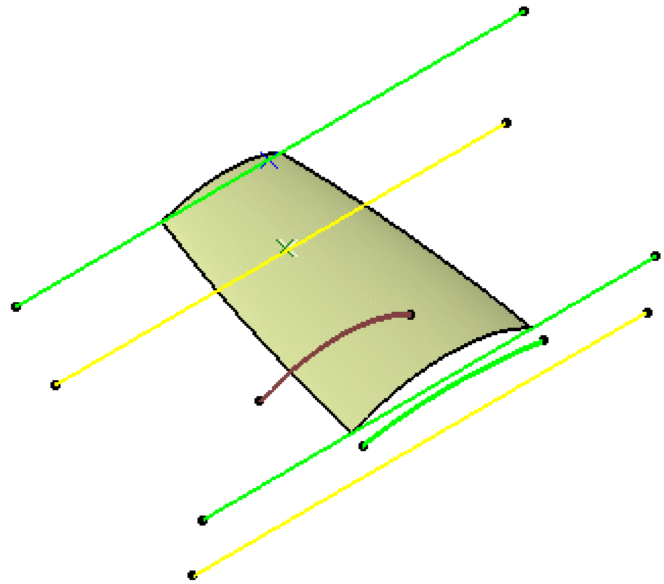


5. Click Preview to previsualize the deformation.

The previsualization shows that:

- the deformation is applied to a group of points
- there is a constraints' mapping between the reference and the target curves.





6. Click OK to create the deformed surface.

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

You can apply a constraint on the target element with the associated support surface.

The combo list displays the available continuity types depending on the reference/target elements you chose.

- You selected a reference **and** a target elements: the **Point** and the **Tangent** continuity are available.

  In the case of a **Point** continuity, the **Support** field is grayed.

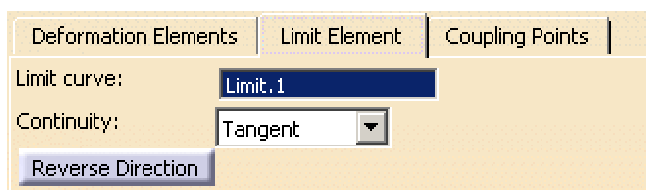
  In the case of a **Tangent** continuity, select a support surface so that the continuity is kept.

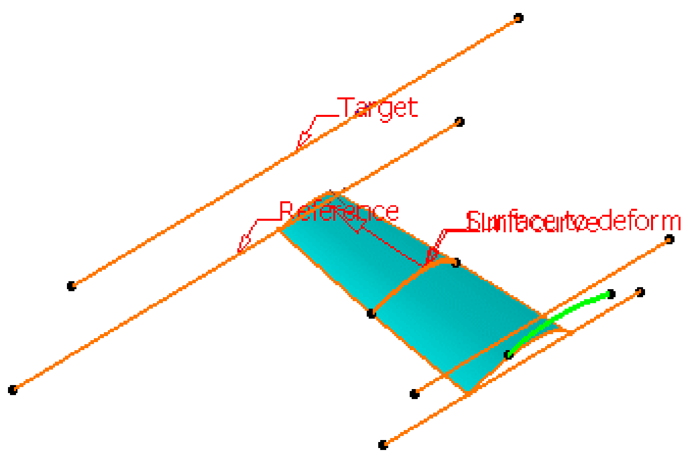
- You selected only reference elements: all continuities (**Point**, **Tangent**, and **Curvature**) are available.

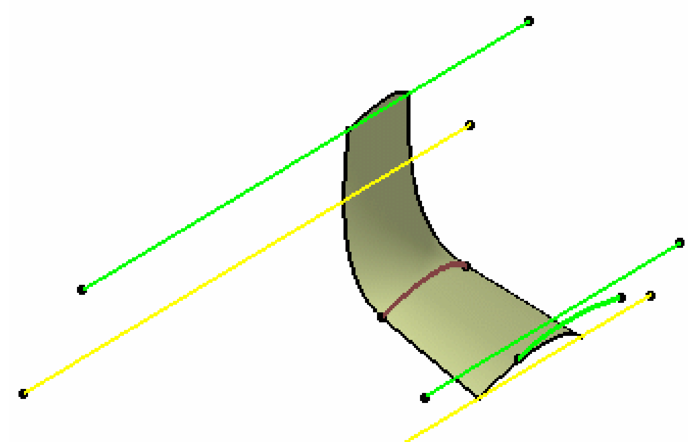
  In the case of a **Tangent** or **Curvature** continuity, you do not need to select a support surface as the surface to deform is taken into account.

Defining a Limit Element

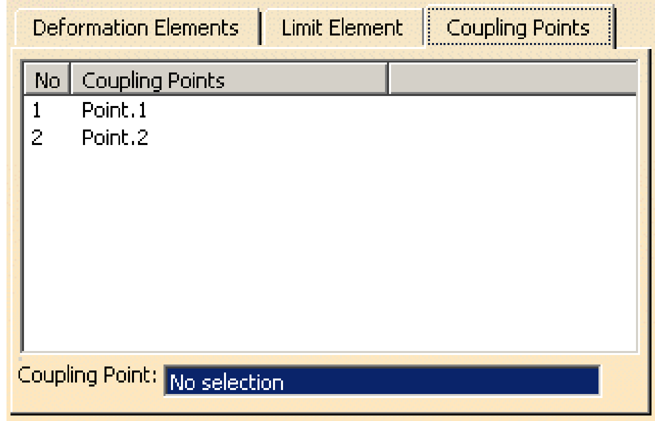
You can define a limit curve to determine the area of the deformation and enable the other part of the surface to remain frozen.
Here is an example using Limit1. as **Limit Curve** and a Tangent **Continuity**.
The Reverse Direction button enables to deform the surface on the other side of the limit curve. You can also click the arrow in the 3D geometry.





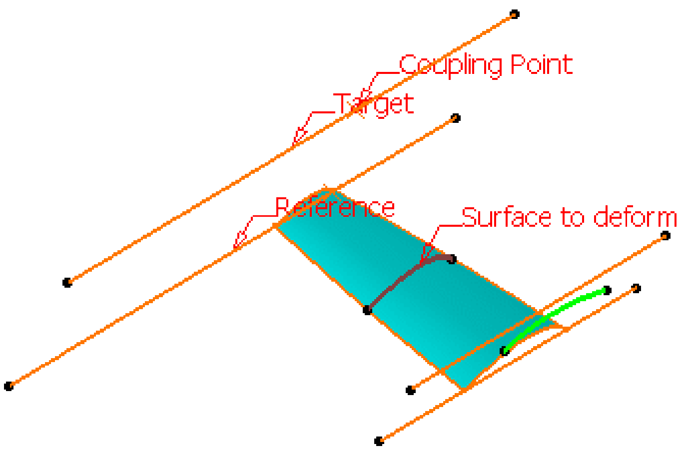


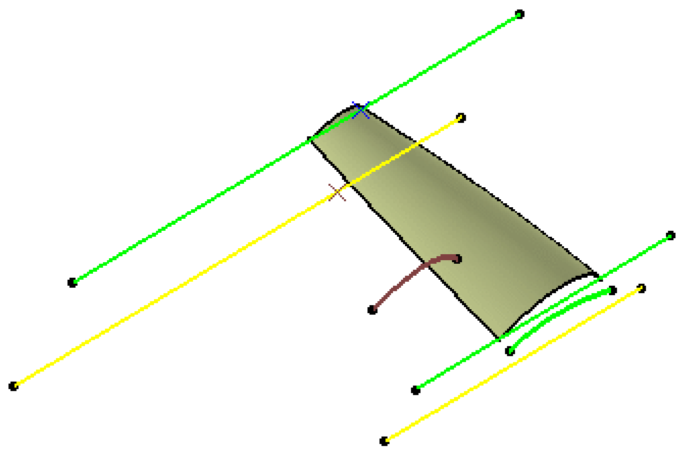
Coupling Points



Use this tab to define coupling points in order to map reference elements with target elements.

Points must be located on reference and target curves.





- You must always select successively a reference then a target element to define a pair. You cannot select all reference elements, then all target elements for example.
- When several pairs of curves are selected, they must be ordered, not randomly selected

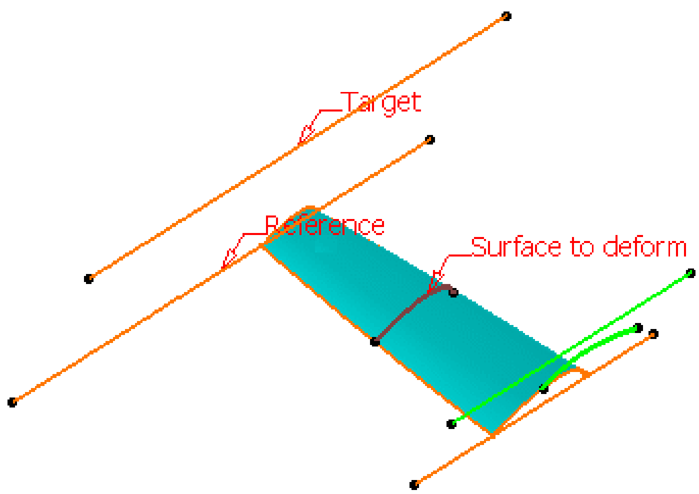
Reference and target curves can be multi-cells. Joined, blended, or matched curves, for example, can be used as reference or target curves.

Shape Morphing Deformation with a Fixed Element

Some times you need to create a deformed surface in relation to another element. The shape morphing capability lets you fix an element that can be used by another one, thus allowing you to retain a connection between elements while deforming the initial surface.

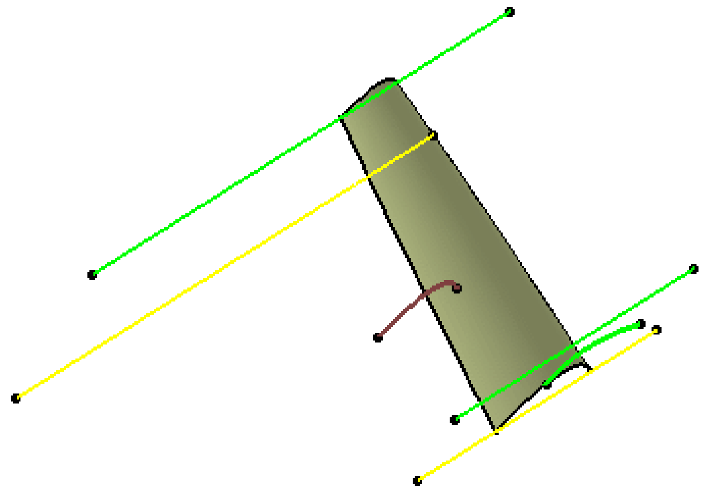
1. Click the **Shape Morphing** icon .

The Shape Morphing Deformation Definition dialog box is displayed.



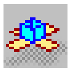
2. Select the surface to be deformed.
3. Select the first reference element.
4. Click the **Add** button to add another reference element.
5. Successively select the second reference element then the target element.

6. Click OK to create the deformed surface.



Working with the Developed Shapes Workbench

 Develop wires and points: select a wireframe contour, a revolution surface, and if needed the developing type, point of origin, and further positioning parameters.

 Unfold a surface: select a surface to unfold, a target plane, and if needed edges to tear, the origin and direction of the target plane.

Developing Wires and Points

This command is only available with the Developed Shapes product.

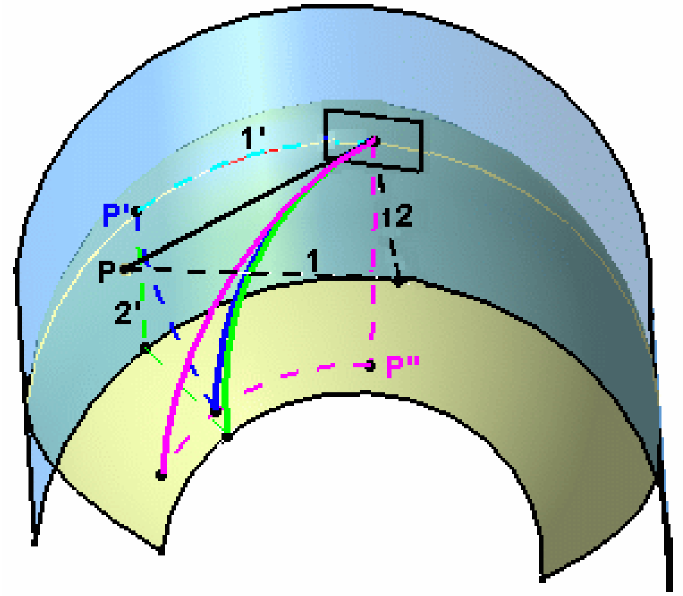
This task shows how to develop wires, and points, onto a revolution surface, that is to create a new wire by mapping a wire's planar abscissa and ordinate with abscissa and ordinate within a local axis-system on a surface, with respect to the surface's curvature.
The wire can be any curve or sketch, provided it is a manifold element. Therefore it cannot be, for example, a T or H-shaped element.

About Developing Wires

There are three modes of developing on a surface:

1. **Develop-Develop**
2. **Develop-Project**
3. **Develop-Develop inverted**

the difference being in the way the points are mapped onto the revolution surface.
The following illustration shows the three developing types, based on developing the black solid wire, the two black dotted wires representing the 1 and 2 coordinate lengths in the wire's axis-system.

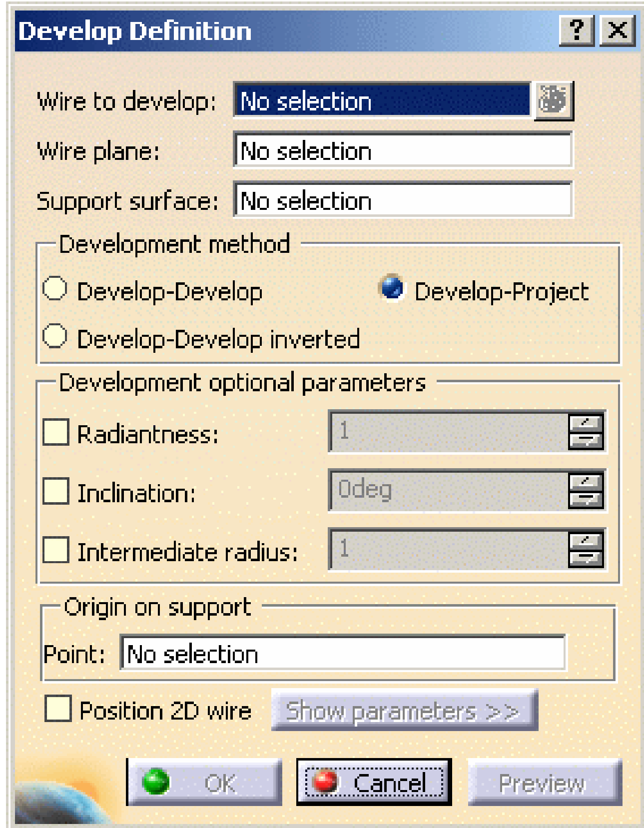


- In the case of the **Develop-Develop** option, a given point (p) of the wire is developed on the revolution surface by mapping its first coordinate as a curvilinear abscissa on the revolution surface (1 into 1') up to a (p') point (represented by the light blue dotted curve), then from that (p') point reporting the other coordinate of (p) as a curvilinear abscissa (2 into 2') along the revolution surface (dark blue dotted curve).
  The resulting developed wire is the dark blue solid curve in the above illustration.
- In the case of the **Develop-Project** option, a given point (p) of the wire is developed on the revolution surface by mapping its first coordinate as a curvilinear abscissa (1 into 1') onto a virtual cylinder passing through the point on support (default or user-defined), to generate a (p') point (represented by the light blue dotted curve), reporting the other coordinate parallel to the cylinder's revolution axis, then projecting normally from that cylinder onto the revolution surface (light green dotted line).
  The resulting developed wire is the light green solid curve in the above illustration.
- In the case of the **Develop-Develop inverted** option, a given point (p) of the wire is developed along the revolution surface by mapping its first coordinate as a curvilinear abscissa on the virtual cylinder up to a (p'') point (represented by the pink dotted line), then from that (p'') point reporting the other coordinate of (p) as a curvilinear abscissa along the revolution surface.
  The resulting developed wire is the pink solid curve in the above illustration
- In the case of a Develop inverse, a given wire is developed from the revolution surface. Therefore, a point on support needs to be specified in order to define the plane, tangent to this point, that will contain the resulting developed wire.
  As an example, if you develop any of the wires in the above illustration using their original development method, the resulting developed wires will be the black solid curve.

As you can see, the results differ slightly, the developed curves not ending on the same point.

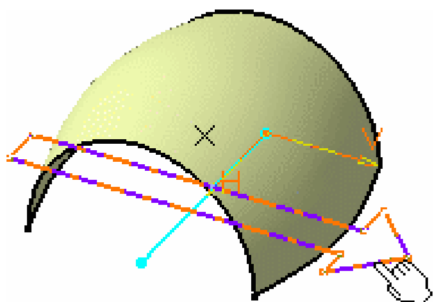
Open the Develop1.CATPart document.

1. Click the **Develop** icon .



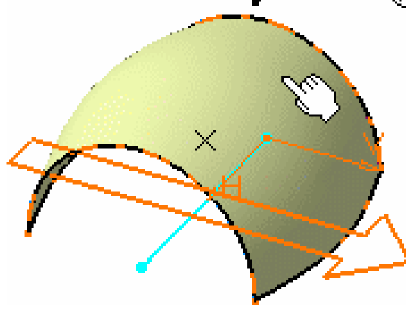
The Develop Definition dialog box is displayed as well as the Multi-Selection dialog box allowing to perform multi-selection.

2. Select the wire to be developed.



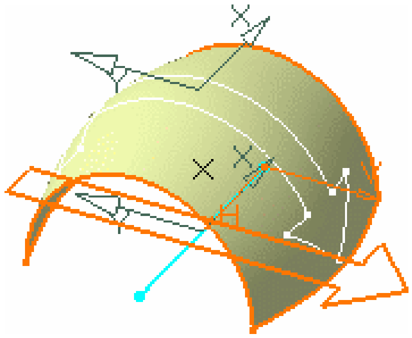
By default, the plane containing this wire is automatically computed. However, when the wire is a line, you need to specify a **Wire plane**.

3. Select the revolution surface onto which the wire is to be developed.



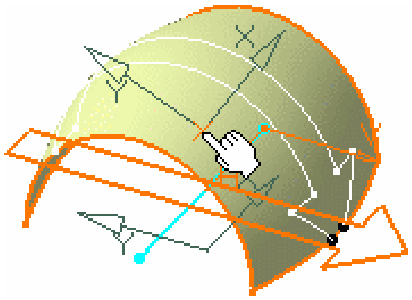
4. Click Preview.

The axis-systems are displayed both on the wire's virtual plane and the surface. These are the default axis-systems. By default, the origin of the support's axis-system is located at a point on the surface where the plane is parallel to the wire's plane.



However, it is usually more pertinent to specify exactly the axis-systems origin.

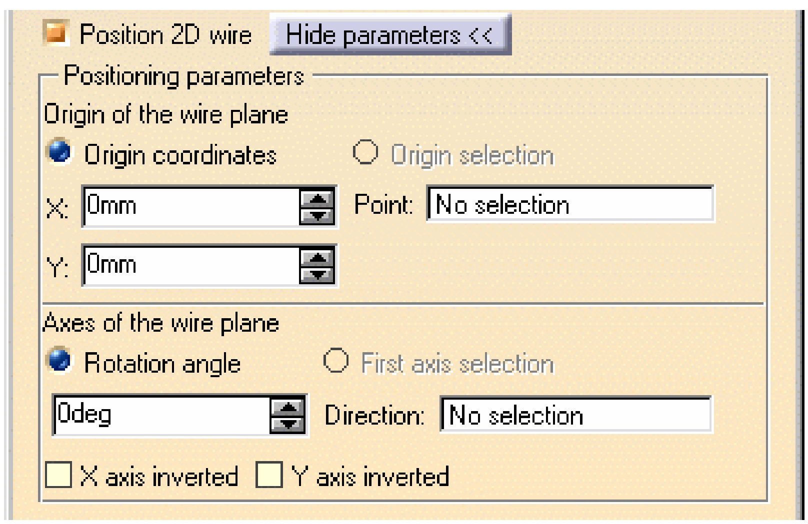
5. Click the **Point** field and select a point, on the surface, defining the support axis-system's origin.



The axis-systems are modified, the support's axis-system to coincide with the selected point, and the wire's axis-system to retain the shortest distance between the two axis-systems' origins.

Consequently, the resulting wire is also modified.

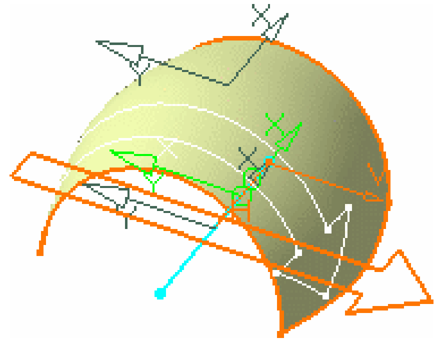
6. If you check the **Position 2D wire** then click the Show Parameters button to expand the dialog box and modify the wire axis-system's positioning.



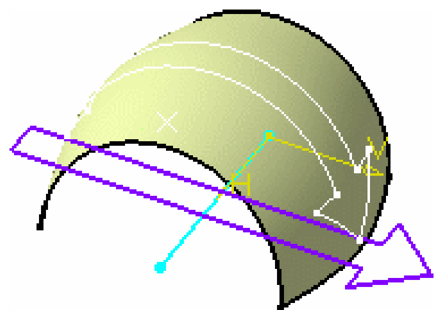
The wire's axis-system turns green, meaning it can be edited, i.e. change location. You can directly move it in the geometry and the dialog box will be updated accordingly.

- Specify the wire axis-system's origin by either entering coordinates, or selecting a point.
- Specify the x-axis of the axis-system by either selecting a line or specifying a rotation angle in relation to the initial lowlight position.
- Select the **X-axis inverted** check box to invert the x-axis orientation (while keeping the y-axis unchanged).
- Select the **Y-axis inverted** check box to invert the x-axis orientation (while keeping the y-axis unchanged).

You could get something like this:



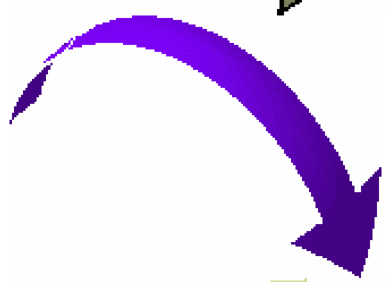
If you want to go back to the initial axis-system positioning, uncheck the **Position 2D wire** button, and collapse the dialog box using the Hide parameters button.



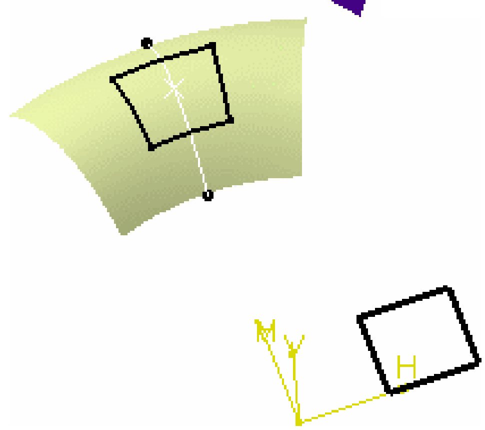
7. Click **OK** to create the developed wire.

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

- You can then fill in the developed wire, to create a developed surface in one click (refer to the Creating Fill Surfaces chapter)



- Three optional parameters are available from the Develop Definition dialog box allowing to apply a transformation to the wire prior to developing it. They are illustrated below by the developing of a square wire onto a surface:



1. Radiantness: allowing to specify a radial deformation ratio on the developed wire. This transformation is defined by the distance between the axis-system origin on the revolution surface and the revolution axis (R), and the ratio you specify in the Develop Definition dialog box.

The formulas used to define the radiantness are:

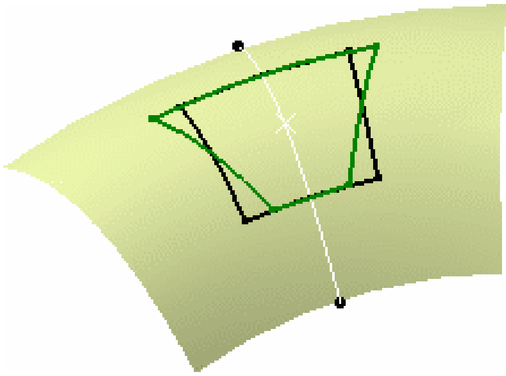
$x' = (R + y_1 * \text{Ratio}) * x_1 / (R + y_1)$

$y' = y_1$

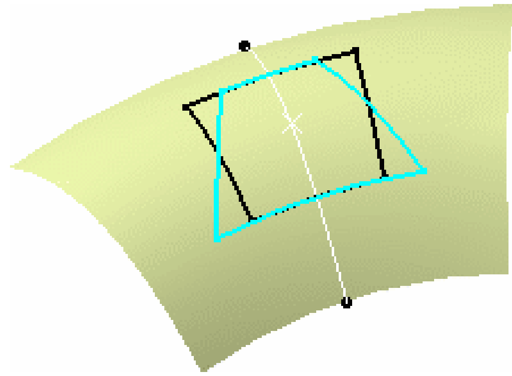
Where:

x_1 and y_1 are the coordinates of any point in the initial axis system of the wire to be developed

x' and y' are the coordinates the same point on the developed wire



Developing with positive radiantness value (green curve)



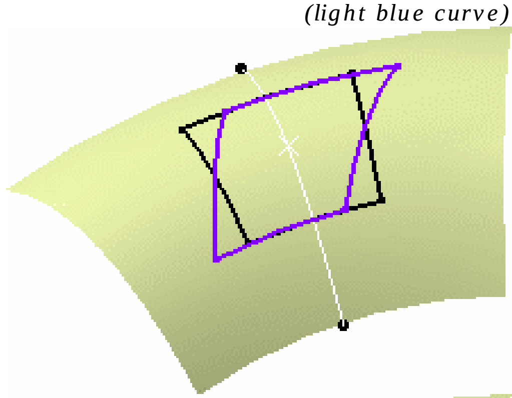
Developing with negative radiantness value (light blue curve)

2. **Inclination**: the angular deviation (d) from the default developing.

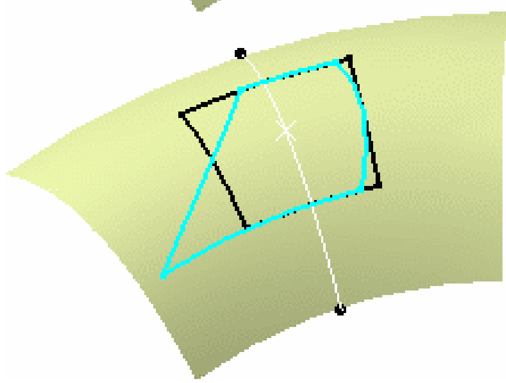
The formulas used to define the inclination are:

$x' = x_1 + y_1 \tan(d)$

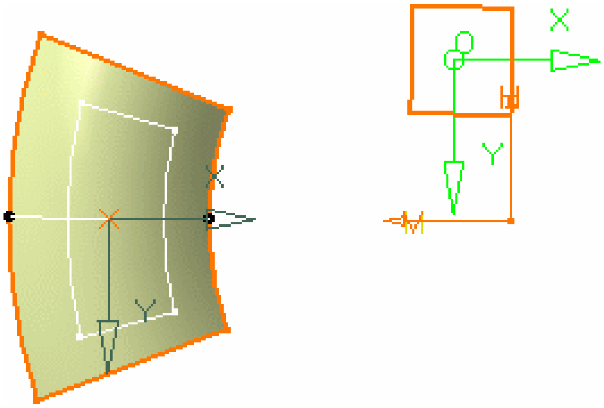
$y' = y_1$



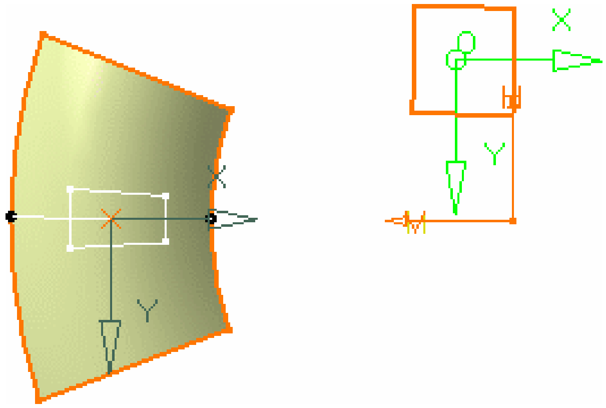
You can combine these two options to develop a wire:



3. **Intermediate radius**: a ratio is applied to the wire's coordinates along the y axis, prior to developing it (i.e. the development operation itself is not affected, only the wire's shape is modified along y before the development).



Developing with intermediate radius value set to 2.
The square's length along y doubles.



Developing with intermediate radius value set to 0.5.
The square's length along y reduced to half its initial length..

Multi-selection of wires to be developed is available. Refer to Selecting Using Multi-Output.

Unfolding a Surface

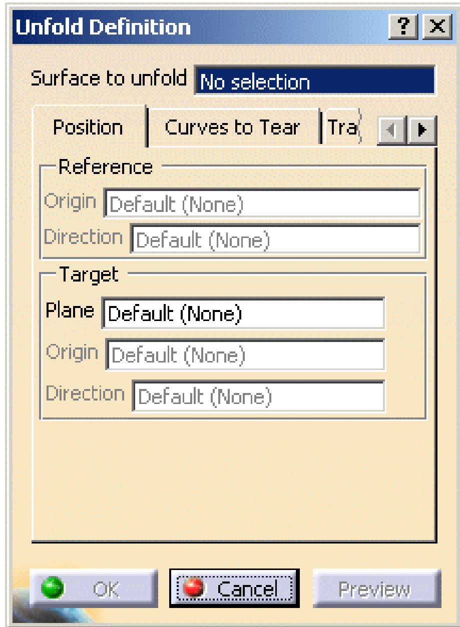
This command is only available with the Developed Shapes product.

This task shows how to unfold a ruled surface.

Open the Unfold1.CATPart document.

1. Click the **Unfold** icon.

The Unfold Definition dialog box appears.

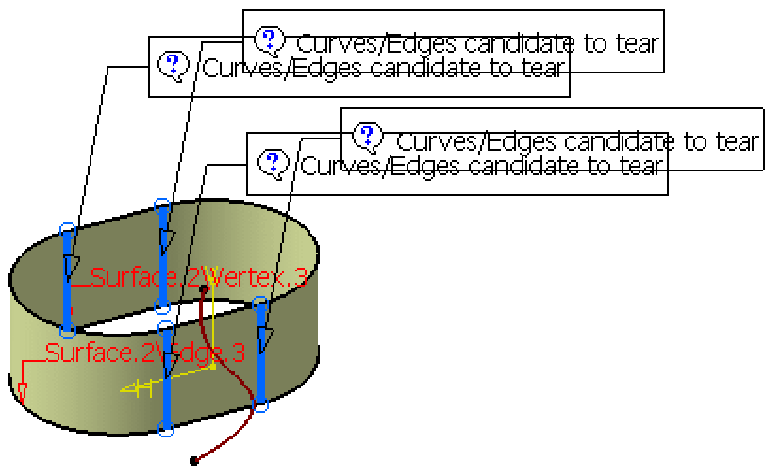


2. Select the **Surface to unfold**.

The unfolded surface is previewed and flag notes display candidate curves to tear (if any) in the 3D geometry.

It is positioned:

- on the selected plane
- such as the image of the selected point on the surface to unfold coincides with the selected point on the plane, and
- such as the image of the tangent to the selected edge on the surface to unfold is collinear with the selected direction on the plane.



If you move the mouse over a flag note, a longer message giving an accurate diagnosis is displayed.

Information on the surface to unfold displays in the dialog box:

- **Origin**: point on the surface to unfold. If no specific origin is selected, it is set to **Default**. By default, when possible, a corner of the surface to unfold is selected.

  If a target plane is defined and a projection is possible, the origin is defined as the projection of the point, selected as the origin on the surface to unfold, onto the target plane. If not, the origin of the axis system of the target plane is selected as the default origin.

- **Direction**: edge of the surface whose extremity is the point. If no specific direction is selected, it is set to **Default**. By default, when possible, an edge of the surface to unfold is selected.

  If a target plane is defined and a projection is possible, the direction is defined as the projection of the tangent to the selected edge onto the target plane. If not, the direction of the target plane is selected as the first direction of the axis system of the target plane.

By default an origin and a direction are selected, and the result is positioned such as this origin and its image as well as the tangent to this direction and its image are coincident.

3. In the Target plane frame, select the plane on which the surface has to be unfolded (here we chose yz plane).

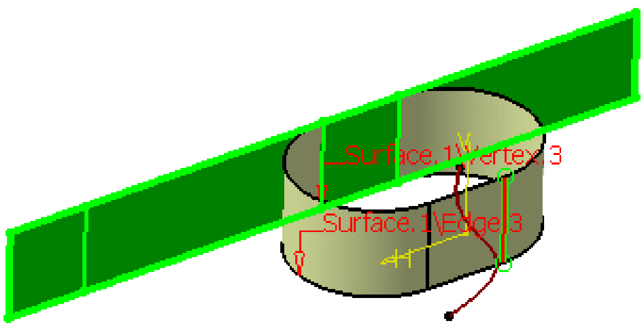
The plane is defined depending on the origin and the direction of the surface to unfold.

4. In the **Curves to Tear** tab, select as many internal and external curves or edges to tear as needed along which the surface is to be developed, so that constraints are solved.

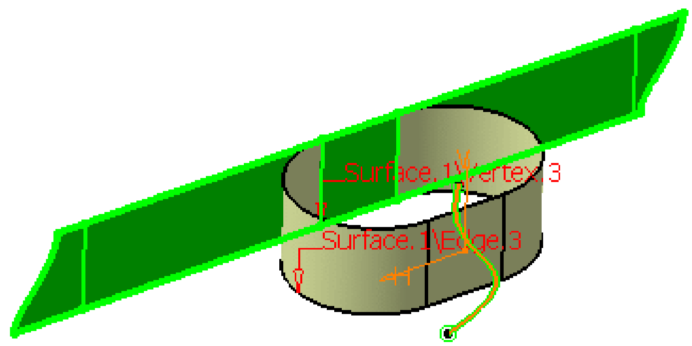
   If no edge of the surface can be defined as candidate, an information message is issued and the Curves to Tear tab displays a list of edges to be selected.

   The selection of curves or edges to tear is optional if there is no curve or edge to tear.

To deselect a curve to tear, simply click on it. The selection is possible again.



Unfold using an internal edge to tear



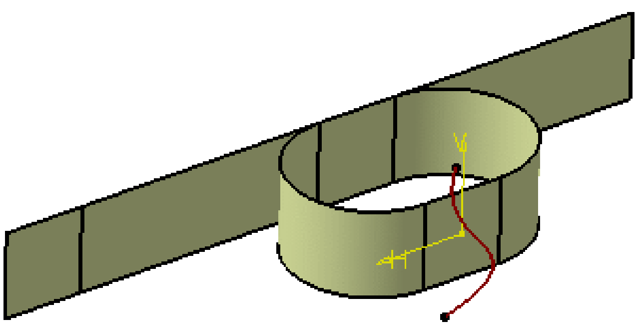
Unfold using an external edge to tear

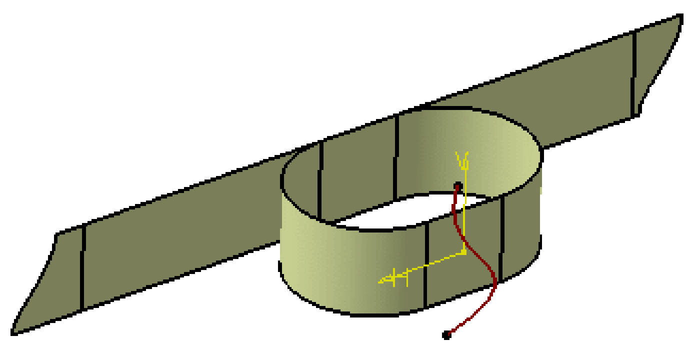
If you do not select edges to tear though you need to, a warning message is issued and the different candidates are displayed in the 3D geometry.

- To select an edge candidate to tear, double-click the information tag or click the edge directly in the 3D geometry (it is highlighted in yellow).
- To select an edge to tear, double-click the information tag or click the edge directly in the 3D geometry (it is highlighted in green).

7. Click OK to unfold the surface.

The developed surface (identified as Unfold.x) is added to the specification tree.



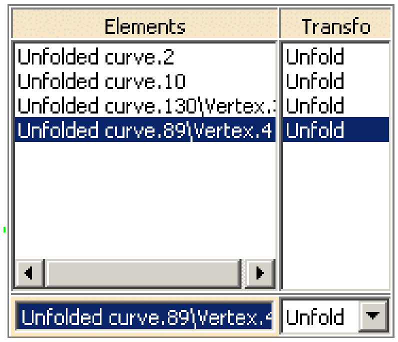


Defining curves or points to transfer

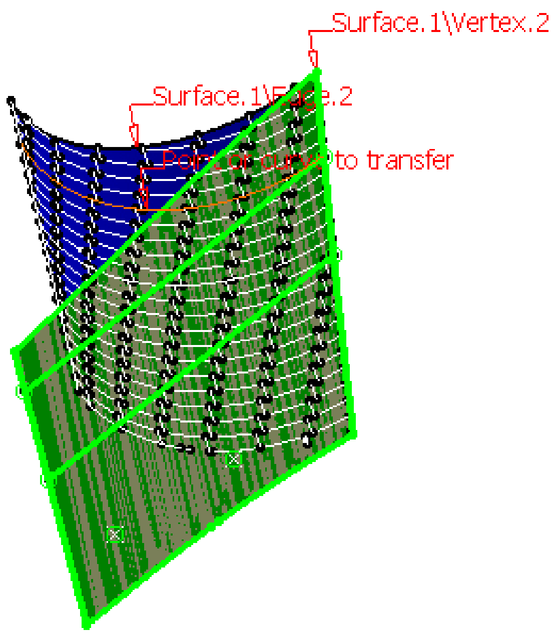
Open the Unfold2.CATPart document.

1. Click the **Unfold** icon .

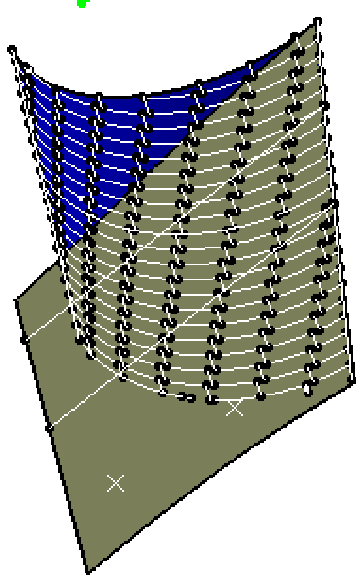
The Unfold Definition dialog box appears.



2. Select the **Surface to unfold**.
3. In the **Transfer** tab, select points or curves on the surface to unfold or on the resulted unfolded surface.
4. Select the type of transformation:
   - **Unfold**: if you selected elements on the surface to unfold
   - **Fold**: if you selected elements on the resulted unfolded surface

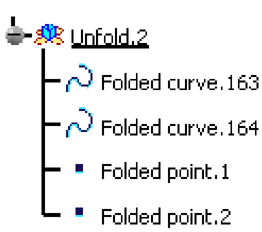


5. Click Preview to see the unfolded surface and elements.



6. Click OK to unfold the surface.

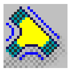
The developed surface (identified as Unfold.x) is added to the specification tree, as well as the transferred elements.

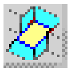


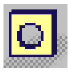
- Mono- and multi-cell surfaces, as well as closed surfaces can be unfolded.
- Multi-cell surfaces and surfaces with internal loops can be unfolded.
- If no point or direction that is not linked to the edges to tear can be selected on the surface to unfold, you can split the surface to unfold (using the **Keep both sides** option to retain the split element after the operation) and unfold both sides.

- Surfaces must be ruled surfaces of degree 1*N. Non ruled surfaces cannot be unfolded.
  A ruled surface is a surface that can be created by sweeping out a linear profile of degree 1 along a guide of degree N.
- Surfaces must have a null Gaussian curvature
- The origin and direction of the surface to unfold must not be located on an edge to tear.

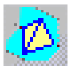
Working With Automotive Body in White Templates

 Create junctions: select two or more sections, define coupling points and tangency constraints on these sections if needed

 Create a diabolo: select the seat surface, the base surface then the draft direction and the draft angle.

 Create a hole: select the center point, the support surface and the punch direction.

 Create a mating flange: select the base surface, the reference element, then define parameters.

 Create a bead: select the base surface, the location point, and the reference direction.

Note that creating macros on the above features is not authorized.

Creating Junctions

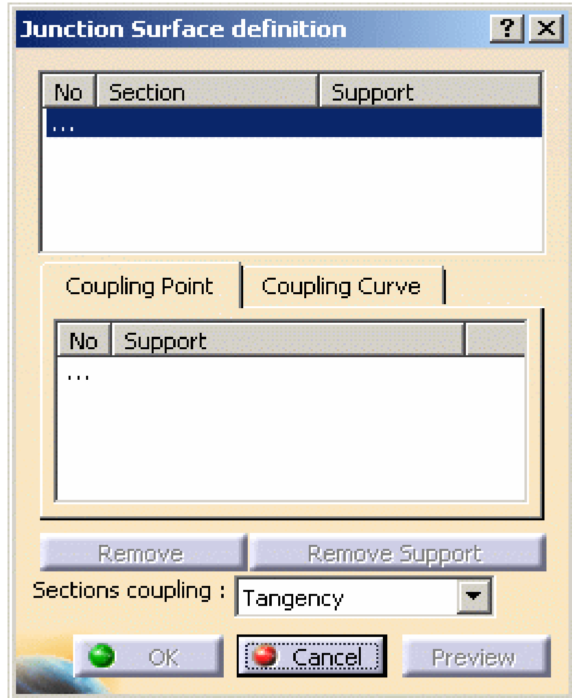
This command is only available with the BiW product.

This task shows how to create junction surfaces between existing surfaces. These surfaces must have been created from contours (sketches, splines, and so forth) provided these are not closed.

Open the Junction1.CATPart document.

1. Click the **Junction** icon.

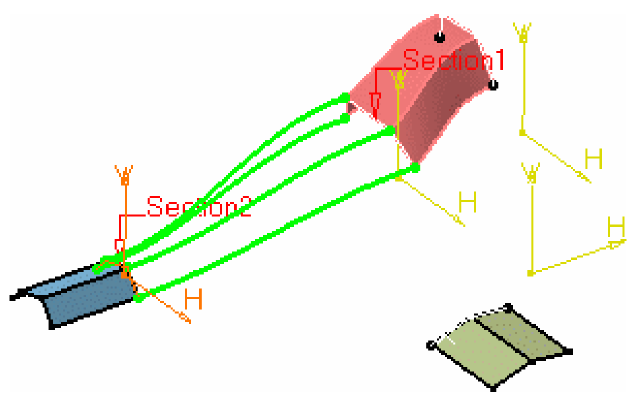
The Junction Surface Definition dialog box is displayed.



2. Select two sections.

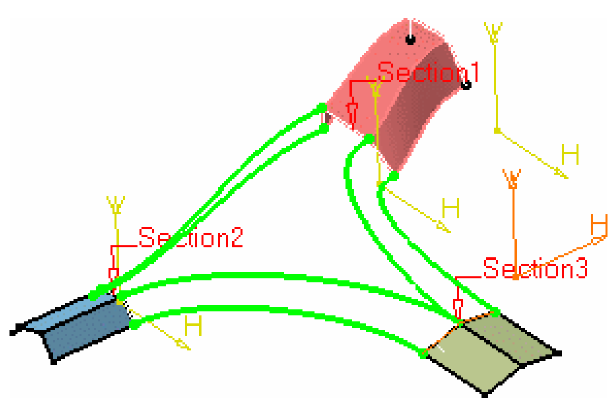
   These can be surface boundaries or contour lying on surfaces.

Coupling curves on which the junction surface will be based are displayed between the two sections.



3. Select another section.

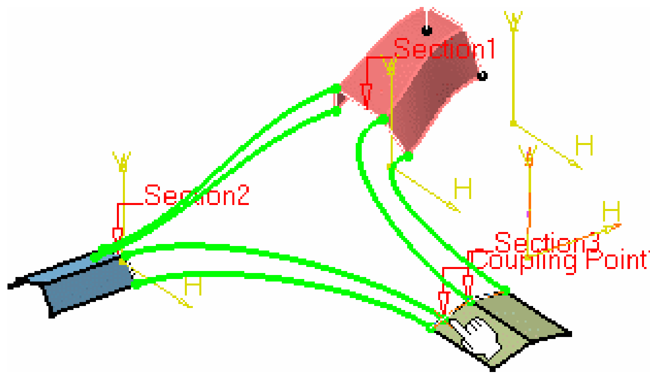
New coupling curves are now displayed.



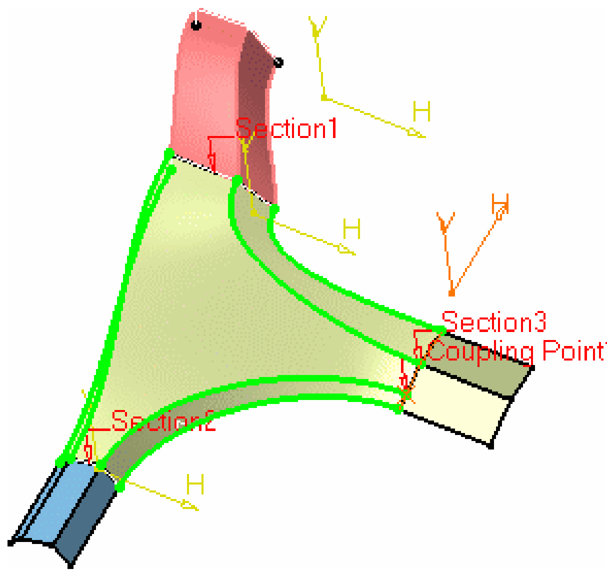
If the sections do not present the same number of vertices, the system automatically links the coupling curves to the sections retaining the maximum number of points.
In our example, two sections present four vertices whereas the last one present only three. The system found a solution by linking two curves to the same vertex on the last section.

Use the **Sections coupling** combo list to specify if the coupling lines are to connect sections on their tangency discontinuity points (**Tangency** option) or on their tangency discontinuity **and** curvature discontinuity points (**Curvature** option).

4. Click within the Coupling Point area then select a point on the section on which you wish to redefine a new passing point for the coupling curve.



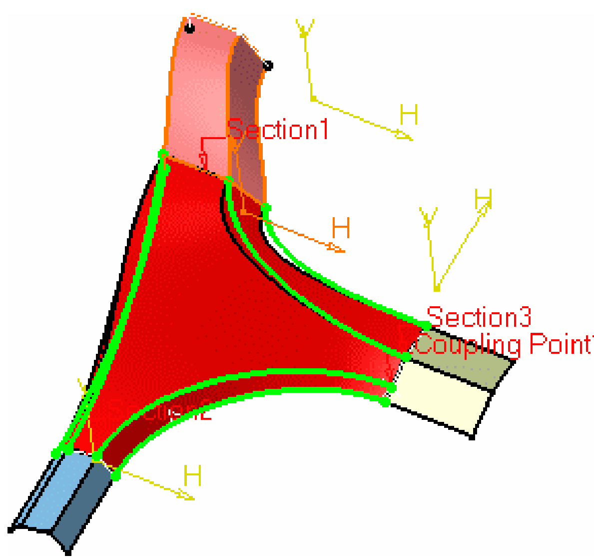
5. Click Preview to preview the junction surface:



By default the coupling curves and the junction surface are tangent to the contour plane's normal.

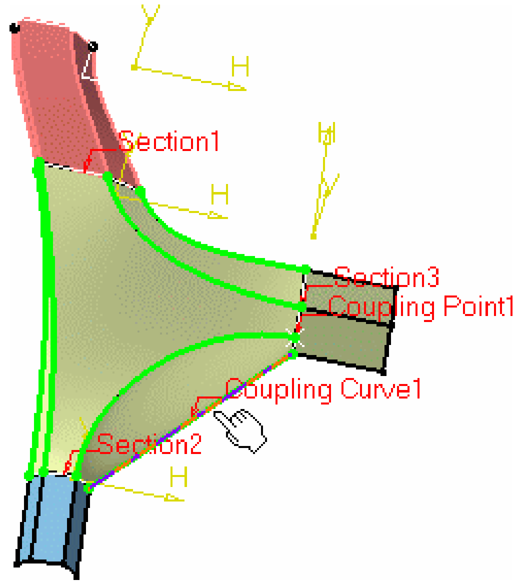
6. Select a section from the list and click the surface on which it lies to add it as a support surface to the section, and therefore define a tangency constraint.

The coupling curves are modified so as to be tangent to the selected surface.

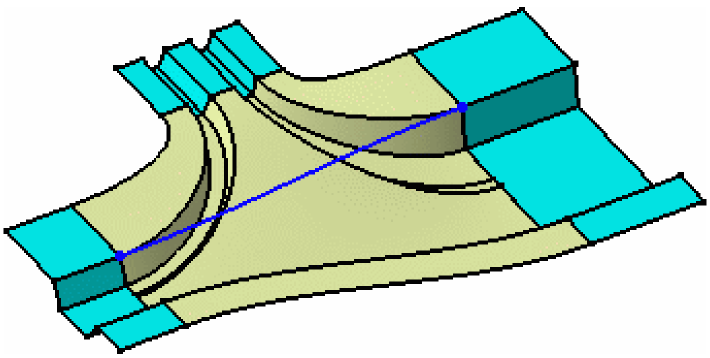


You can also specify a user-defined coupling curve rather than an automatic one, by clicking the **Coupling Curve** tab, then selecting another curve connecting two sections of the junction.

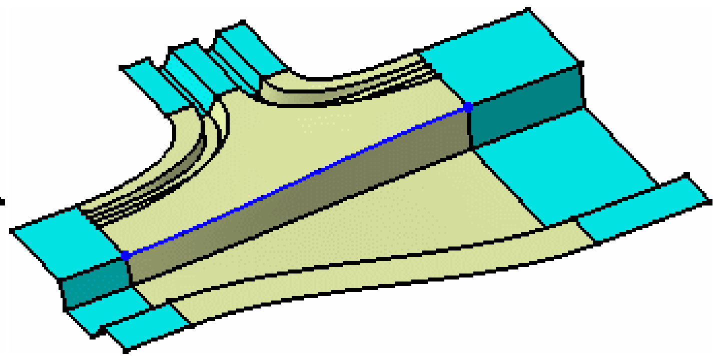
This new coupling curve either replaces an automatic one, or results in a new computation of automatic coupling curves.



Indeed, in the following example, the user-defined coupling curve lies across the automatic ones. These are therefore recomputed to comply with the new constraint:



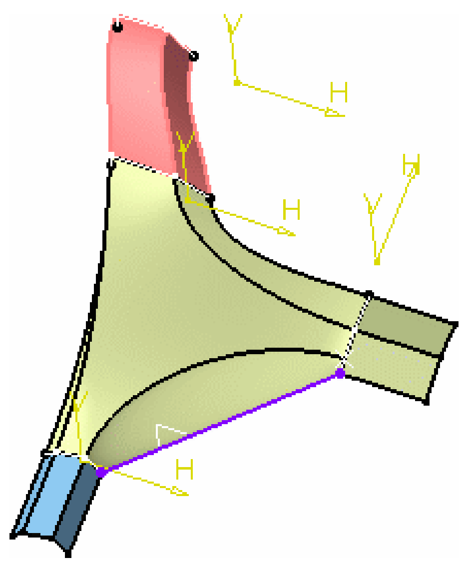
Using automatic coupling curves only



Recomputed automatic coupling curves when using a user-defined coupling curve (blue curve)

7. Click OK to create the junction surface:

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



- You can select as many sections as you wish.
- There is no specific selection order. You can select sections randomly and obtain the same result.
- User-defined coupling curves must end on sections.
- You cannot use a coupling point and a user-defined coupling curve ending on this coupling point.

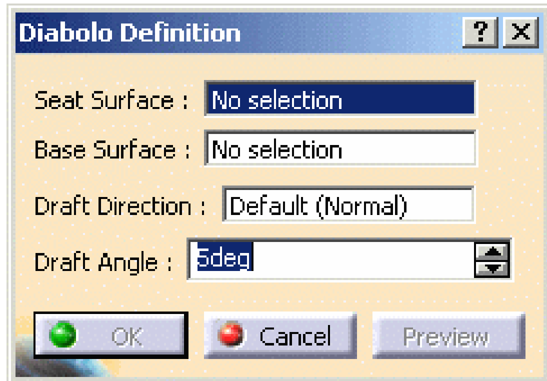
Creating a Diabolo

This command is only available with the BiW product.

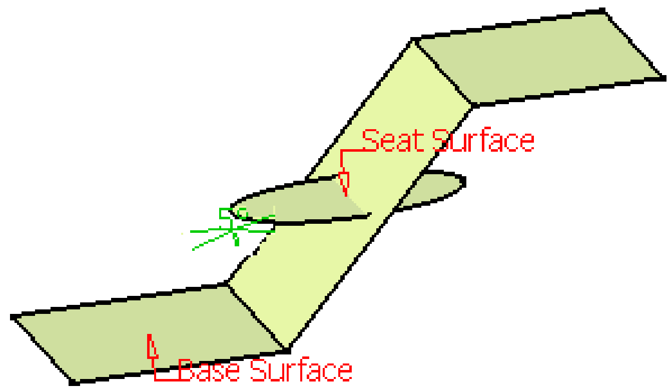
This task shows how to include a seat surface onto a base surface.

Open the Diabolo1.CATPart document.

1. Click the **Diabolo** icon .



The Diabolo Definition dialog box is displayed.



2. Select the Seat surface.

3. Select the Base Surface.

4. Select the Draft direction.

   To define this direction, you can select either a plane, a line or an axis X, Y, Z.

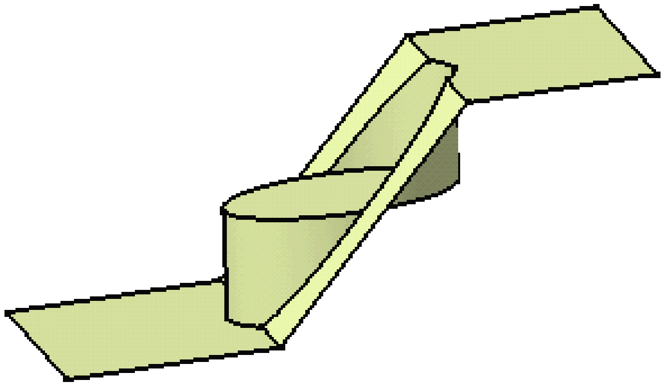
   The draft direction is not mandatory: the default direction is the normal direction to the seat surface.

5. Select the Draft Angle.

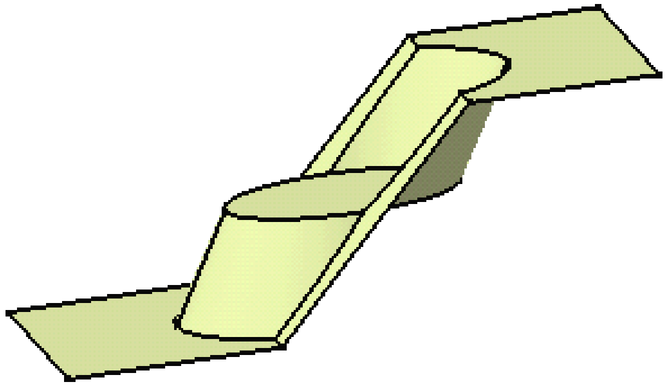
   The default draft angle value is 5 degree.

6. Click OK.

The diabolo (identified as Diabolo.xxx) is added to the specification tree.



Diabolo with a 5 degrees as draft angle



Diabolo with a 30 degrees as draft angle

Both seat and base surfaces must have a close contour and belong to **one** domain.

Creating a Hole

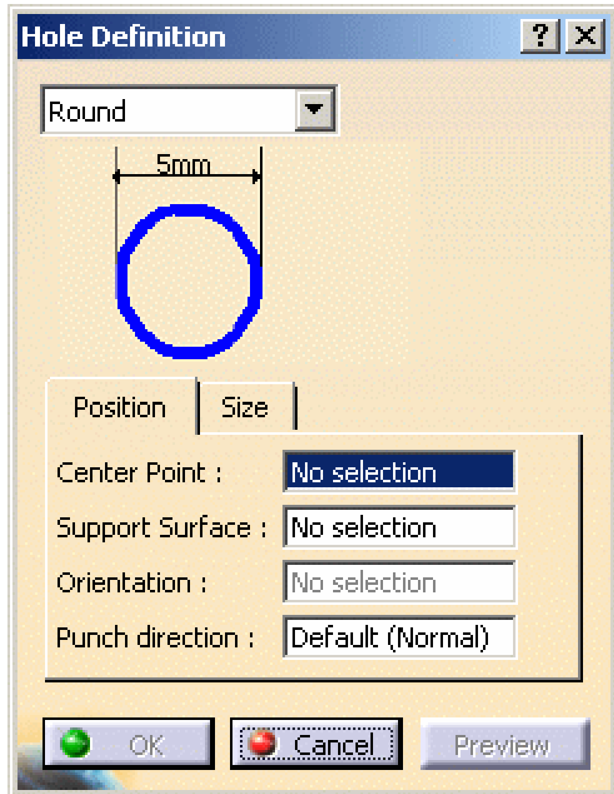
This command is only available with the Automotive BiW Template product.

This task shows how to create a hole, that consists of removing material from a body.

Open the Hole1.CATPart document.

1. Click the **Hole** icon.

The Hole Definition dialog box is displayed.



Various shapes can be created:

- round
- slot (elongated hole)
- rectangular
- square

The shape is defined on a plane and projected along a direction on the surface. In that case, the nearest projection is used to create the hole.

2. Select the shape in the drop-down list.

3. Click a point to be the **Center Point** in the geometry or in the specification tree.

4. Select the **Support Surface**.

5. Define the **Orientation** to align the major axis along a direction.

You do not need to define an orientation for the round holes.

6. Define the **Punch Direction**.

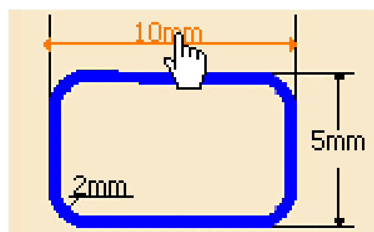
- If the point lies on the support surface, by default it is the normal direction at the center point.
- If the point does not lie on the support surface, you must define a punch direction.

7. Click Preview to visualize the hole.

8. Define the shape dimensions.

To do so, either :

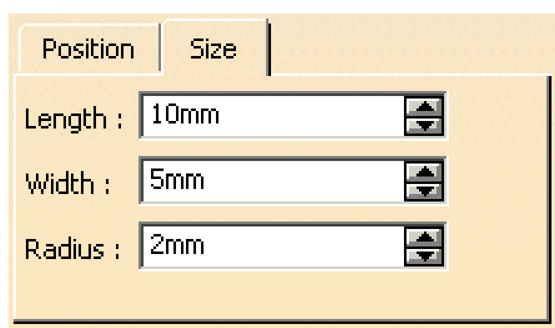
- click the value to edit (here the rectangular length)



The Length dialog box opens to let you modify the dimension.

- click the Size tab.

All dimensions related to the selected shape are displayed and can be modified.

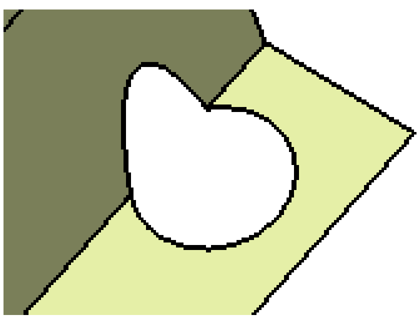


Here are the parameters to be defined depending on the shape hole:

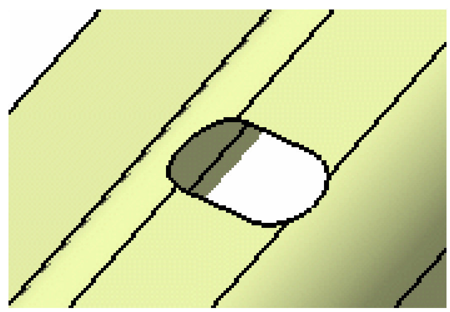
	Length	Width	Radius
Round			X
Slot	X	X	
Rectangular	X	X	X
Square	X		X

9. Click OK to create the hole.

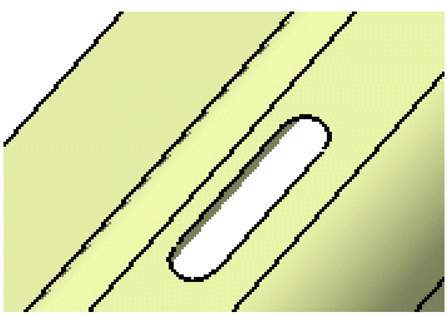
The element (identified as Hole.xxx) is added to the specification tree.



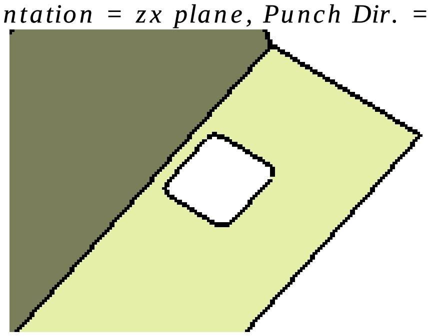
Round hole
Radius = 5mm, Punch Dir. = Default



Slot hole
Length = 30mm, Width = 15mm
Orientation = zx plane, Punch Dir. = Line.1



Rectangular hole
Length = 40mm, Width = 10mm, Radius = 5mm
Orientation = yz plane, Punch Dir. = Default



Square hole
Length = 15mm, Radius = 2mm, Punch Dir. = xy plane

Creating a Mating Flange

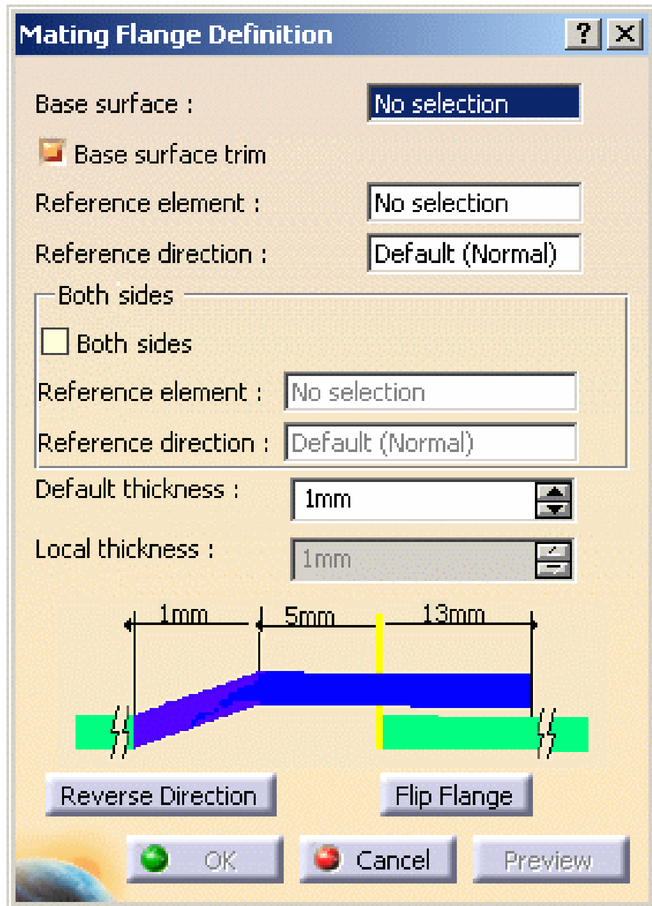
This command is only available with the Automotive BiW Template product.

This task shows how to create a mating flange, in order to add a shape to a part.
This shape is a surface and can be used as a contact zone with another part in an assembly purpose.

Open the MatingFlange1.CATPart document.

1. Click the **Mating Flange** icon.

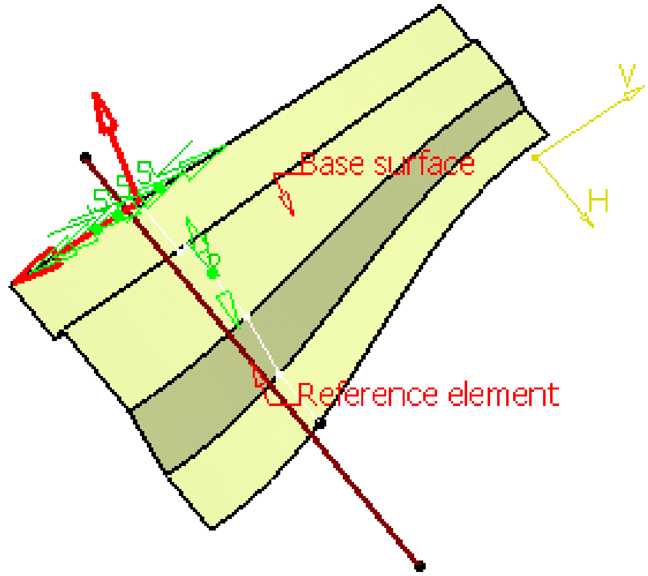
The Mating Flange Definition dialog box is displayed.



2. Select the **Base surface**.

The base surface can have several faces and internal sharp edges.

3. Define the reference location to position the mating flange on the base surface:

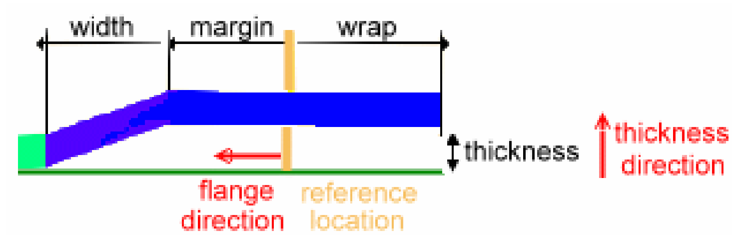


- Select a **Reference element.** It can be:
  - a plane or a surface. The reference location is computed as an intersection with the base surface.
  - a curve (as in our scenario): the curve can be either a 3D curve or a planar curve and must have a projection on the base surface along the reference direction.
- Select a **Reference direction** only if the reference element is a curve. The reference location is a curve computed as a projection along the direction.

In both cases, the intersection or the projection curve must be long enough to join the base surface boundaries.

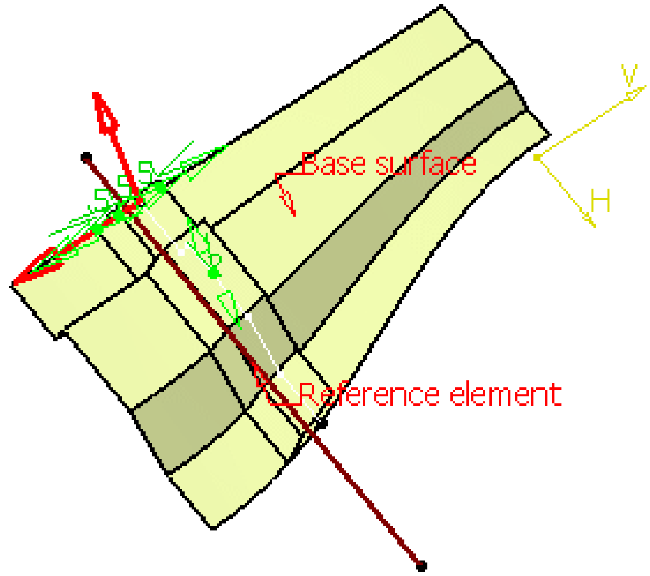
The mating flange reference location feature is created in hidden mode and is temporarily shown during edition.

4. Define the mating flange parameters by clicking the value to edit in the dialog box or by clicking the manipulators in the 3D geometry.



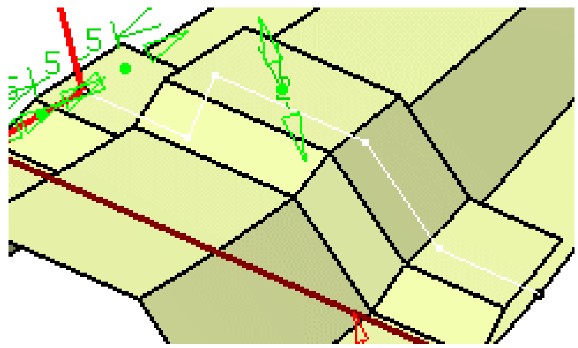
- **Width**
- **Margin**
- **Wrap**

5. Click **Preview**.

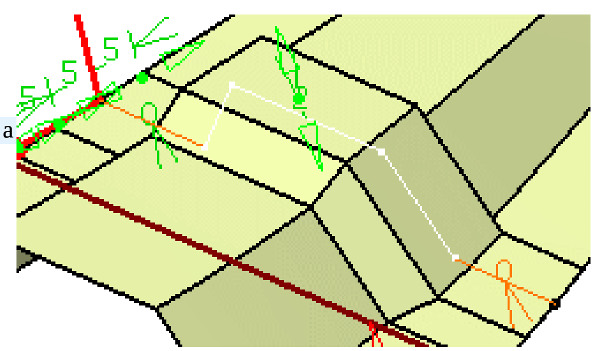


6. Define the thickness:

- **Default Thickness**: is generally the part thickness and is used as the default offset value. You can define its value either by entering a value in the field or using the manipulators in the 3D geometry.

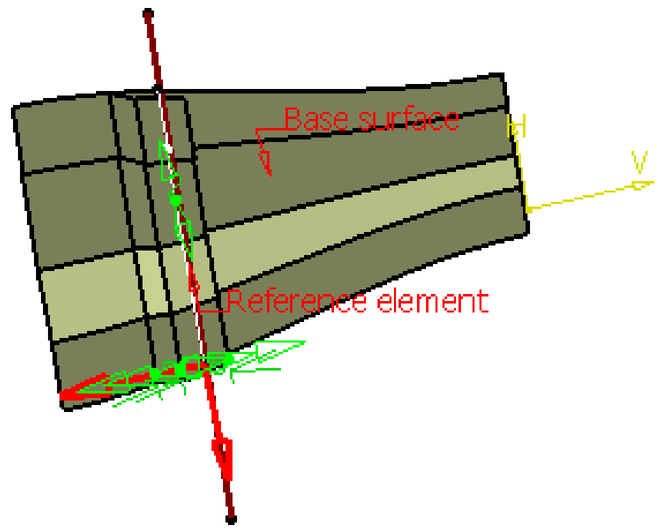


- **Local Thickness**: enables you to define multiple thickness values. They replace the default value and can be positive, negative, or null.
  Select a sub-part of the reference element and define its value either by entering a value in the field or using the manipulators in the 3D geometry.
  You can select several sub-parts, each one having its own local thickness. For each value, a corresponding 3D dimension is created in the 3D geometry and can be edited by double-clicking it.

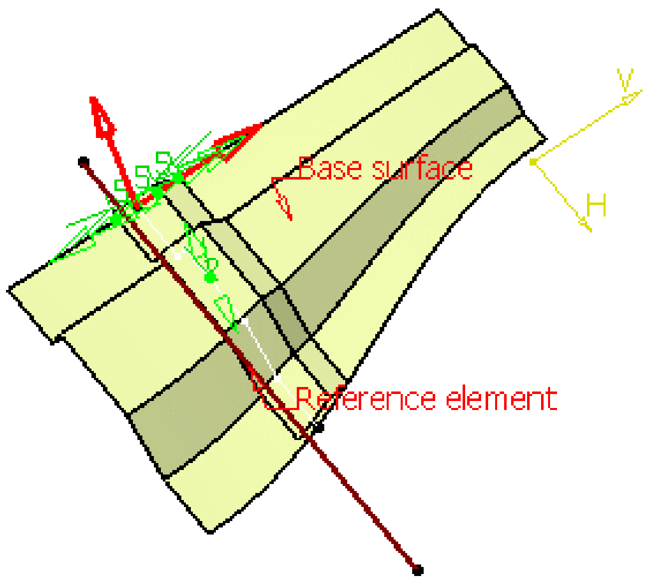


  In case no local value is defined, the **Local Thickness** field is grayed out. Otherwise, the corresponding sub-part and the 3D dimension are highlighted in the 3D geometry. If you select the highlighted sub-part, the local value is deleted and the default thickness value is used.

The thickness that is aggregated under the mating flange feature is the default thickness.

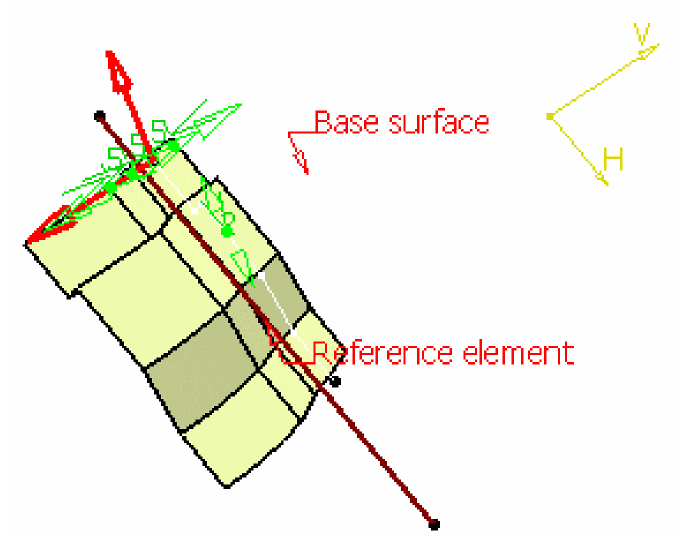


- The **Reverse Direction** button allows you to inverse the thickness direction, according to the orientation of the reference element.
  As as consequence, the mating shape is displayed on the other side of the base surface.

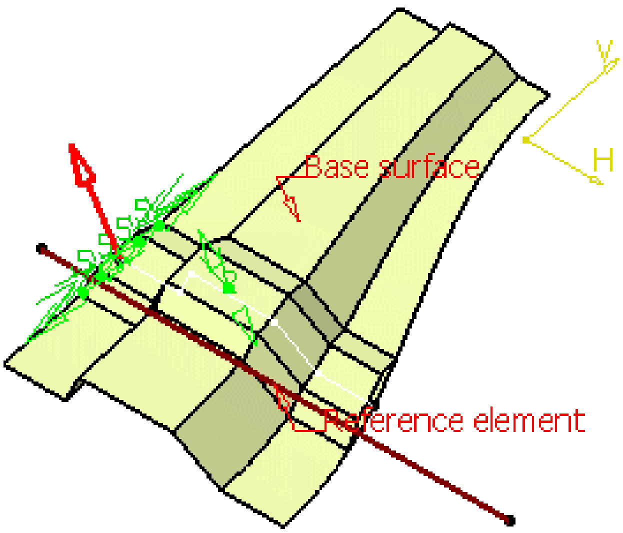


- The **Flip Flange** button allows you to inverse the mating flange direction, according to its orientation.
  As as consequence, the mating shape is displayed on the other side of the reference location.

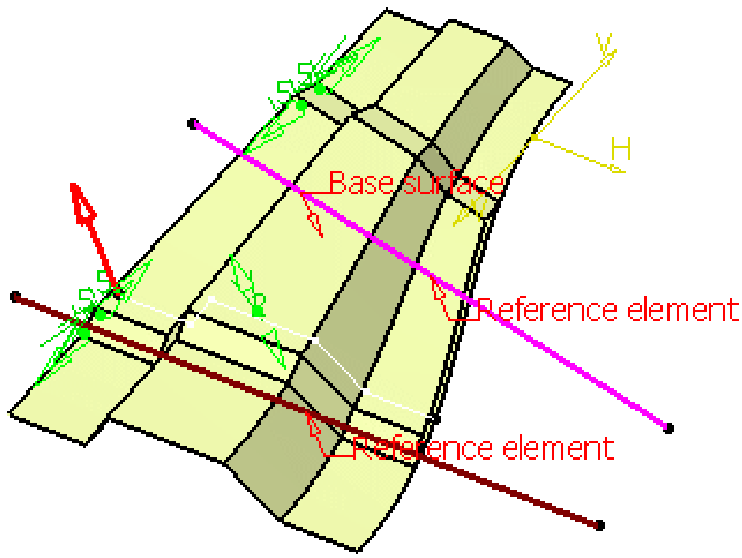
- The **Base surface trim** button enables you to trim the surface with the mating flange.



- The **Both sides** button enables to create a both-side mating flange using a second reference element. By default, the **Reference element**, as well as the second **Reference direction**, are the same as the first reference element and direction, but you can choose other ones.



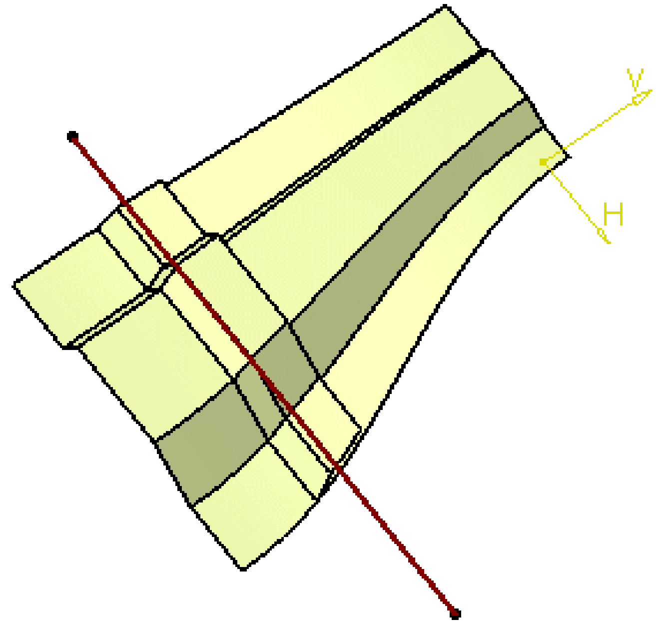
With the same reference element



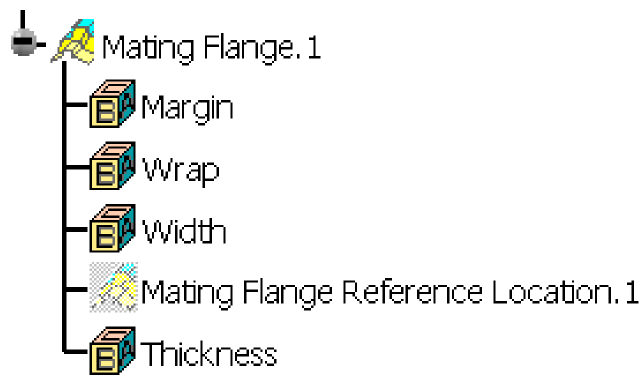
With a second reference element

The **Flip Flange** button is greyed out.

7. Click OK to create the mating flange.



The new shape (identified as Mating Flange.xxx) is added to the specification tree.
Its reference location is aggregated under the Mating Flange feature and can be used as an input for a further operation.



Creating a Bead

P2

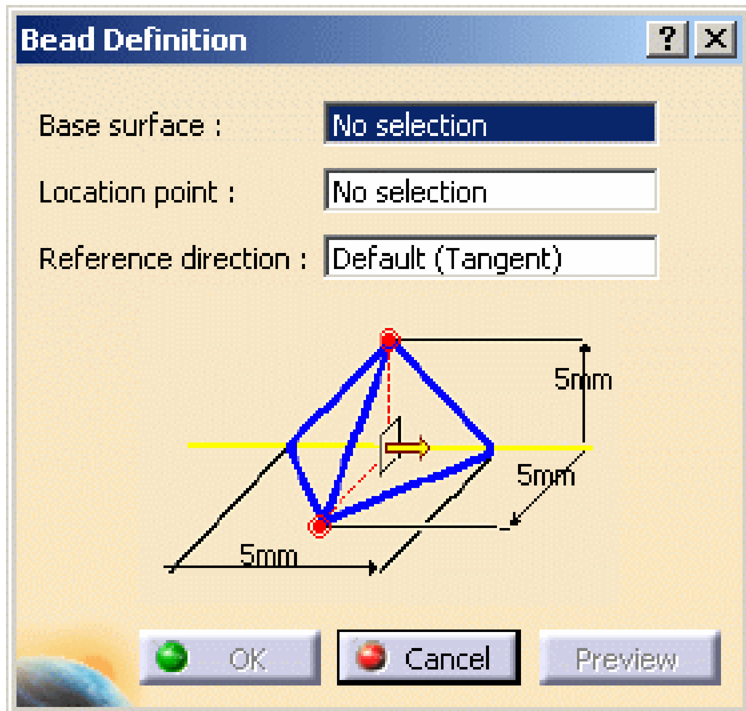
This command is only available with the Automotive BiW Template product.

This task shows how to create a bead, in order to add strength a part.
This shape is a surface and is a triangle bead shape.

Open the Bead1.CATPart document.

1. Click the **Bead** icon .

   The Bead Definition dialog box is displayed.



2. Select the **Base surface**.

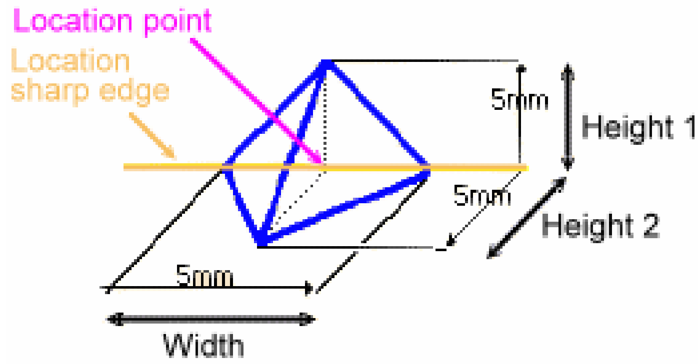
The base surface must have at least one internal sharp edge.

3. Select a point on the sharp edge.

4. Define a **Reference direction**.

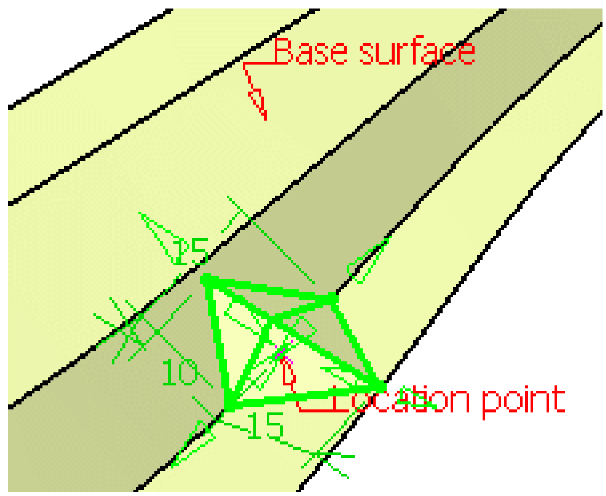
   By default, it is the tangent direction to the location edge at the location point.

5. Define the bead parameters by clicking the value to edit in the dialog box or by clicking the manipulators in the 3D geometry.

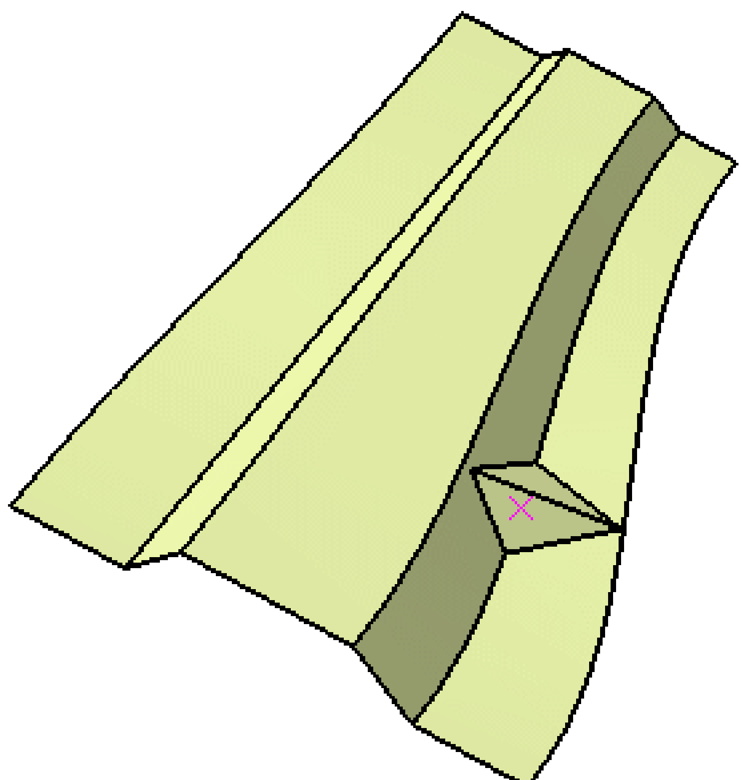


- **Height**
- **Width**

These values must be positive.



6. Click **Preview**.



7. Click **OK** to create the bead.

Creating Volumes

The Volumes Toolbar is only available with the Generative Shape Optimizer product.

Creation of volumetric features

Volumetric features can be created in both Geometrical Set and Ordered Geometrical Set environments and are considered as creation features.

Modification of volumetric features

Several modification features enable to modify a volumetric feature into another volumetric feature:

- Split
- All fillets (but the shape fillet) : edge fillet, variable radius fillet, face-face fillet, and tritangent fillet
- All transformations: rotate, translate, symmetry, scaling, affinity, and axis to axis
- Patterns: circular pattern

General Behavior

The following commands can be used with volumetric features:

- Delete
- Deactivate/Activate
- Parent/Children
- Datum mode
- Reorder (note that it is not possible to reorder the volumetric feature before its parents)
- Replace (a volumetric feature can only be replaced by another volumetric feature)
- Show/no show
- Stacking
- Search: "Volumes" type

When working with solids and volumes, the selection of the feature prevails over the selection of the sub-element.

To select a sub-element, you need to apply the "Geometrical Element" filter in the User Selection Filter toolbar. You can activate this toolbar by selecting the **View -> Toolbars** command and clicking User Selection Filter.

- The icons on the left lets you filter elements according to their type (point, curve, surface, volume)
- The next two icons correspond to the filter modes:
  -  the "Feature Element Filter" selects the whole feature whether it is a sketch, product, pad, join, etc.
  -  the "Geometrical Element Filter" enables to sub-elements of a feature such as faces, edges or vertices

For further information about this toolbar, refer to the Selecting Using a Filter chapter in the *CATIA Infrastructure User's Guide*.

 Auto-intersections can occur: all of them may not be supported.

 Create extruded volumes: select a profile, specify the extrusion direction, and define the start and end limits of the extrusion

 Create revolution volumes: select a profile, a rotation axis, and define the angular limits of the revolution volume

 Create multi-sections volumes: select section curves, guide curves if needed, then enter the required parameters

 Create swept volumes: select the sub-type, then enter the required parameters

 Create a thick surface: select the object to be thickened, define the offset directions and enter offset values

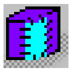
 Create a close surface: select the surface to be closed

 Create a draft: set the Selection by neutral face selection mode or select the face to be drafted, then enter the required parameters

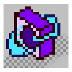
 Create a variable angle draft: select the face to be drafted, click as many points as you wish and then enter the required parameters

 Create a draft from reflect lines: select the face to be drafted, then enter the required parameters

 Create a shell: select the faces to be shelled and enter the thickness values

 Create a sew surface: select the volume and the object to be sewn

 Add volumes: select the volume to be added then the target volume

 Remove volumes: select the volume to be removed then the target volume

 Intersect volumes: select the first volume then the second volume

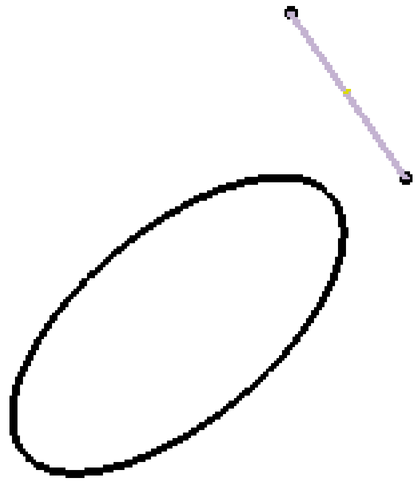
 Trim volumes: select the volume to trim then the cutting volume

Creating Extruded Volumes

P2

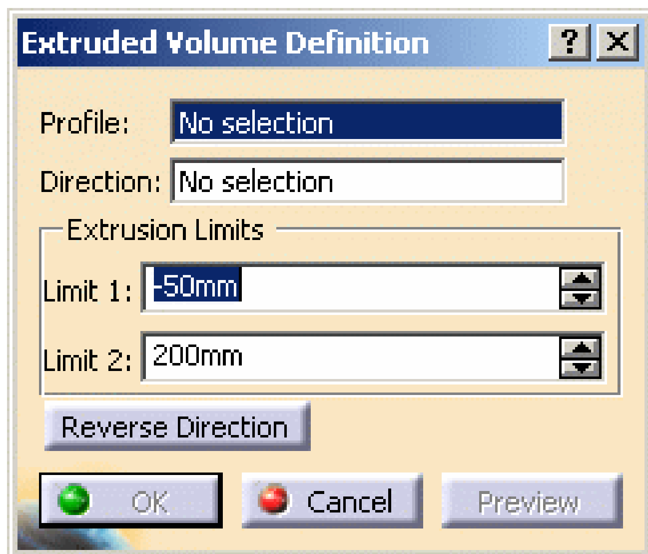
This task shows how to create a volume by extruding a profile along a given direction.

Open the ExtrudedVolume1.CATPart document.



1. Click the **Volume Extrude** icon .

The Extruded Volume Definition dialog box appears.



2. Select the **Profile** to be extruded.

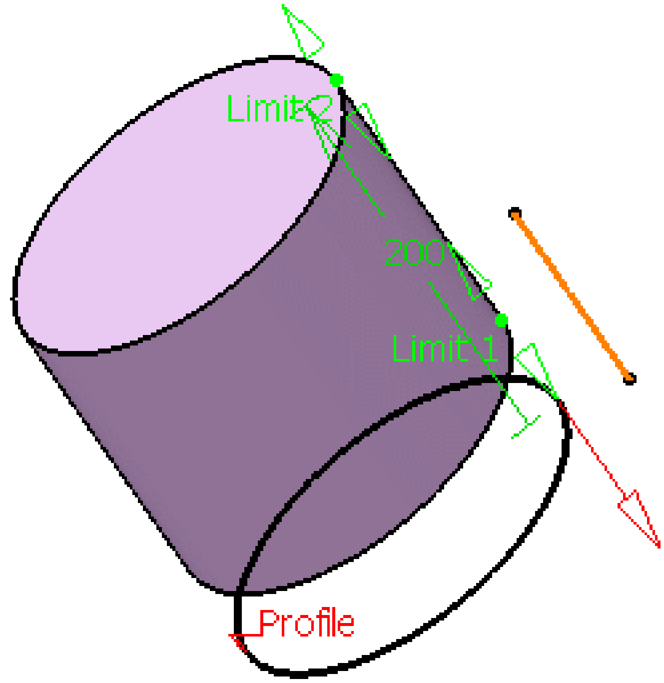
   It can be either a profile or a surface.

The profile must be closed and planar and must not self-intersect.

3. Specify the **Direction** of extrusion.

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.

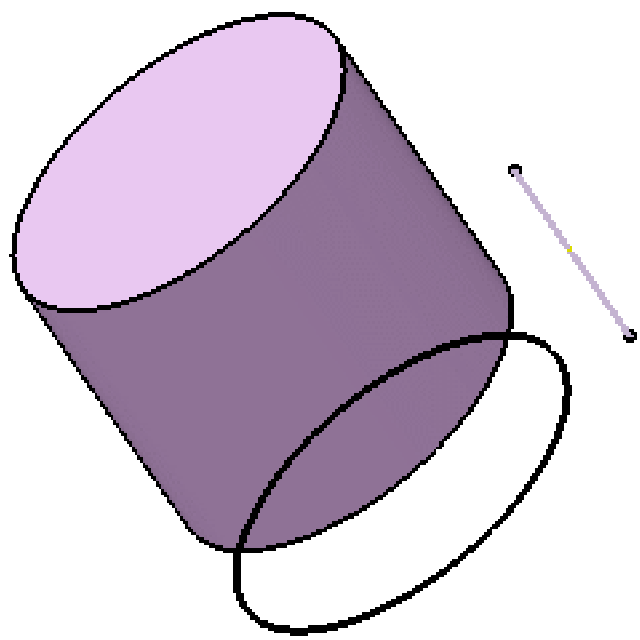


The direction must not be tangent (locally or not) to the profile to be extruded.

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

5. Click OK to create the volume.

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



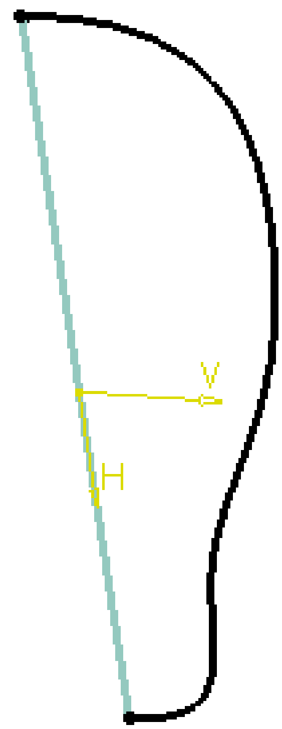
You can click the **Reverse Direction** button to display the extrusion on the other side of the selected profile or click the red arrow in the 3D geometry.

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

Creating Revolution Volumes

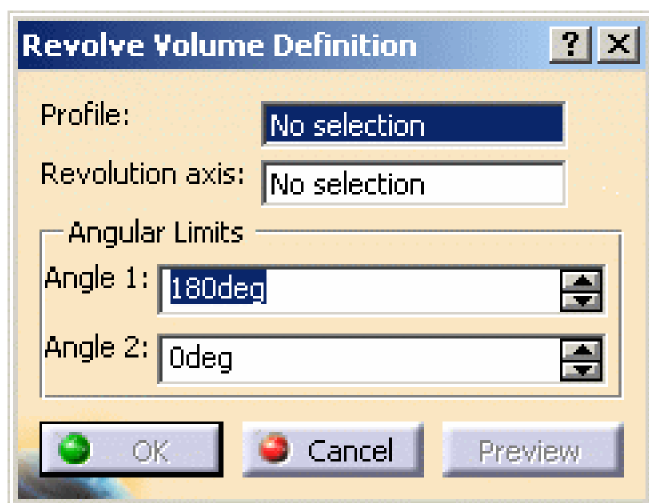
This task shows how to create a surface by revolving a planar profile about an axis.

Open the RevolutionVolume1.CATPart document.



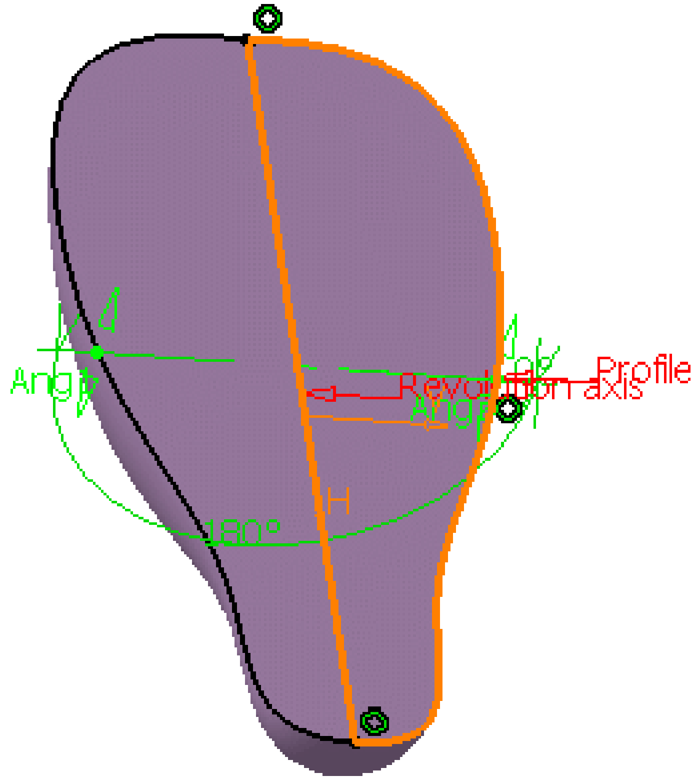
1. Click the **Volume Revolve** icon .

   The Revolution Volume Definition dialog box appears.



2. Select the **Profile**.

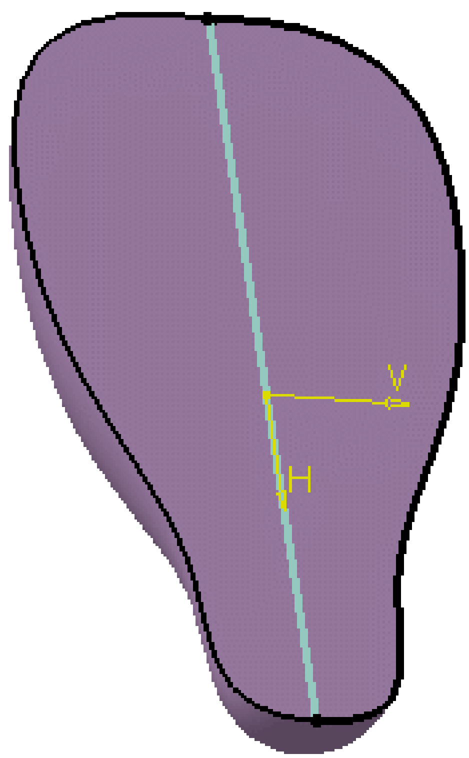
- The profile must be planar, closed or closed on the axis of the sketch.
- There must be no intersection between the axis and the profile. However, if the result is topologically consistent, the surface will still be created.
- The profile must not be perpendicular to the revolution axis.
- If the profile 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.



3. Select a line indicating the desired **Revolution axis**.

   It can be a line or the axis of a sketch.

4. Enter angle values or use the graphic manipulators to define the angular limits of the revolution volume.



5. Click OK to create the surface.

   The volume (identified as Volume Revolve.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.

Creating Multi-Sections Volumes

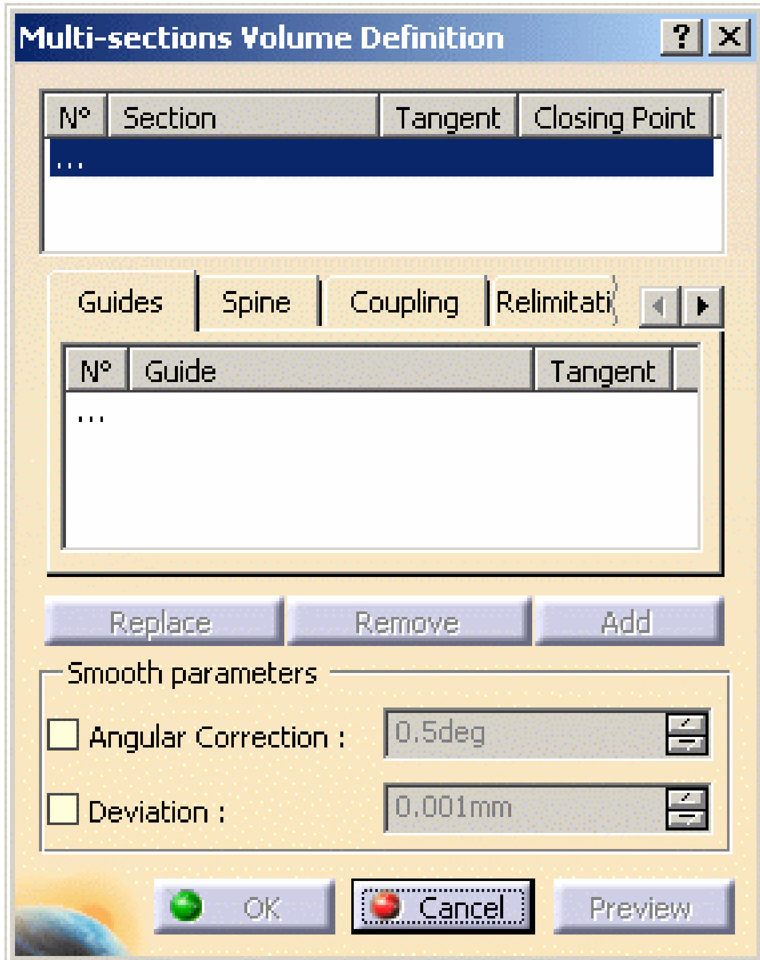
P2

This task shows how to create a multi-sections volume by sweeping two or more **closed** section curves along an automatically computed or user-defined spine. The volume can be made to respect one or more guide curves.

Open the VolumeLoft1.CATPart document.

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

The Multi-sections Volume Definition dialog box appears.

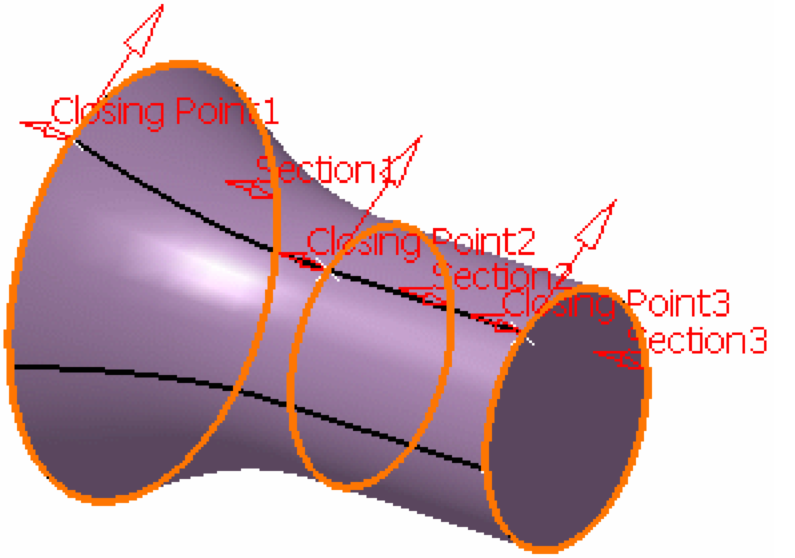


Example of a multi-sections volume defined by three planar sections:

2. Select two or more planar section curves.

The curves must be continuous in point.

A closing point can be selected for a closed section curves.



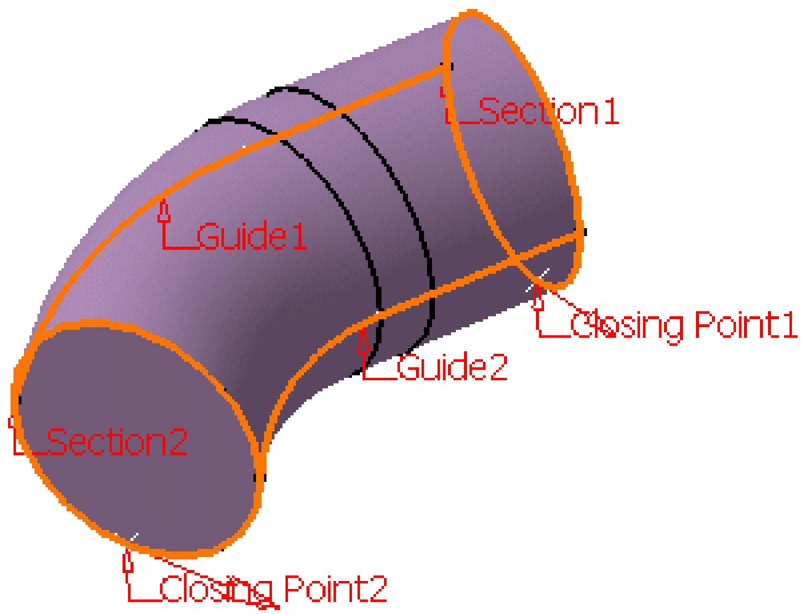
3. If needed, select one or more guide curves.

Example of a multi-sections volume defined by 2 planar sections and 2 guide curves:

Guide curves must intersect each section curve and must be continuous in point.

The first guide curve will be a boundary of the multi-sections volume if it intersects the first extremity of each sections curve.

Similarly, the last guide curve will be a boundary of the multi-sections volume if it intersects the last extremity of each section curve.



You can make a multi-sections volume tangent to an adjacent volume by selecting an end section that lies on the adjacent volume. In this case, the guides must also be tangent to the volume.

In Figure 2 a multi-sections volume tangent to the existing volume has been created:

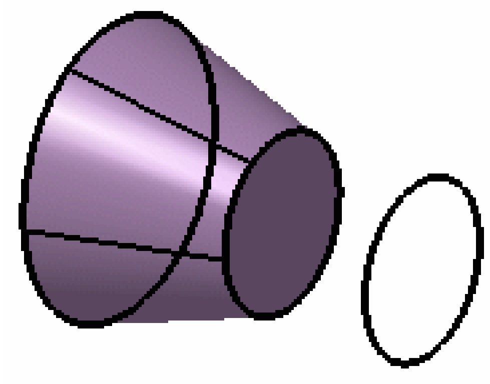


Figure 1

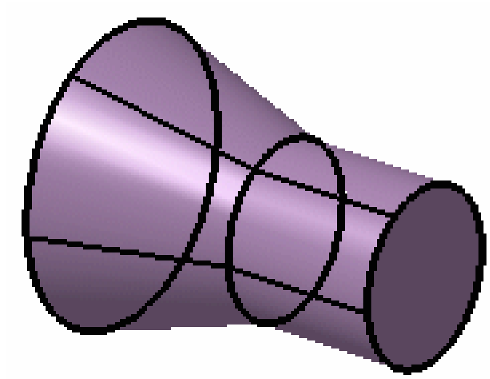


Figure 2

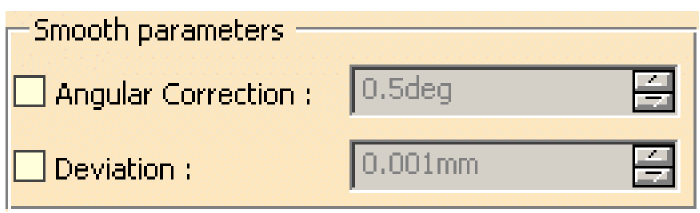
You can also impose tangency conditions by specifying a direction for the tangent vector (selecting a plane to take its normal, for example). This is useful for creating parts that are symmetrical with respect to a plane. Tangency conditions can be imposed on the two symmetrical halves.
Similarly, you 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 volume.

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

Note that the spine curve must be normal to each section plane and must be continuous in tangency.

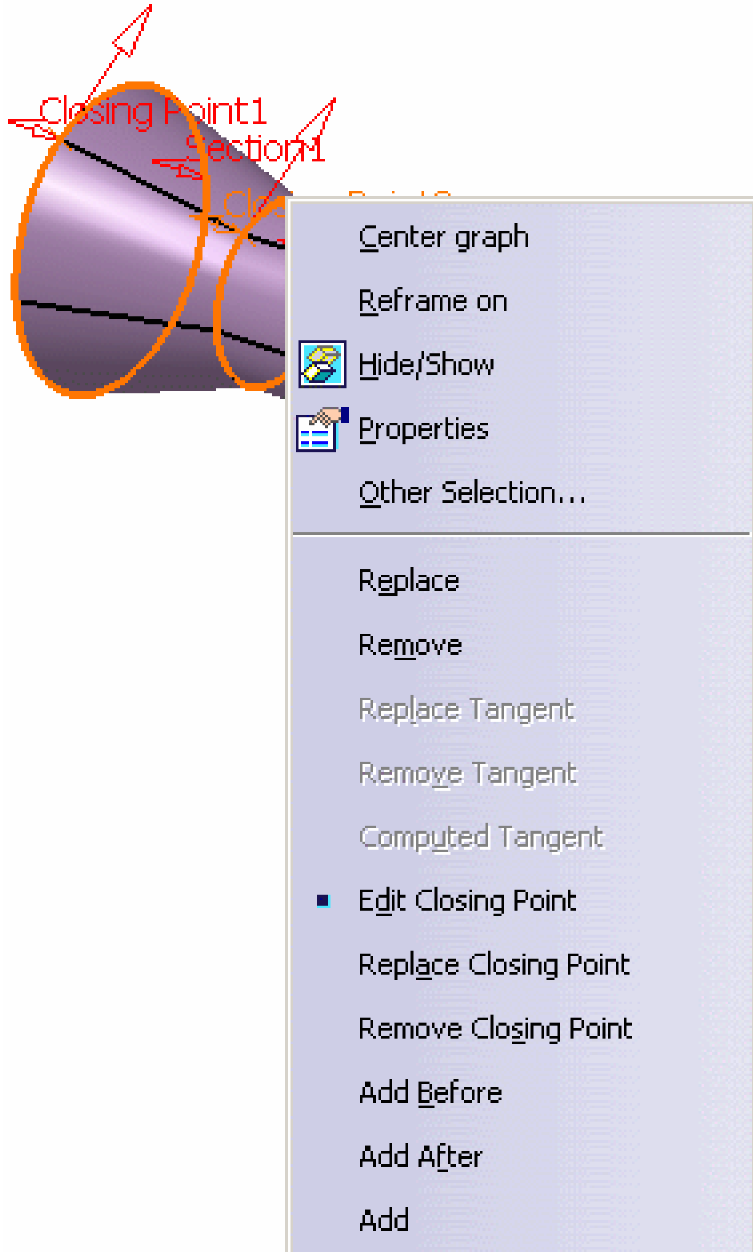
In the Smooth parameters section, you can check:

- 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-sections volume.



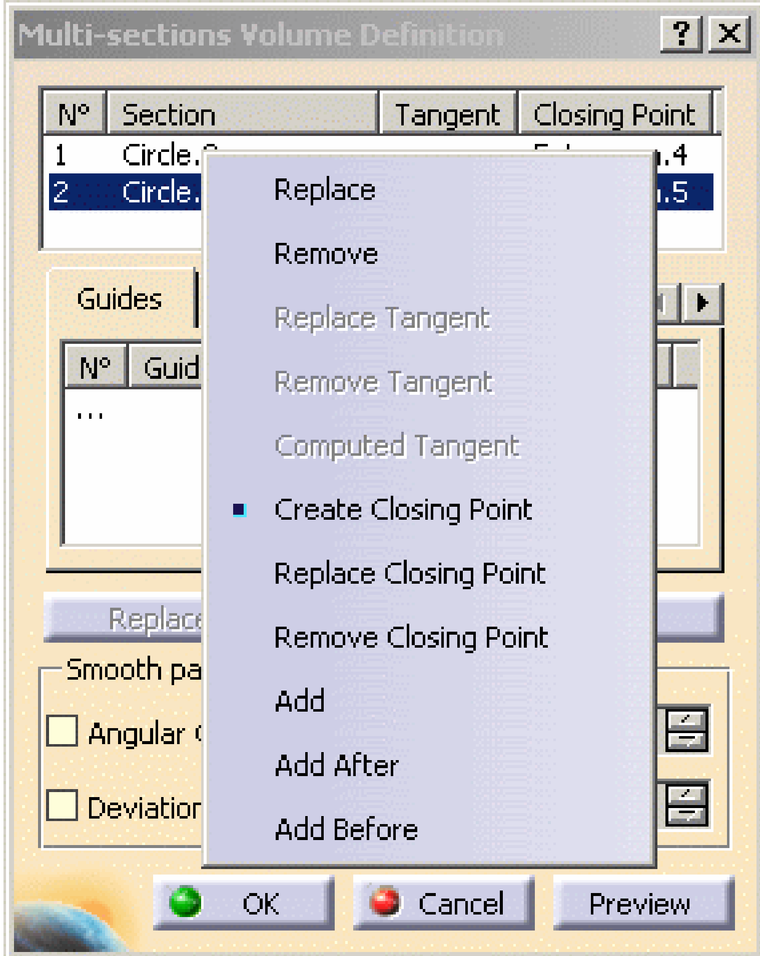
- the **Deviation** option to smooth the lofting motion by deviating from the guide curve(s).

5. It is possible to edit the multi-sections volume reference elements by first selecting a curve in the dialog box list, or by selecting the text on the figure, 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 volumes and closing points.



6. Click **OK** to create the multi-sections volume.

The volume (identified as Multi-sections Volume.xxx) is added to the specification tree.

For further information about the other tabs, please refer to the Creating Multi-Sections Surfaces chapter.

Creating Swept Volumes

This task shows how to create swept volumes that use an explicit or an implicit circular profile.

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

Explicit Profile

The following sub-types are available:

- With reference surface
- With two guide curves
- With pulling direction

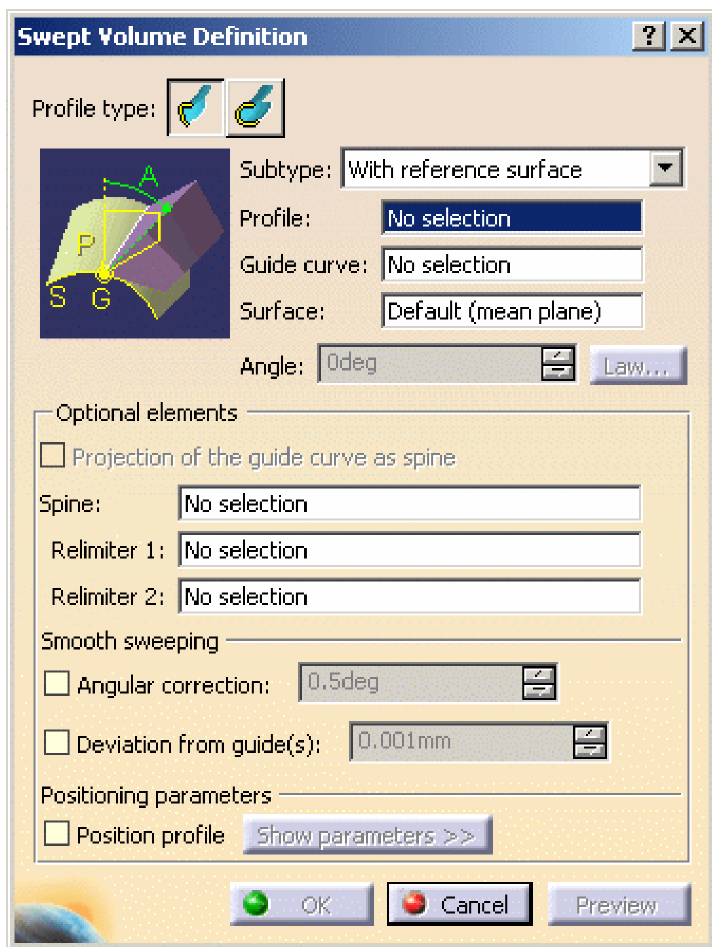
Open the VolumeSweep1.CATPart document.

1. Click the **Volume Sweep** icon .

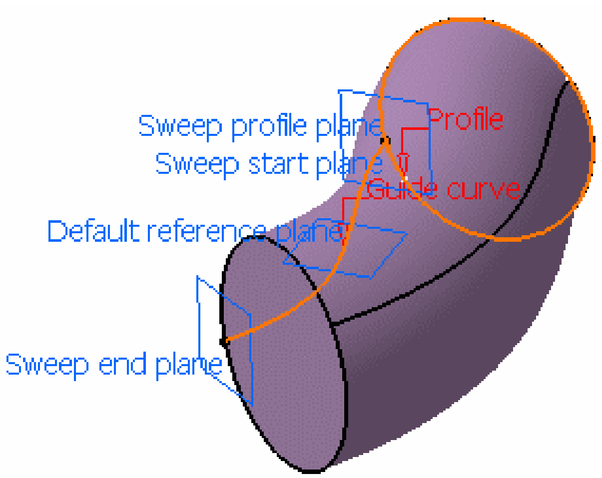
The Swept Volume Definition dialog box appears.

2. Click the **Explicit** profile icon, then use the drop-down list to choose the subtype.

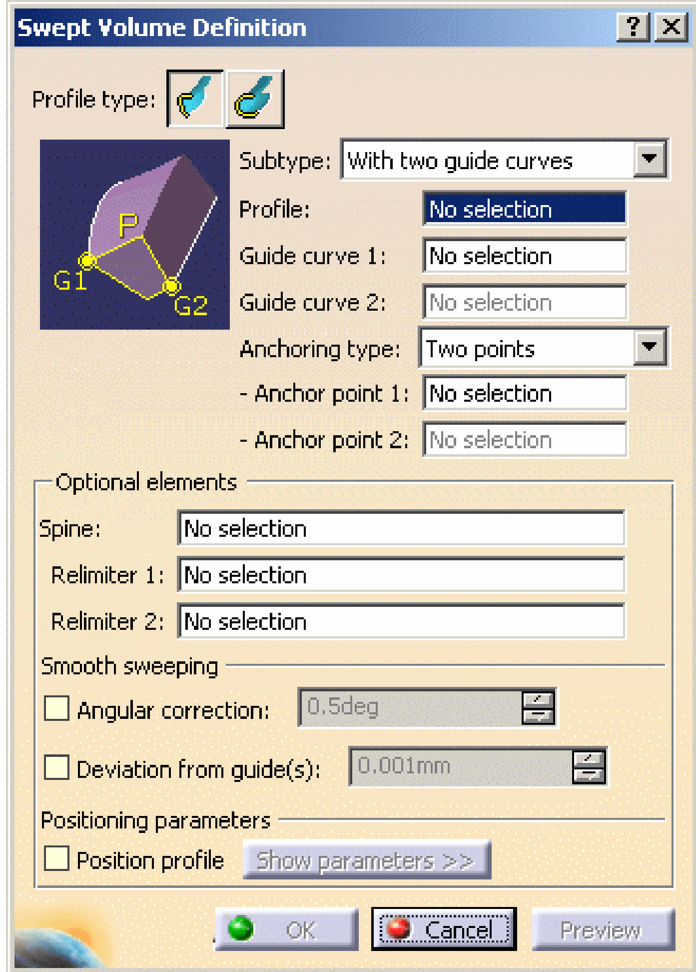
With reference surface



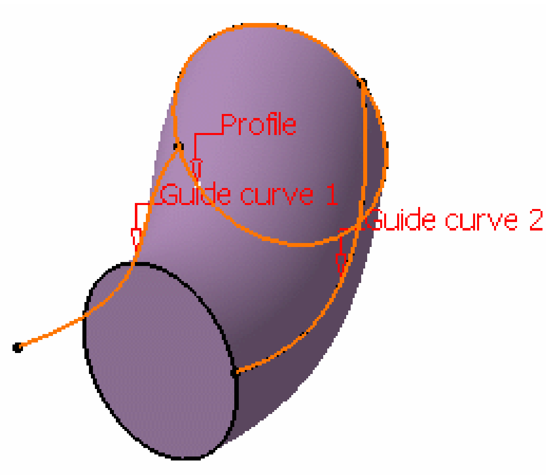
- Select the **Profile** to be swept out (DemoProfile1).
- Select a **Guide curve** (DemoGuide1).
- Select a surface (by default, the reference surface is the mean plane of the spine) in order to control the position of the profile during the sweep.
  Note that in this case, the guiding curve must lie completely on this reference surface, except if it is a plane. You can impose an **Angle** on this surface.



With two guide curves



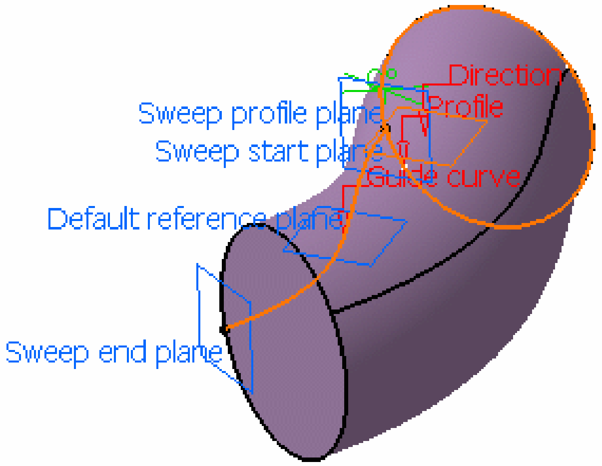
- Select the **Profile** to be swept out (DemoProfile1).
- Select a first **Guide curve** (DemoGuide1).
- Select a second **Guide curve** (DemoGuide2).

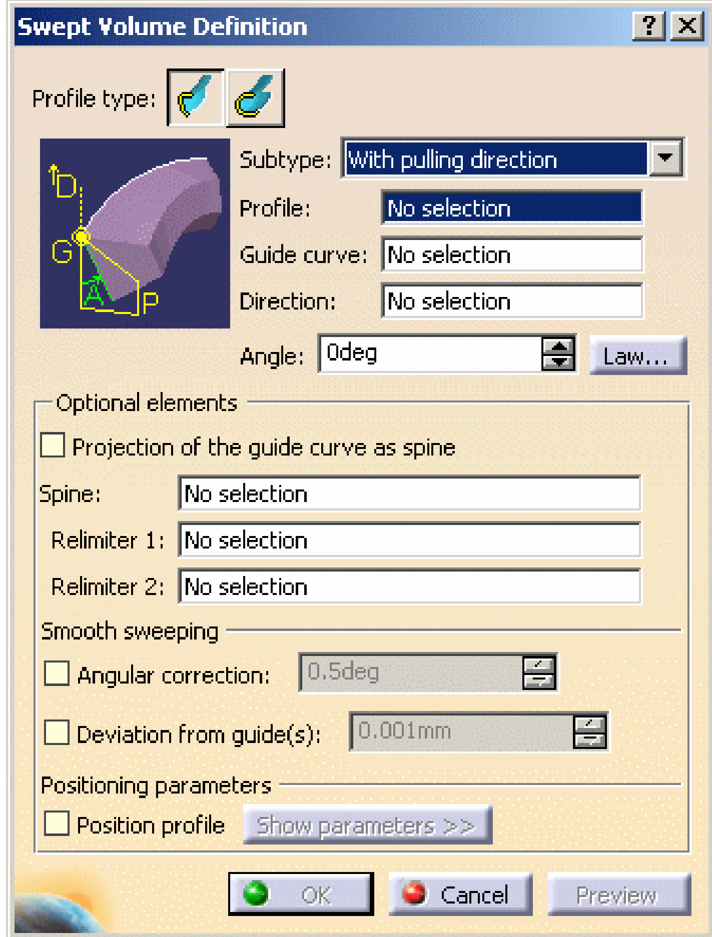


With pulling direction

The With pulling Direction subtype is equivalent to the With reference surface subtype with a reference plane normal to the pulling direction.

- Select the **Profile** to be swept out (DemoProfile1).
- Select a first **Guide curve** (DemoGuide1).
- Select a **Direction** (xy plane)





Circular Profile

The following subtypes are available:

- Center and two angles
- Center and radius

Open the VolumeSweep2.CATPart document.

1. Click the **Volume Sweep** icon.

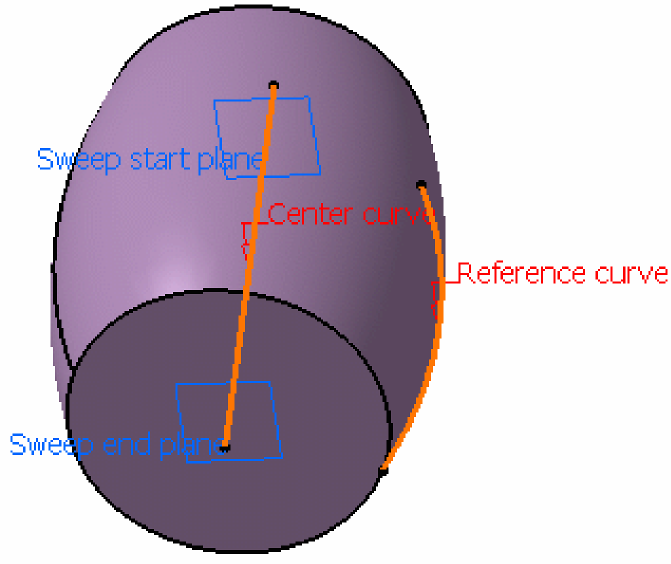
The Swept Volume Definition dialog box appears.

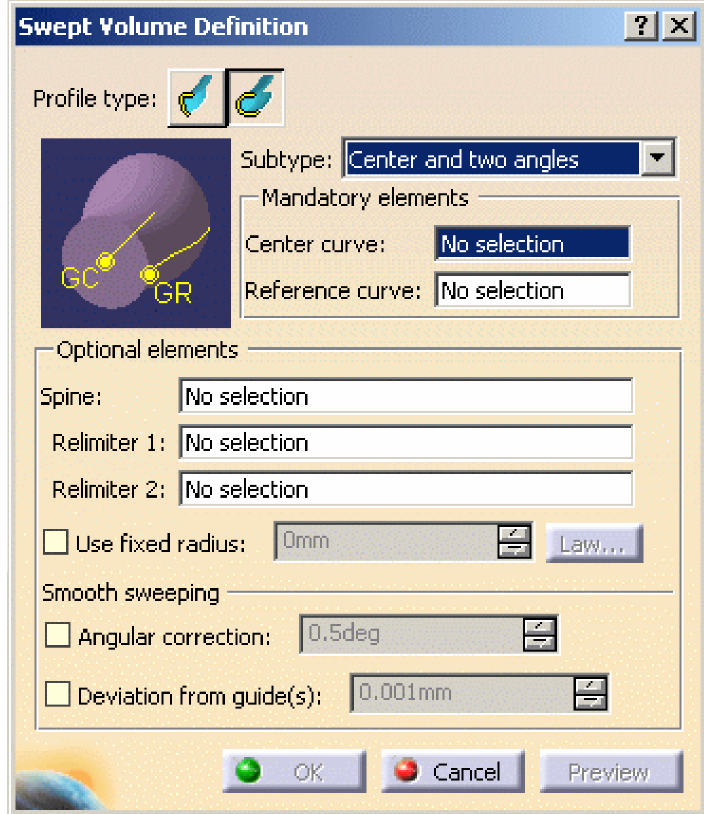
2. Click the **Circle** profile icon, then use the drop-down list to choose the subtype.

Center and two angles

- Select a **Center Curve** (DemoCurve1) and a **Reference curve** (DemoCurve2).

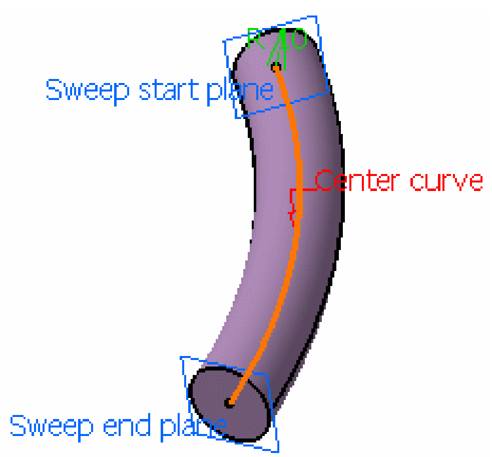
Angles cannot be modified and have fixed values, that is 0 deg and 360 deg.

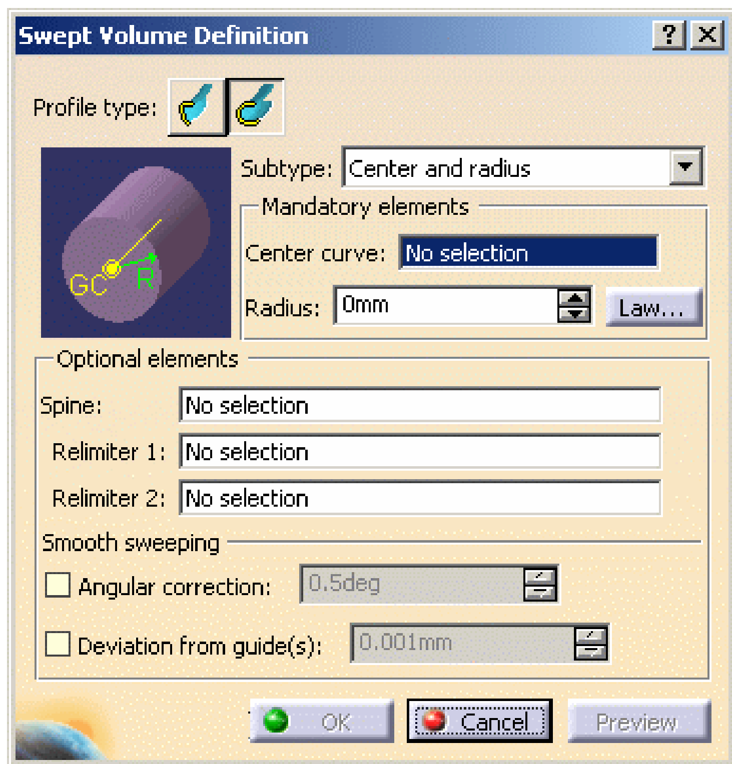




Center and radius

- Select a **Center Curve** (DemoCurve2) and enter a **Radius** value (10mm).





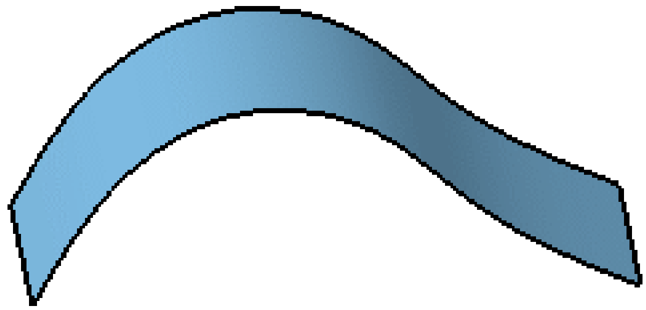
For further information about the optional elements, please refer to the Creating Swept Surfaces Using an Explicit Profile and Creating Swept Surfaces Using a Circular Profile chapters.

Creating a Thick Surface

This task shows you how to add material to a surface in two opposite directions.

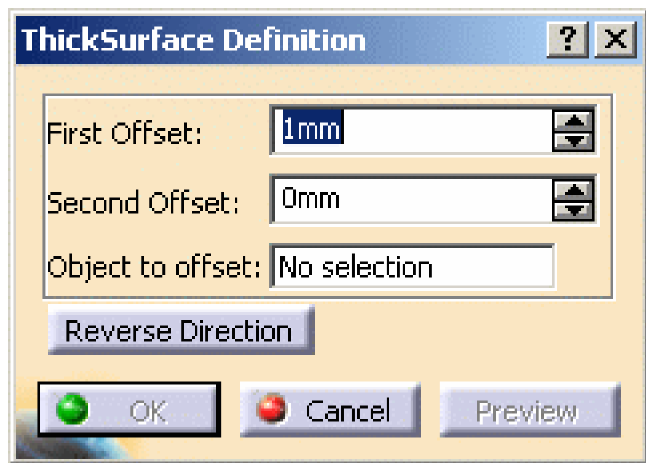
Open the ThickSurface1.CATPart document.

1. Select the surface you wish to thicken, that is the extrude element.

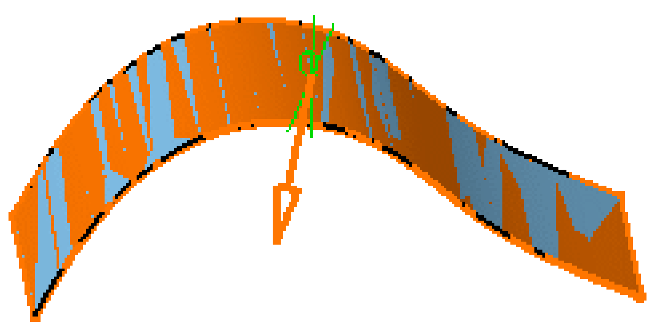


2. Click the **Thick Surface** icon.

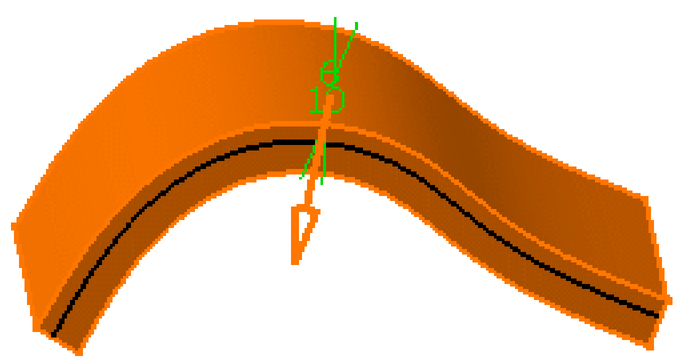
The ThickSurface Definition dialog box opens.



In the geometry area, the red arrow that appears on the extrude element indicates the first offset direction. If you need to reverse the arrow, just click on it or click the **Reverse Direction** button in the dialog box.



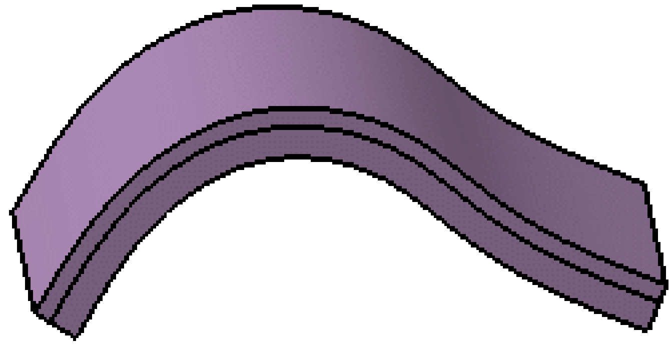
3. Enter 10mm as the **First Offset** value and 6mm as the **Second Offset** value.



4. Click OK.

   The surface is thickened. The operation (identified as ThickSurface.x) is added to the specification tree.

   Note that the resulting feature does not keep the color of the original surface, but is displayed in purple indicating it is a volume.

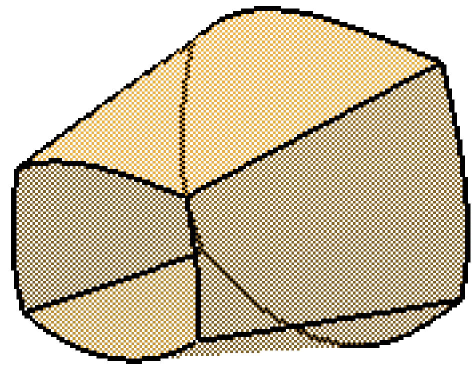


Creating a Close Surface

This task shows you how to close surfaces.

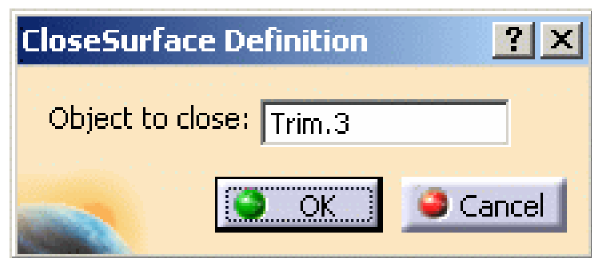
Open the CloseSurface1.CATPart document.

1. Select the surface to be closed.



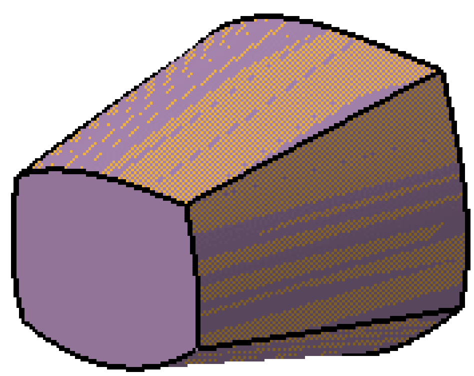
2. Click the **Close Surface** icon .

   The CloseSurface Definition dialog box opens.



3. Click OK.

   The surface is closed. The operation (identified as CloseSurface.x) is added to the specification tree.



   Note that the resulting feature does not keep the color of the original surface, but is displayed in purple indicating it is a volume.

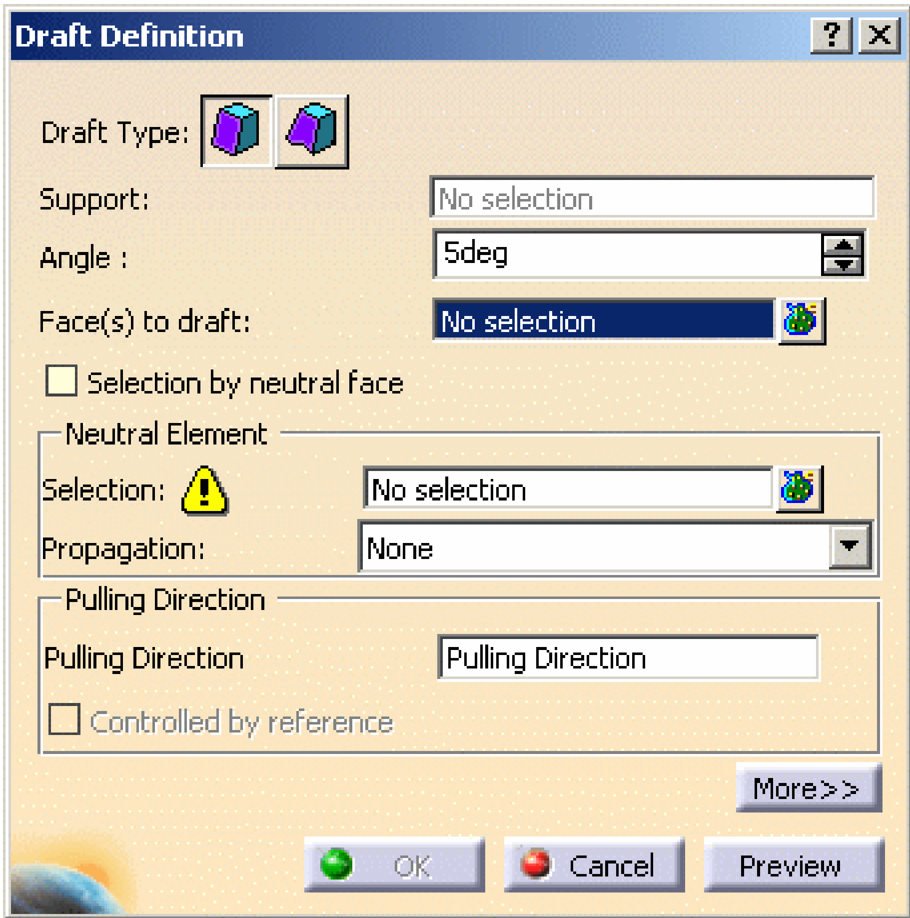
Creating a Draft

Drafts are defined on molded parts to make them easier to remove from molds.
There are two ways of determining the objects to draft: either by explicitly selecting the object or by selecting the neutral element, which makes the application detect the appropriate faces to use.

This task shows you how to create a basic draft by selecting the neutral element.

Open the Draft1.CATPart document.

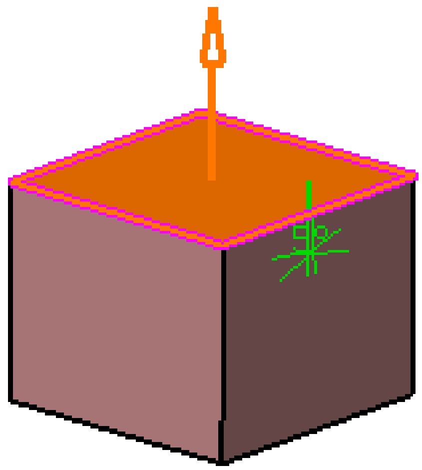
1. Click the **Draft Angle** icon from the Volume drafts sub-toolbar.



The Draft Definition dialog box is displayed and an arrow appears on a plane, indicating the default pulling direction. This dialog box displays the constant angle draft option as activated. If you click the icon to the right, you then access the command for creating variable angle drafts.

2. Check the **Selection by neutral face** option to determine the selection mode.

3. Select the upper face as the neutral element. This selection allows the application to detect the face to be drafted.



The neutral curve is displayed in pink. The faces to be drafted are in dark red.

The **Support** field is filled with the volume owning the selected face.

4. Set the Propagation option:

- **None**: there is no propagation
- **Smooth**: the application integrates the faces propagated in tangency onto the neutral face to define the neutral element.
  For more about the neutral element, refer to A Few Notes about Drafts.

5. Define the Propagation option..

By default, it is normal to the neutral face and is displayed on top of the part.

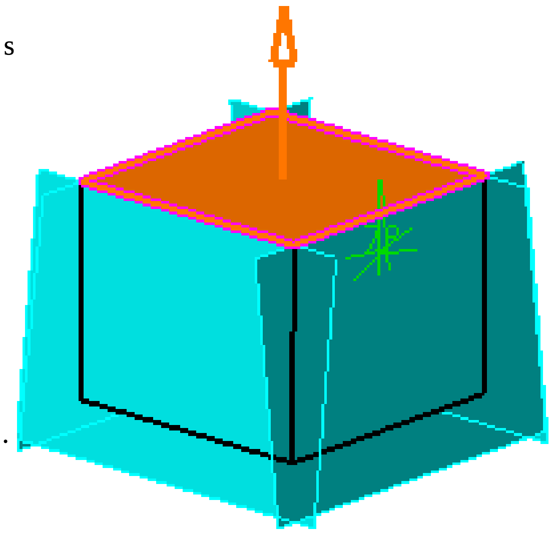
The **Controlled by reference** option is now activated, meaning that whenever you will edit the element defining the pulling direction, you will modify the draft accordingly.
Note that when using the other selection mode (explicit selection), the selected objects are displayed in dark pink.

6. The default angle value is 5. Enter 7 degrees as the new angle value.

   The application displays the new angle value in the geometry.

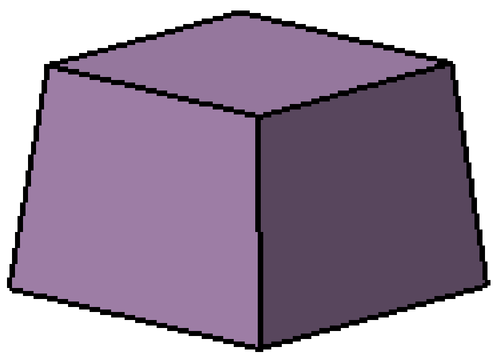
7. Click Preview to see the draft to be created.

   It appears in blue.



8. Click OK to confirm the operation.

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



For further information about drafts, refer to the Creating Basic Drafts and Creating Drafts with Parting Elements chapters in the *Part Design* documentation.

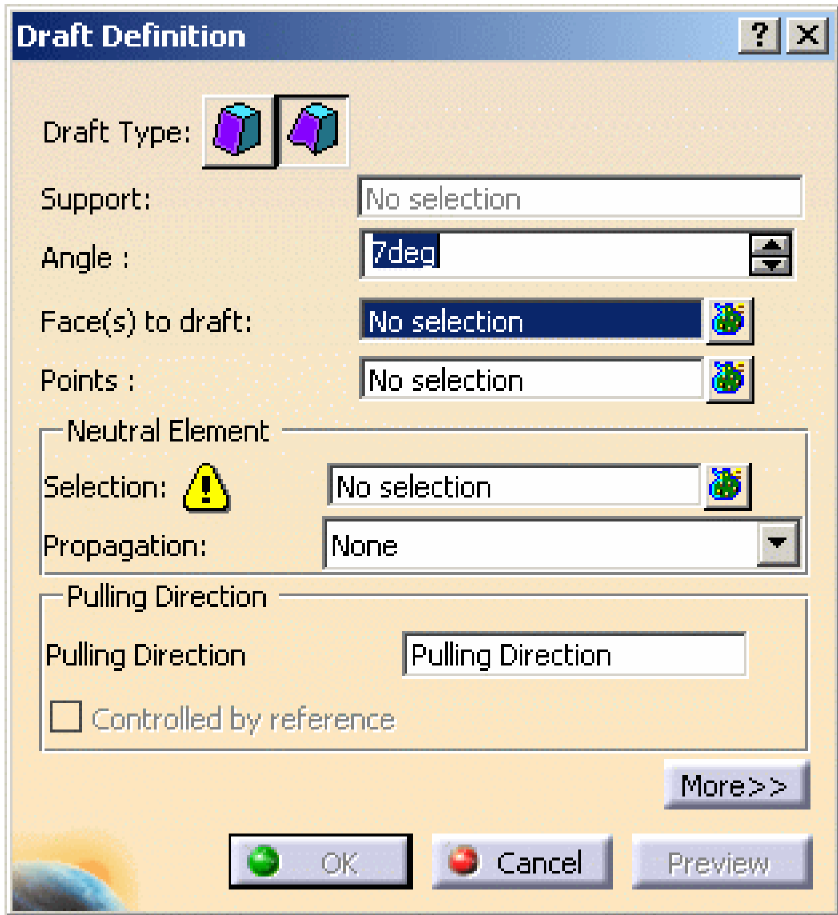
Creating a Variable Angle Draft

Drafts are defined on molded parts to make them easier to remove from molds.
There are two ways of determining the objects to draft: either by explicitly selecting the object or by selecting the neutral element, which makes the application detect the appropriate faces to use.

Sometimes, you cannot draft faces by using a constant angle value. This task shows you an another way of drafting: by using different angle values.

Open the Draft1.CATPart document.

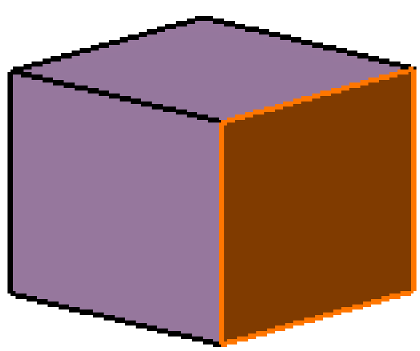
1. Click the **Draft Angle** icon from the Volume drafts sub-toolbar.



As an alternative, you can use the **Draft Angle** command, then click the **Variable Angle Draft** icon available in the dialog box. For more information, see Creating a Draft.

The Draft Definition dialog box appears, displaying the variable angle draft option as activated. If you click the icon to the left, you then access the command for performing basic drafts.

2. Select the **Face to draft**.

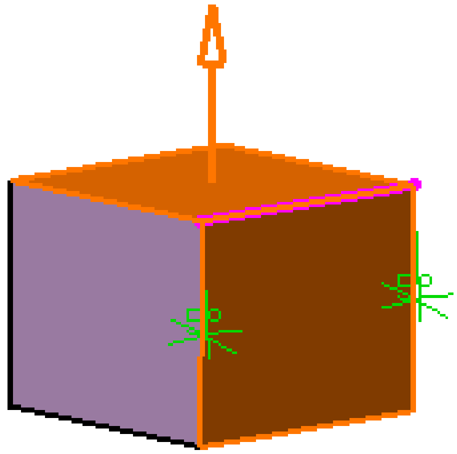


Multi-selecting faces that are not continuous in tangency is not allowed for this command.

3. Select the upper face as the **Neutral Element**.

   An arrow appears on the part, indicating the default pulling direction. The application detects two vertices and displays two identical radius values.

   The **Support** field is filled with the volume owning the selected face.



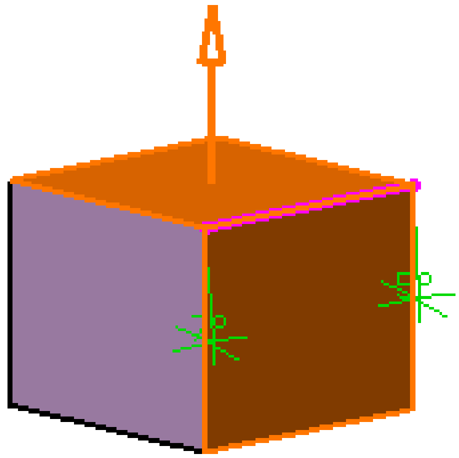
4. Increase the **Angle** value: only one value is modified accordingly in the geometry.

5. To edit the other angle value, select the value in the geometry and increase it in the dialog box. For instance, enter 9.

Alternatively, double-click this value to display the Parameter Definition dialog box, then edit the value.

6. To edit the other angle value, select the value in the geometry and increase it in the dialog box. For instance, enter 9.

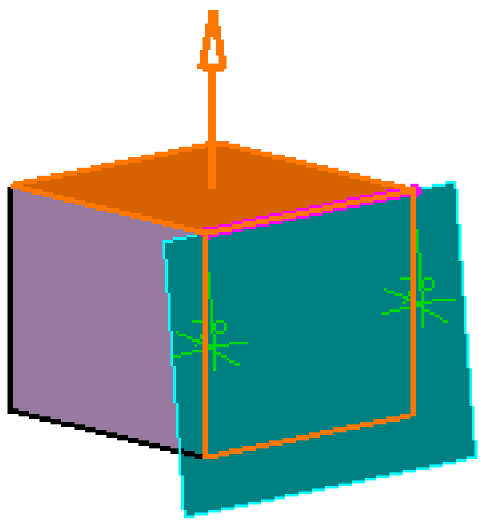
7. Click **Preview** to see the draft to be created.



8. Click the **Points** field to add a point.

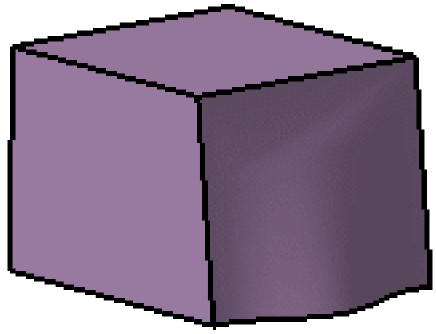
9. Click a point on the edge.

10. Enter a new angle value for this point: for example, enter 17. The new radius value is displayed.



11. Click OK to confirm the operation.

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



For further information about drafts, refer to the Creating Basic Drafts and Creating Drafts with Parting Elements chapters in the *Part Design* documentation.

Creating a Draft from Reflect Lines

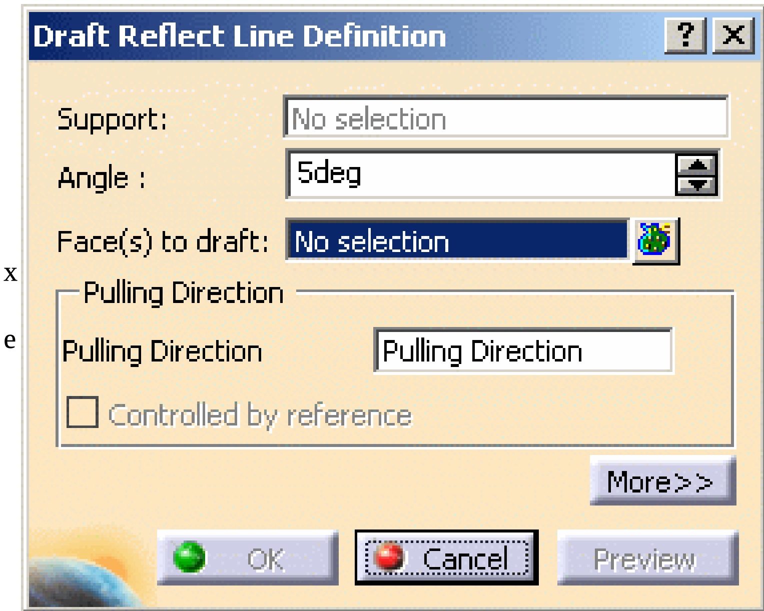
P2

This task shows you how to draft a face by using reflect lines as neutral lines from which the resulting faces will be generated. In this scenario, you will also trim the material to be created by defining a parting element.

Open the VolumeDraft2.CATPart document.

1. Click the **Draft Reflect Line** icon from the Volume drafts sub-toolbar.

The Draft Reflect Line Definition dialog box is displayed and an arrow appears, indicating the default pulling direction. The default direction is normal to the face.

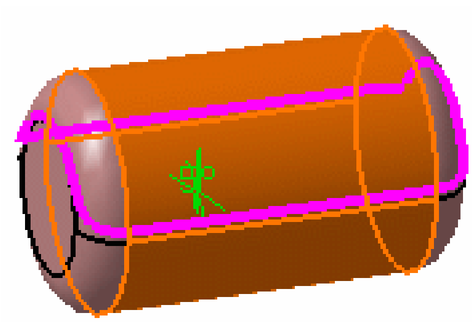


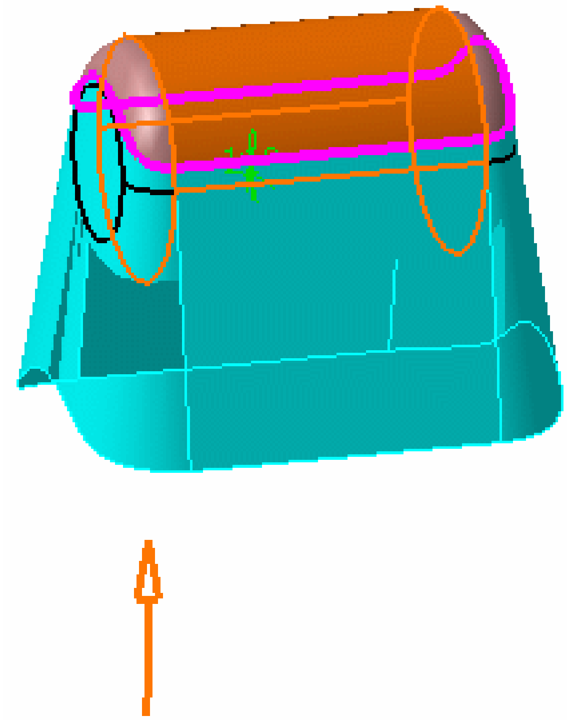
Clicking the arrow reverses the direction.

2. Select the cylinder.

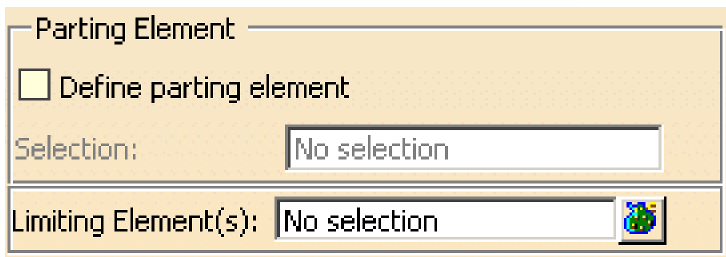
The application detects one reflect line and displays it in pink. This line is used to support the drafted faces.

The **Support** field is filled with the volume owning the selected face.



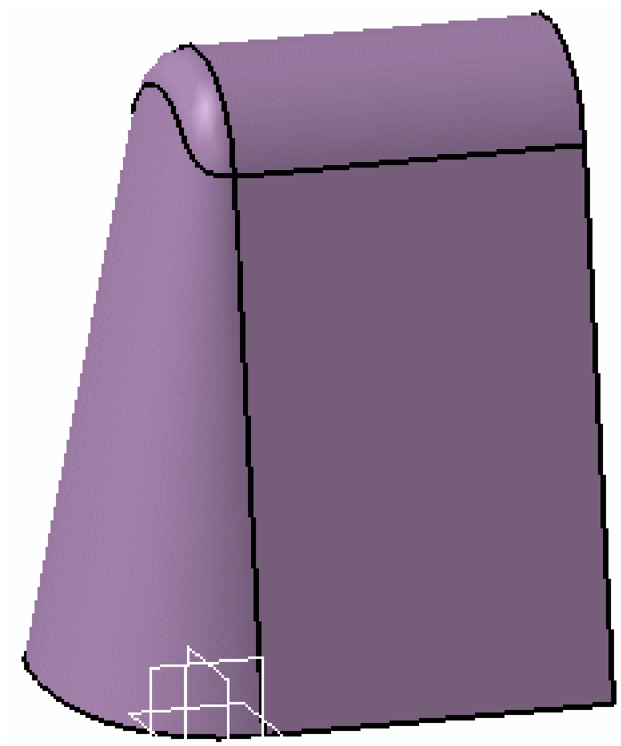


3. Enter an angle value in the Angle field. For example, enter 11. The reflect line is moved accordingly.
4. Click **Preview** to get an idea of what the draft will look like.



5. Click **More>>** to access further options.
6. Check the **Define parting element** option and select plane zx as the parting element.
7. Click OK to confirm the operation.

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



For further information about limiting elements, refer to the Creating Basic Drafts chapter in the *Part Design* documentation.

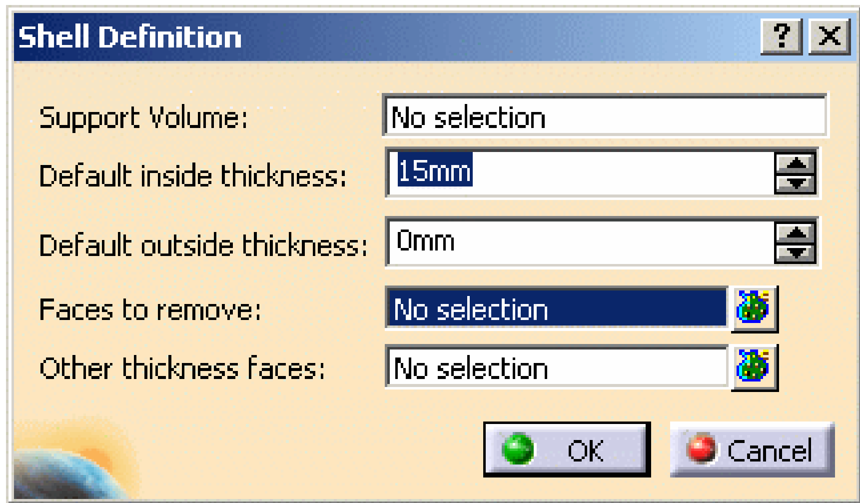
Creating a Shell

Shelling a feature means emptying it, while keeping a given thickness on its sides. Shelling may also consist in adding thickness to the outside. This task shows how to create a cavity.

Open the VolumeShell1.CATPart document.

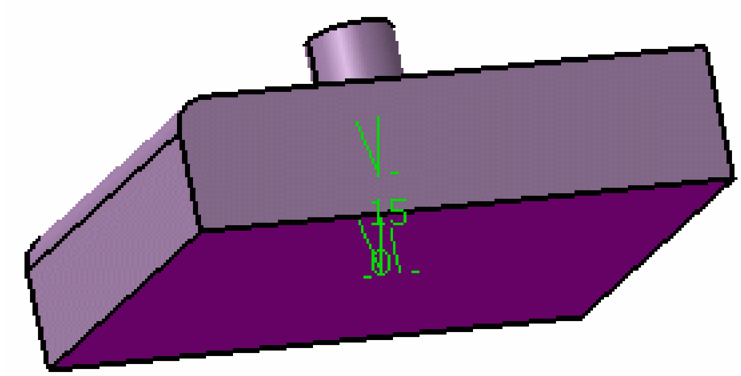
1. Click the **Shell** icon.

   The Shell Definition dialog box is displayed.



2. Select the **Face to remove**.

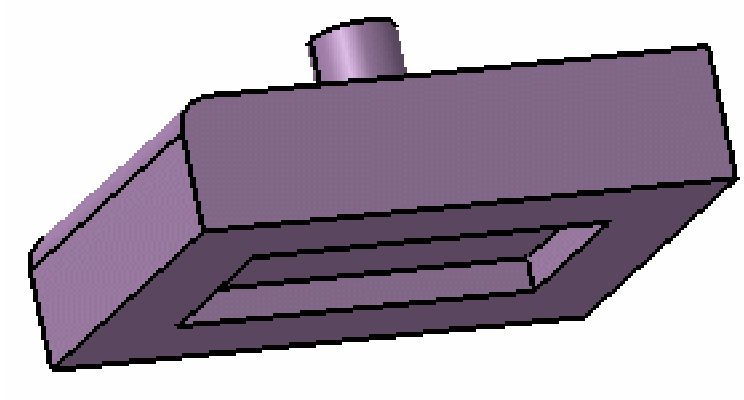
   The **Support Volume** field is filled with the volume owning the selected face.



3. Enter 15mm in the **Default inside thickness** field.

4. Click OK.

   The feature is shelled: the selected face is left open. This element (identified as Shell.xxx) is added to the specification tree.



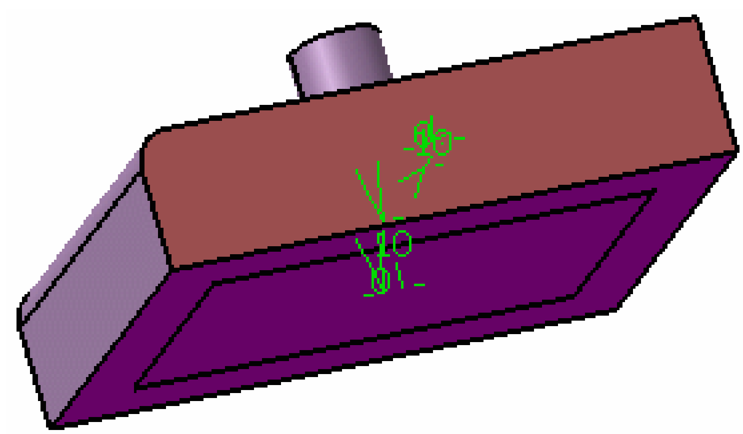
5. Double-click the shell to edit it.

6. Click the **Other thickness faces** field.

7. Double-click the thickness value displayed on this face.

   10. In the dialog box that appears, enter 10mm and click OK.

   The length between the selected face and the shell is 10mm.



For further information about shells, please refer to the Creating Shells chapter in the *Part Design* documentation.

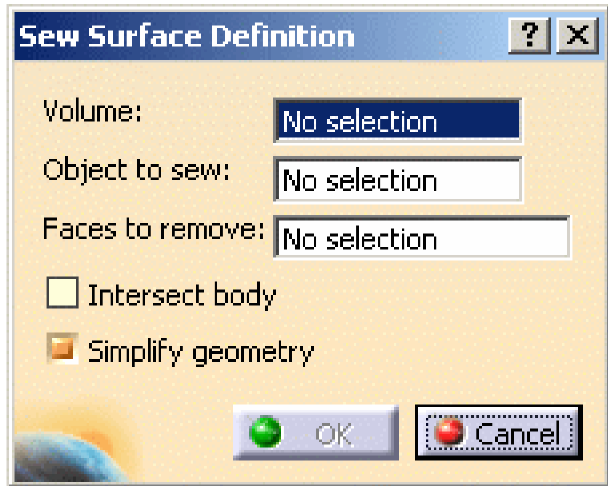
Creating a Sew Surface

Sewing is a Boolean operation combining a surface with a body. This task shows you how to add or remove material by modifying the surface of the volume.

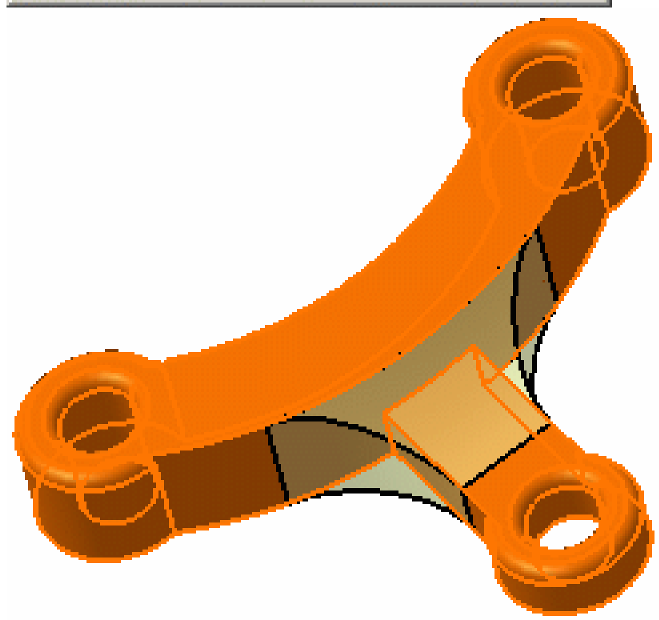
Open the VolumeSew1.CATPart document.

1. Click the **Sew Surface** icon .

   The Sew Surface definition dialog box is displayed.



2. Select the sewing **Volume** (here Add.1).
3. Select the **Object to sew** onto the volume (here Join.1).

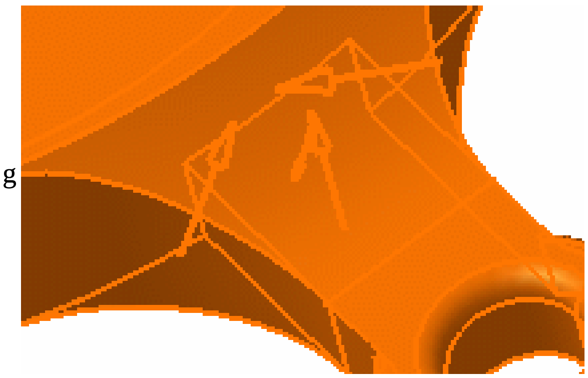


With topology simplification

Keep the **Simplify geometry** option active.

Using this option, if in the resulting volume there are connected faces defined on the same geometric support (faces separated by smooth edges), these faces will be merged into one single face.

Arrows appear indicating the side where material will be added or kept. Note that clicking an arrow reverses the given direction.
The arrows must point towards the volume.



4. Click OK.

   The surface is sewn onto the body. You may notice that the bottom of the volume is made of one single face.

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

   Click OK.

5. To see the simplification, just hide Join.1.

Without topology simplification

6. Double-click SewSurface.1 in the specification tree to edit it and deactivate the **Simplify geometry** option.

7. Click OK.

   The bottom of the volume is made of three connected faces. The smooth edges resulting from the sewing appear because no topological simplification has been performed.

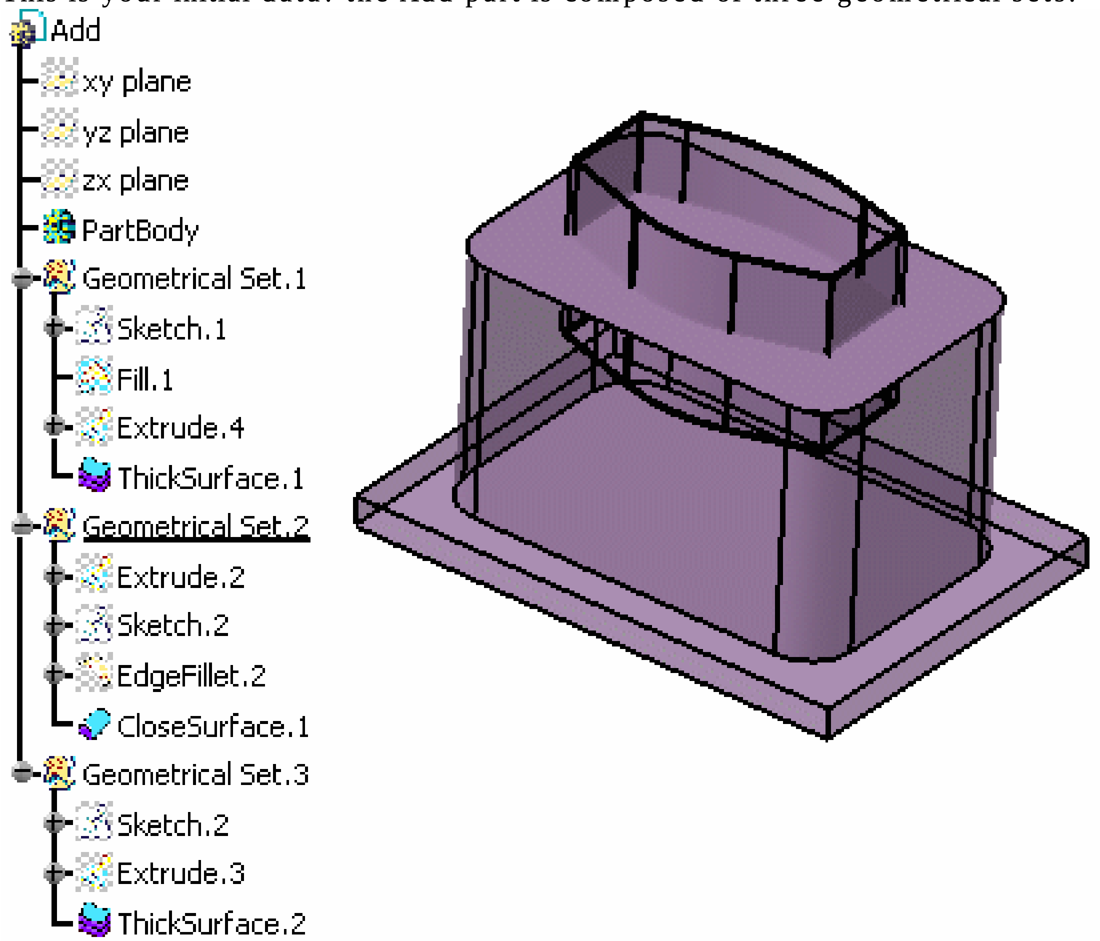
For more information about the **Intersect body** option, please refer to the *Sewing Surfaces* chapter in the Part Design documentation.

Adding Volumes

This task shows how to add a volume to another volume, that is uniting them.

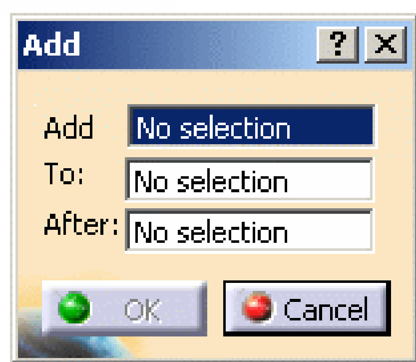
Open the AddVolume1.CATPart document and make sure Geometrical Set.2 is the current body.

This is your initial data: the Add part is composed of three geometrical sets.



1. Click the **Add** icon from the Volumes operations sub-toolbar.

   The Add dialog box opens.



2. Select the volume to operate (ThickSurface.2).

3. Select the destination volume (CloseSurface.1).

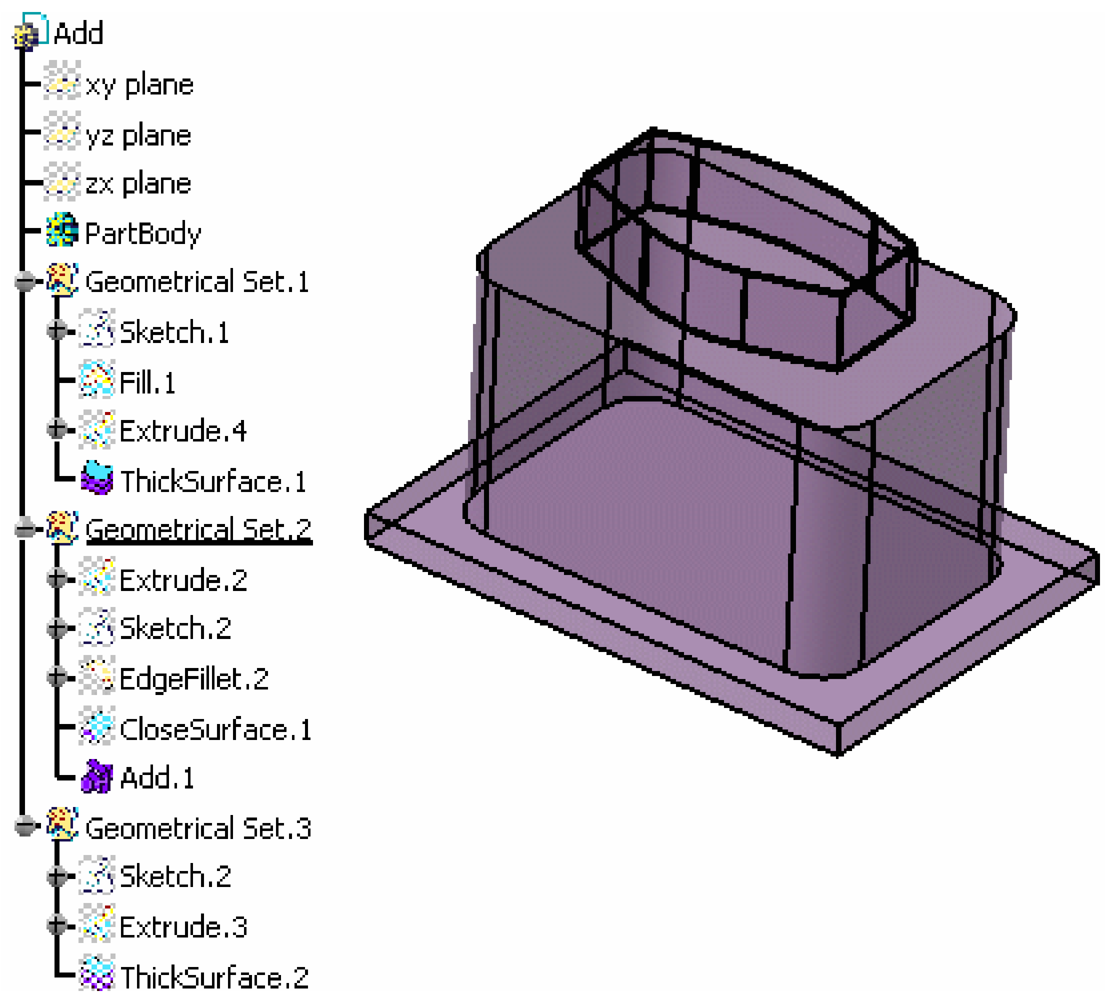
4. Select a destination body after which the added volume will be located.

   If the Geometrical Set or the Ordered Geometrical Set is current, the **After** field is valuated with the current body and will be located after its last feature.

5. Click OK.

   The operation (identified as Add.xxx) is added to the specification tree.

   The specification tree and the Add part now look like this:



You will note that:

- the material common to ThickSurface.2 and CloseSurface.1 has been removed,
- both volumes keep their original colors.

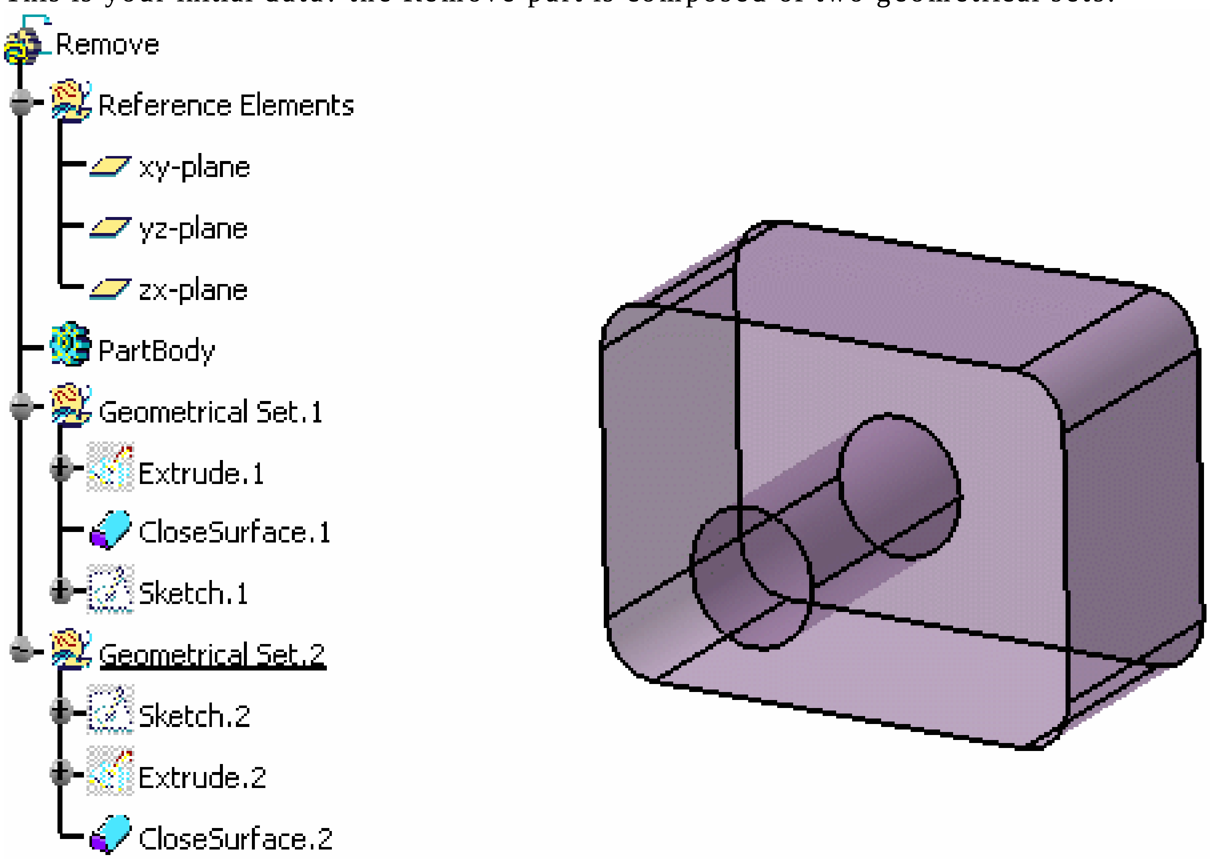
- Multi-selection is not possible.
- You cannot edit the Add operation.

Removing Volumes

This task illustrates how to remove a volume from another volume.

Open the RemoveVolume1.CATPart document.

This is your initial data: the Remove part is composed of two geometrical sets.



1. Click the **Remove** icon from the **Volumes operations** sub-toolbar.

   The Remove dialog box opens.

2. Select the volume to remove (CloseSurface.2).

3. Select the volume from which it is removed (CloseSurface.1).

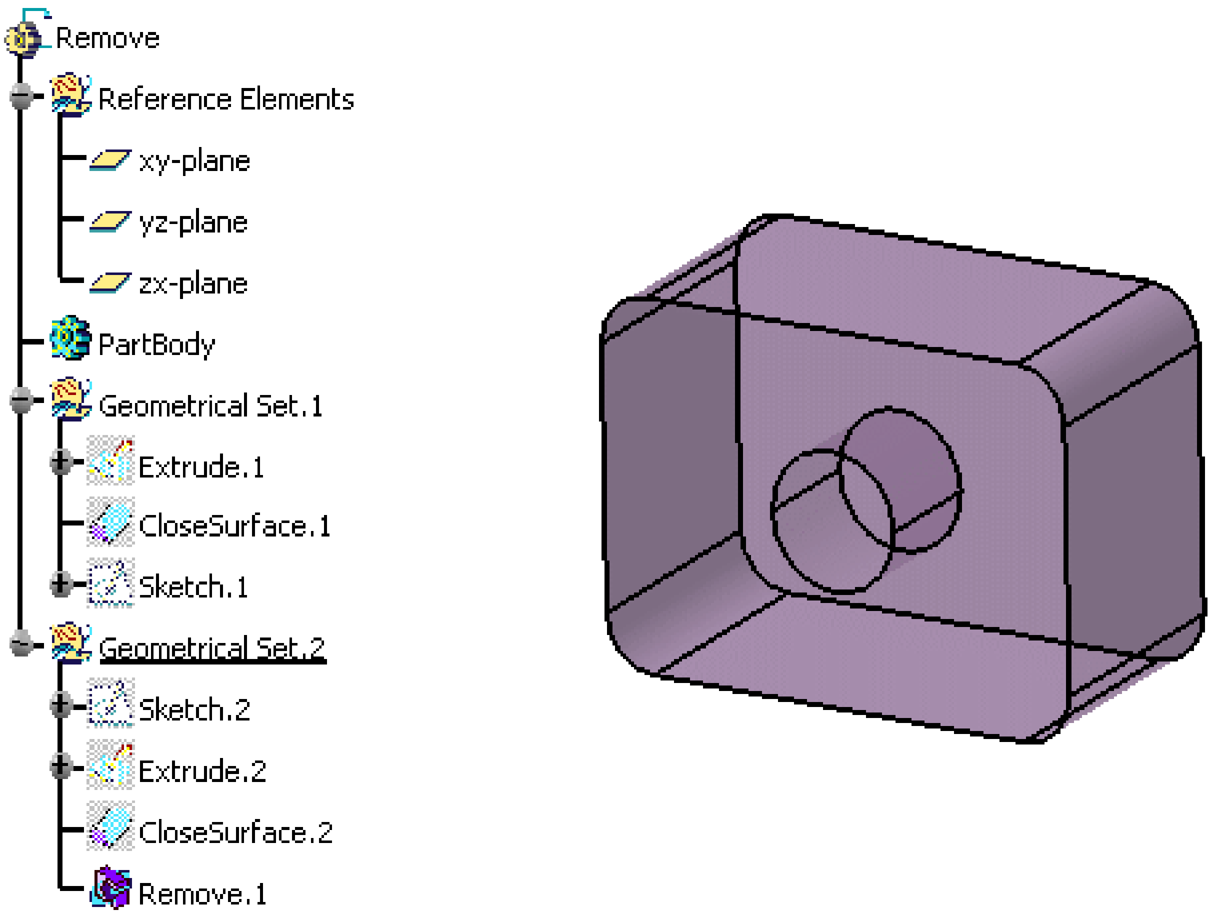
4. Select a destination body after which the removed volume will be located.

   If the Geometrical Set or the Ordered Geometrical Set is current, the **After** field is valuated with the current body and will be located after its last feature.

5. Click OK.

   The operation (identified as Remove.xxx) is added to the specification tree.

   The specification tree and the Remove part now look like this:



- Multi-selection is not possible.
- You cannot edit the Remove operation.

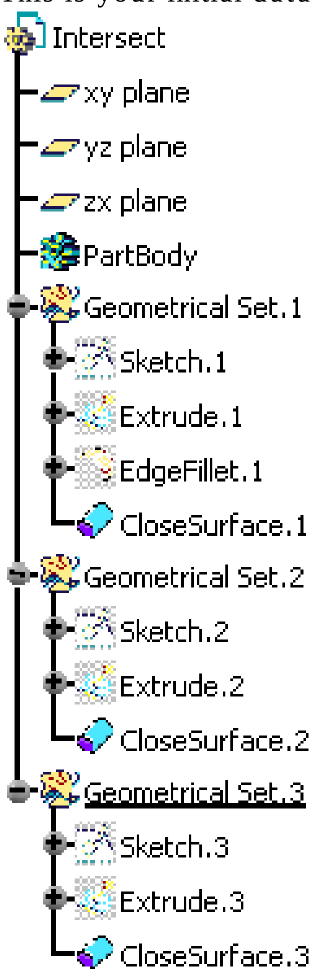
Intersecting Volumes

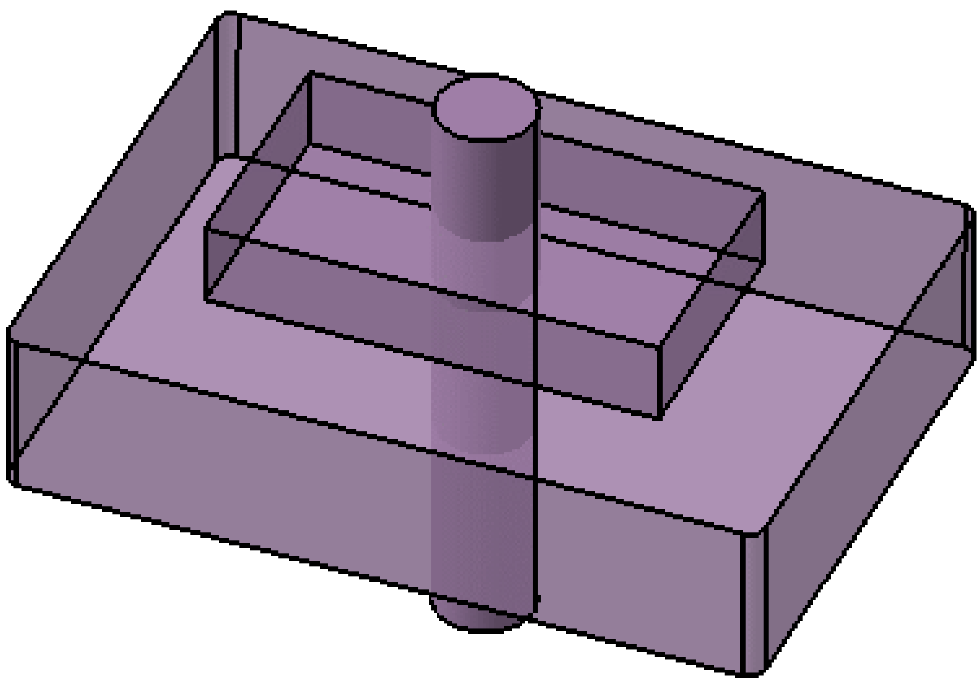
P2

The material resulting from an intersection operation between two volumes is the material shared by these volumes. This tasks illustrates how to compute two intersections.

Open the IntersectVolume1.CATPart document.

This is your initial data: the Intersect part is composed of three geometrical sets.





1. Click the **Intersect** icon from the **Volumes operations** sub-toolbar.

   The Intersect dialog box opens and to lets you determine the second body you wish to use.

2. Select the volume to intersect (CloseSurface.3).

3. Select the volume to be intersected with (CloseSurface.1).

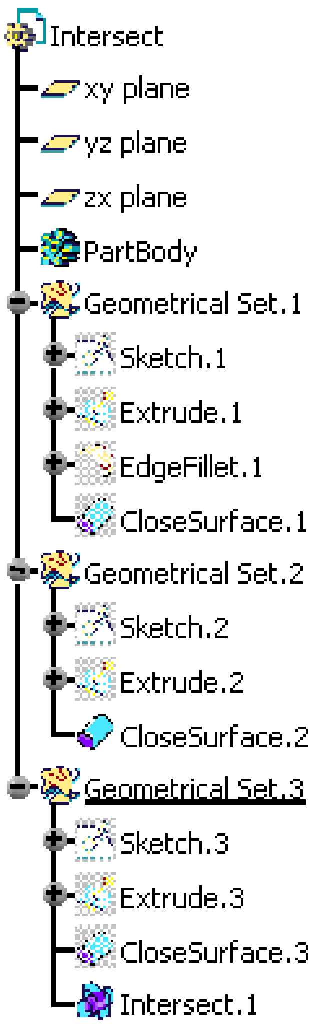
4. Select a destination body after which the intersected volume will be located.

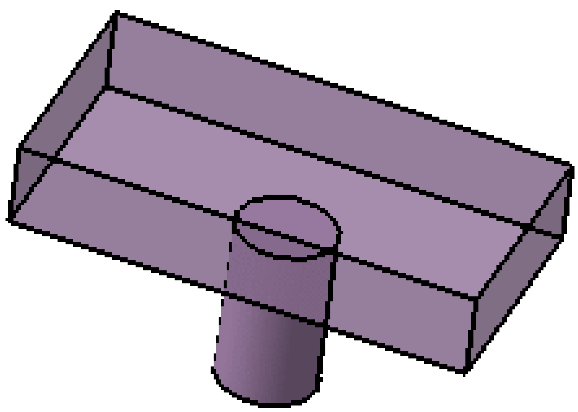
   If the Geometrical Set or the Ordered Geometrical Set is current, the **After** field is valuated with the current body and will be located after its last feature.

5. Click OK.

   The operation (identified as Intersect.xxx) is added to the specification tree.

   The specification tree and the Intersect part now look like this:





- Multi-selection is not possible.
- You cannot edit the Remove operation.

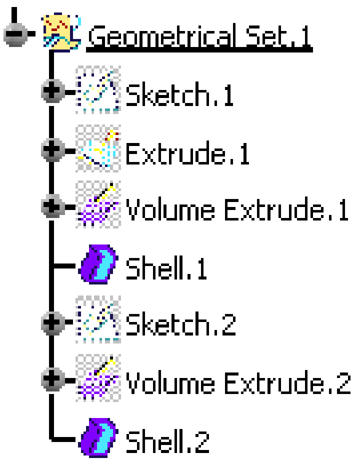
Trimming Volumes

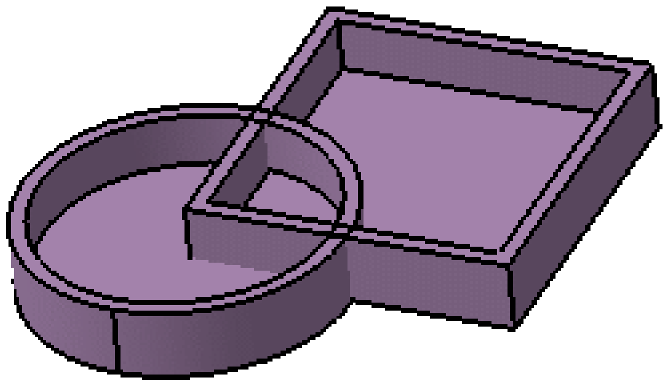
P2

This task shows how to trim a volume to define the elements to be kept or removed while performing the union operation.

Open the TrimVolume1.CATPart document and make sure Geometrical Set.2 is the current body.

This is your initial data: the Trim part is composed of two shells contained in one geometrical set.

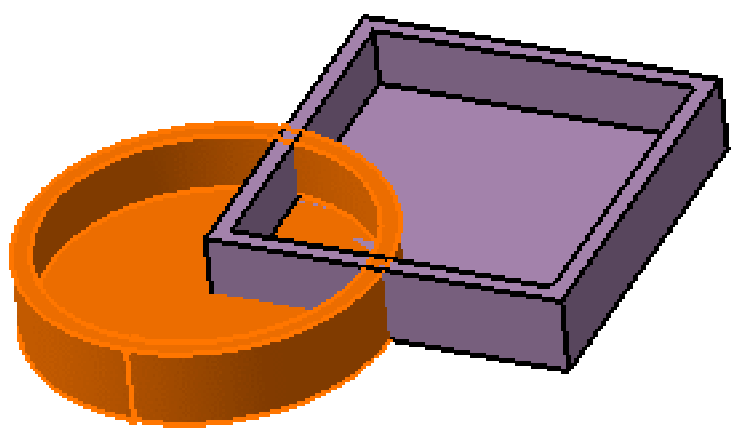




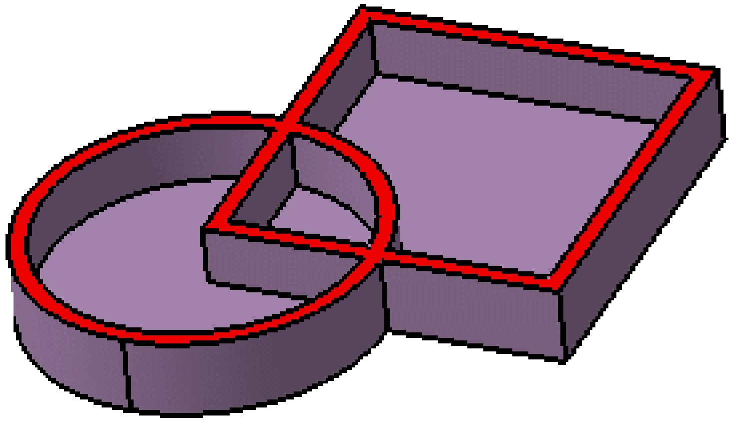
1. Click the **Union Trim** icon from the Volumes operations sub-toolbar.

   The Trim Definition dialog box opens.

2. Select the **Volume to trim**, i.e. Shell.2.

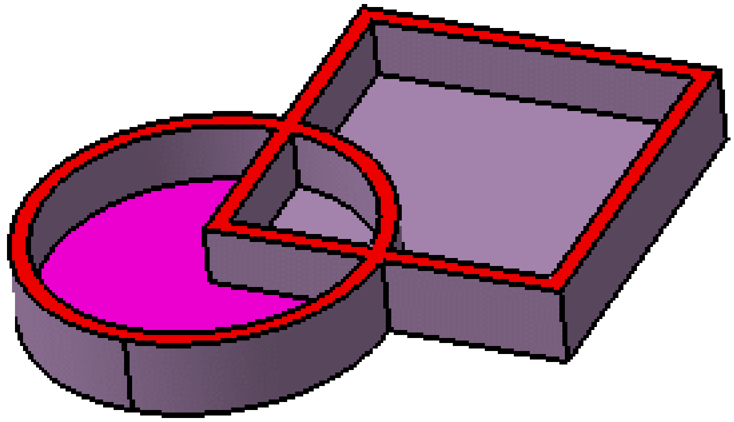


3. Select the **Cutting Volume**, i.e. Shell.1.



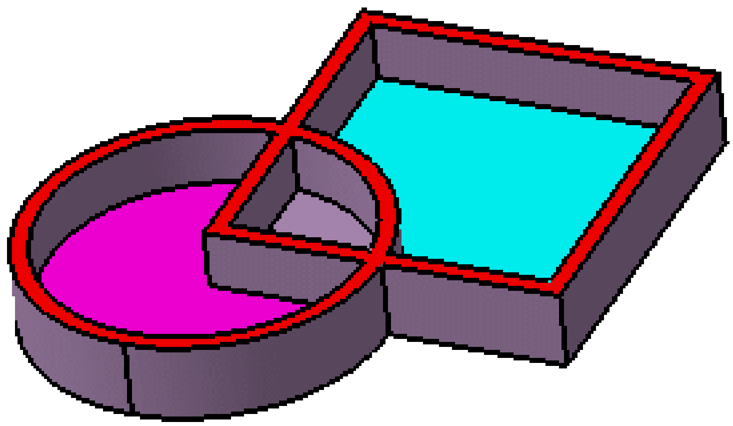
4. Click the **Faces to remove** field and select Shell.2 's inner face.

   The selected face appears in pink, meaning that the application is going to remove it.



5. Click the **Faces to keep** field and select Shell.1 's inner face.

   The selected face appears in light blue, meaning that the application is going to keep it.



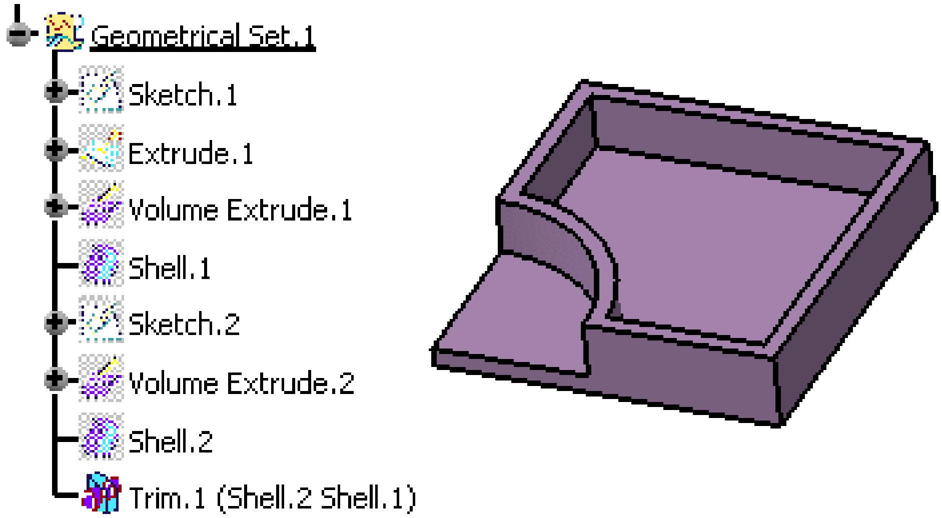
Faces to remove and **Faces to keep** must belong to the same volumes as the selected **Volume to trim** and **Cutting volume**.

Clicking the **Preview** button lets you check if your specifications meet your needs or not.

6. Click OK to compute the material to be removed.

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

   The specification tree and the Add part now look like this:



- Avoid using input elements that are tangent to each other since this may result in geometric instabilities in the tangency zone.
- As much as possible, avoid selecting volumes trimmed by the operation. In some cases, defined trimmed volumes have the same logical name: the application then issues a warning message requiring a better selection.

Generative Shape Design Interoperability

Optimal CATIA PLM Usability for Generative Shape Design

Optimal CATIA PLM Usability for Generative Shape Design

When working with ENOVIA V5, the safe save mode ensures that you only create data in CATIA that can be correctly saved in ENOVIA. Therefore, in interoperability mode, some CATIA V5 commands are grayed out / hidden in the Generative Shape Design workbench.

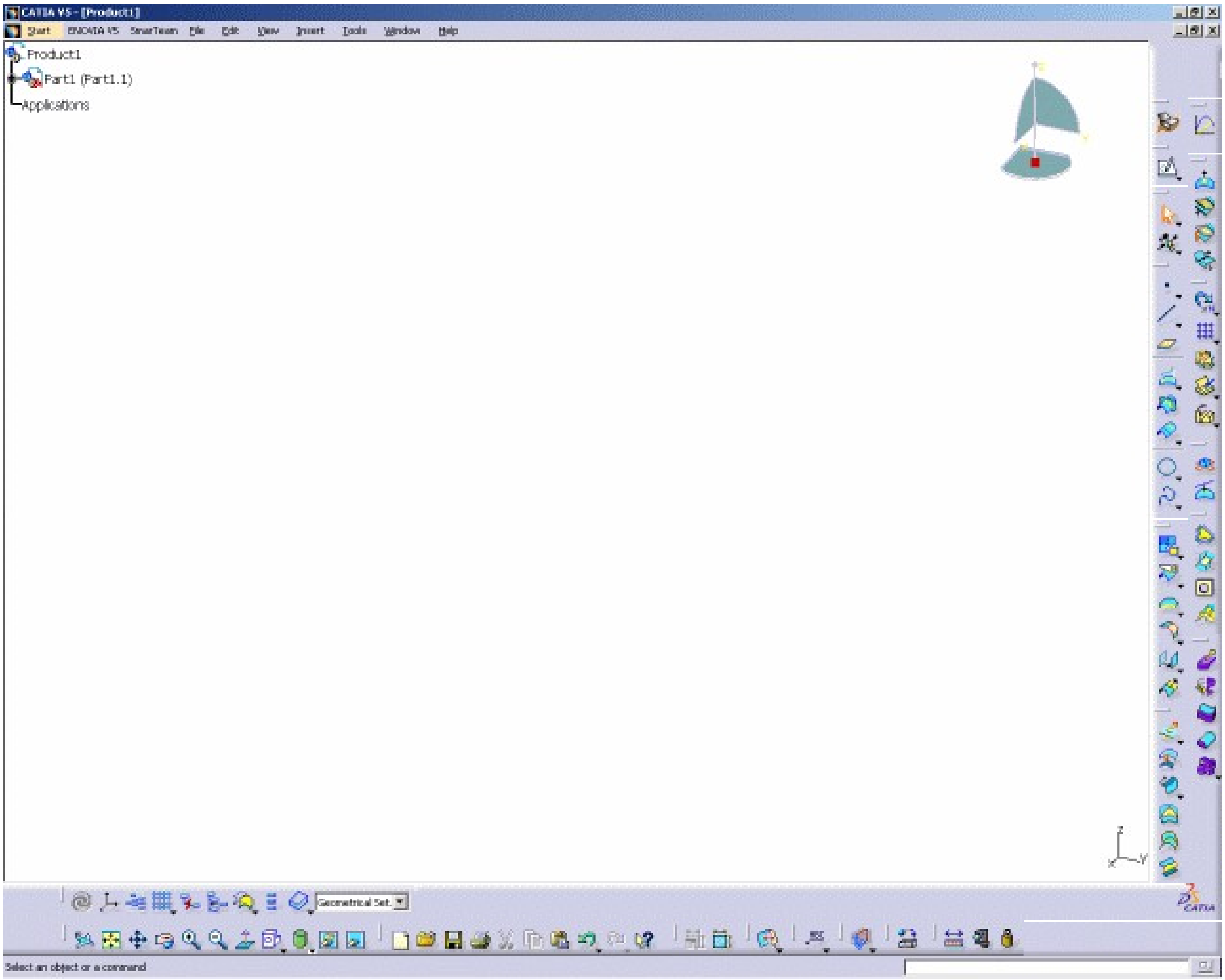
ENOVIA V5 offers two different storage modes: Workpackage (Document kept - Publications Exposed) and Explode (Document not kept).
In Generative Shape Design workbench, when saving data into ENOVIA V5, the global transaction is guaranteed but only if the target is in Workpackage mode. All Generative Shape Design commands are thus available at all times in this mode.

To ensure seamless integration, you must have both a CATIA and ENOVIA session running.

Workbench Description

This section contains the description of the icons, menus and Historical Graph that are specific to the CATIA - Generative Shape Design workbench, which is shown below.

You can click the hotspots on this image to see the related documentation.



Menu Bar
Select Toolbar
Wireframe Toolbar
Surfaces Toolbars
Operations Toolbar
Law Toolbar
Tools Toolbar
Generic Tools Toolbars
ReplicationToolbar
Selection Filter Toolbar
Advanced Surfaces Toolbar
Developed Shapes Toolbar
Volumes Toolbar
BiW Templates Toolbar
Historical Graph
Specification Tree

Generative Shape Design Menu Bar

The various menus and menu commands that are specific to Generative Shape Design are described below.

Start	File	Edit	View	Insert	Tools	Windows	Help

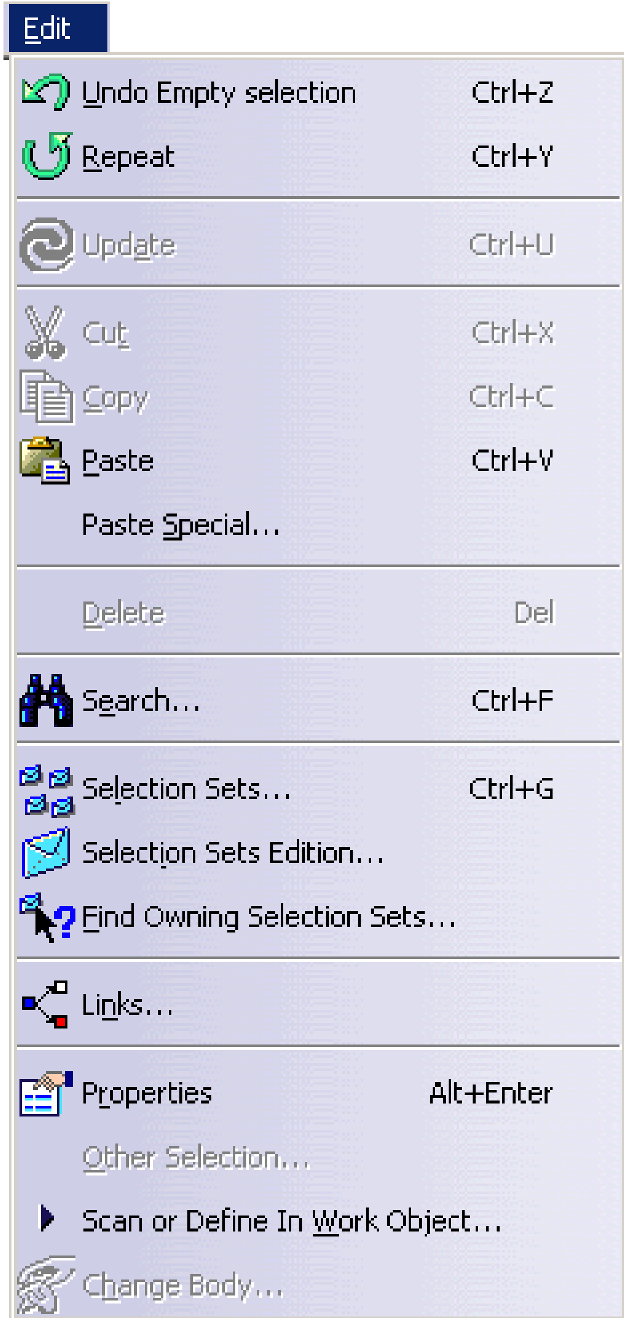
Tasks corresponding to general menu commands are described in the *Infrastructure User's Guide*.

Edit

Please note that most of the edit commands available here are common facilities offered with the Infrastructure.

The specific Generative Shape Design edit commands depend on the type of object being edited: Geometrical Set, Linear Geometrical Set 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	See Deleting Geometry
Search...	Allows searching and selecting objects
Selection Sets... **Selection Sets Edition...** **Find Owning Selection Sets...**	Allows to define and modify selected objects as sets
Links...	Manages links to other documents. See Editing Document Links
Properties	Allows displaying and editing object properties



Other Selection...	See Selecting Using the Other Selections... Command
Scan or Define in Work Object...	See Scanning a Part and Defining In Work Objects
Change Body...	Allows Managing Geometrical Sets

Edit Inputs...	Allows to edit the object inputs and parameters. Refer to the chapter Creating a User Feature in the *Product Knowledge Template User's Guide*.
Activate Deactivate	See Deactivating Elements
Change Body...	Allows Managing Geometrical Sets
AutoSort	Allows to reorder the Geometrical Set's children according to the logical construction order
Reorder Children	See Editing Definitions

Create Group	Allows Managing Groups
Show Components	See Hiding/Showing Geometrical Sets
Hide Components	
Reset Properties	Allows resetting object properties

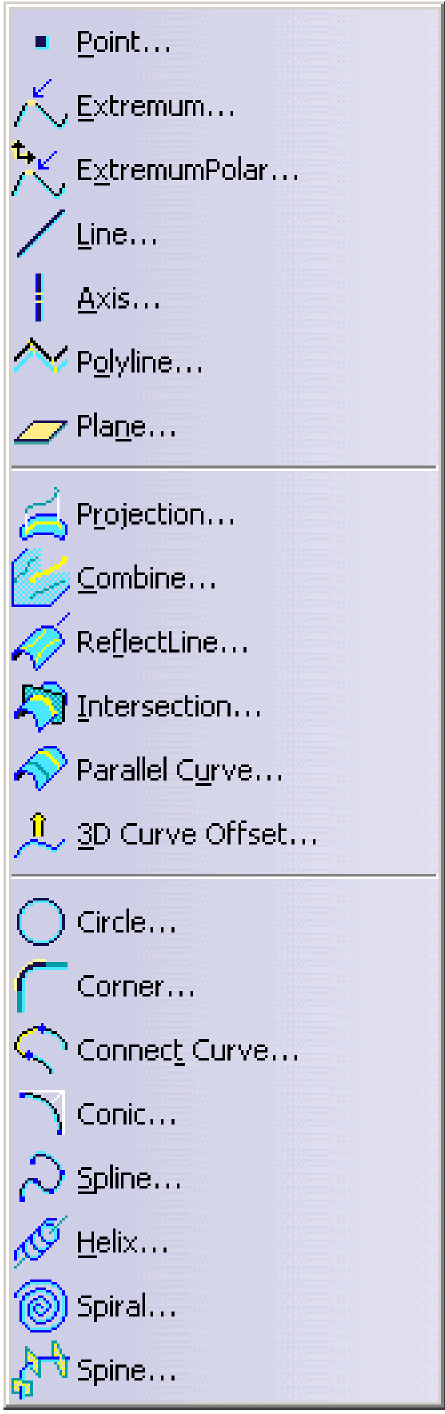
Insert

For...	See...
Body...	Refer to Inserting a New Body in the *Part Design User's Guide*
Body in a Set...	Inserting a Body into an Ordered Geometrical Set
Geometrical set...	Managing Geometrical Sets
Ordered Geometrical Set...	Managing Ordered Geometrical Sets
Sketches	Refer to the *Sketcher User's Guide*
Axis System...	Allows the creation of local axis-system
Wireframe	Insert -> Wireframe
Law	See Creating Laws
Surfaces	Insert -> Surfaces
Volumes	Insert -> Volumes
Operations	Insert -> Operations
Constraints	Insert -> Constraints
Annotations	Insert -> Annotations

Views/Annotation Planes	Insert -> Views/Annotation Planes
Analysis	Insert -> Analysis
Advanced Replication Tools	Insert -> Replication Tools
Knowledge Templates	Insert -> Knowledge Templates
Instantiate From Document...	Instantiating PowerCopies
Instantiate From Selection...	Allows the creation of part templates. Refer to the chapter Creating a Part Template in the *Product Knowledge Template User's Guide*.
Advanced Surfaces	Insert -> Advanced Surfaces
Developed Shapes	Insert-> Developed Shapes
BiW Templates	Insert -> BiW Templates

Insert -> Wireframe

For...	See...
Point...	Creating Points



Extremum...	Creating Extremum Elements
Extremum Polar...	Creating Polar Extremum Elements
Line...	Creating Lines
Axis...	Creating an Axis
Polyline	Creating Polylines
Plane...	Creating Planes
Projection...	Creating Projections
Combine...	Creating Combined Curves
Reflect Line...	Creating Reflect Lines
Intersection...	Creating Intersections
Parallel Curve...	Creating Parallel Curves
3D Curve Offset...	Creating a 3D Curve Offset
Circle...	Creating Circles
Corner...	Creating Corners
Connect Curve...	Creating Connect Curves
Conic...	Creating Conic Curves
Spline...	Creating Splines
Helix...	Creating a Helix
Spiral...	Creating Spirals
Spine...	Creating a Spine

Insert -> Surfaces

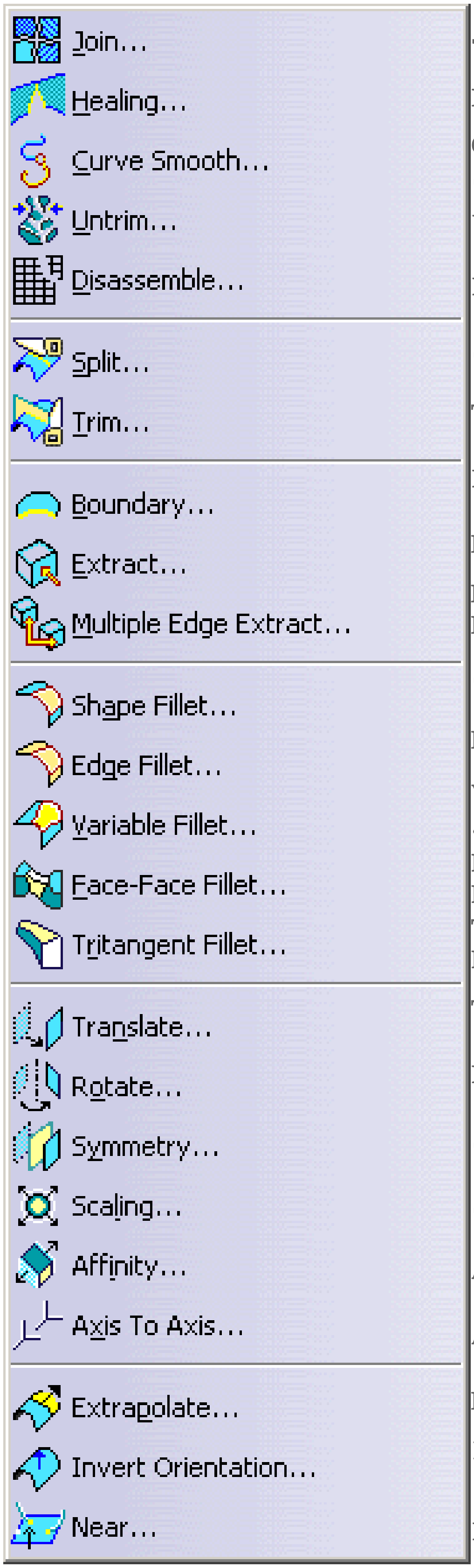
For...	See...
Extrude...	Creating Extruded Surfaces
Revolve...	Creating Revolution Surfaces
Sphere...	Creating Spherical Surfaces
Cylinder...	Creating Cylindrical Surfaces
Offset...	Creating Offset Surfaces

Variable Offset...	Creating Variable Offset Surfaces
Rough Offset...	Creating Rough Offset Surfaces
Sweep...	Creating Swept Surfaces
Adaptive Sweep ...	Creating Adaptive Swept Surfaces
Fill...	Creating Fill Surfaces
Multi-sections Surface...	Creating Multi-Sections Surfaces
Blend...	Creating Blended Surfaces

Insert -> Volumes

For...	See...
Volume Extrude...	Creating Extruded Volumes
Volume Revolve...	Creating Revolution Volumes
Multi-sections Volume...	Creating a Multi-Sections Volume
Volume Sweep...	Creating a Swept Volume
Thick Surface...	Creating a Thick Surface
Close Surface...	Creating a Close Surface
Draft...	Creating a Draft
Draft Variable Angle	Creating a Variable Angle Draft
Draft Reflect Line...	Creating a Draft from Reflect Lines
Shell...	Creating a Shell
Sew Surface...	Creating a Sew Surface
Add...	Adding Volumes
Remove...	Removing Volumes
Intersect...	Intersecting Volumes
Union Trim	Trimming Volumes

Insert -> Operations



For...	See...
Join...	Joining Curves and Surfaces
Healing...	Healing Geometry
Curve Smooth...	Smoothing Curves
Untrim...	Restoring a Surface
Disassemble...	Disassembling Elements
Split...	Splitting Geometry
Trim...	Trimming Geometry
Boundary...	Creating Boundary Curves
Extract...	Extracting Geometry
Multiple Edge Extract...	Extracting Multiple Edges
Shape Fillet...	Shape Fillets
Edge Fillet...	Edge Fillets
Variable Fillet ...	Variable Radius Fillets and Variable Bi-Tangent Circle Radius Fillets Using a Spine
Face-Face Fillet...	Face-Face Fillets
Tritangent Fillet...	Tritangent Fillets
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
Extrapolate...	Extrapolating Geometry
Invert orientation...	Inverting the Orientation of Geometry
Near...	Creating Nearest Entity of a Multiple Element

Insert -> Constraints

For...	See...
Constraint	Creating Constraints
Constraint Defined in a Dialog Box...	

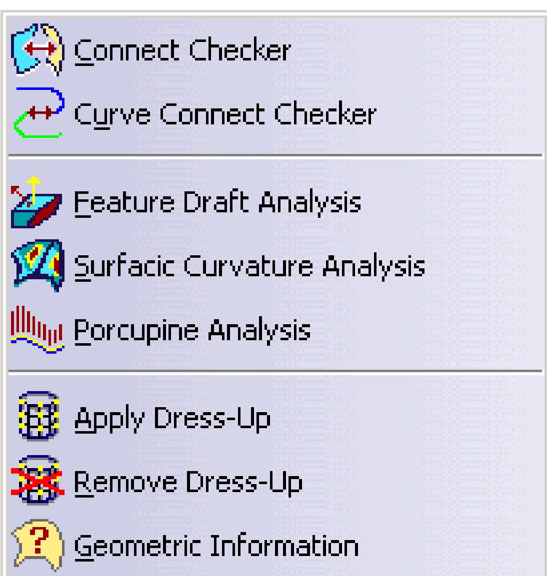
Insert -> Annotations

For...	See...
Text with Leader	Creating a Textual Flag With Leader
Flag Note with Leader	Creating a Flag Note With Leader

Insert -> Views/Annotation Planes

For...	See...
Projection View	Creating a Projection View/Annotation Plane
Section View/Annotation Plane	Creating a Section View/Annotation Plane
Section Cut View/Annotation Plane	Creating a Section Cut View/Annotation Plane

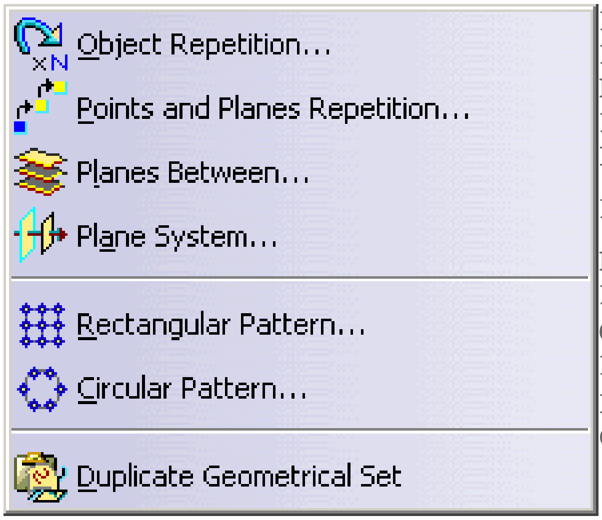
Insert -> Analysis



For...	See...
Connect Checker	Checking Connections Between Surfaces
Curve Connect Checker	Checking Connections Between Curves
Feature Draft Analysis	Performing a Draft Analysis
Surfacic Curvature Analysis	Performing a Surfacic Curvature Analysis
Porcupine Analysis	Performing a Curvature Analysis
Apply Dress-Up	Applying a Dress-Up
Remove Dress-Up	
Geometric Information	Displaying Geometric Information on Elements

Insert -> Advanced Replication Tools

For...	See...
Object Repetition...	Repeating Objects

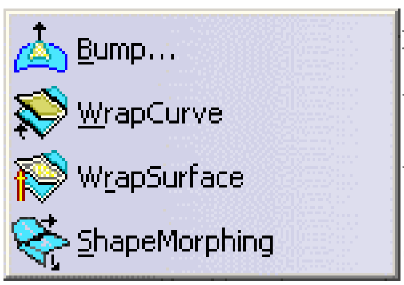


Points and Planes Repetition...	Creating Multiple Points
Planes Between...	Creating Planes Between Other Planes
Plane System...	Creating Plane Systems
Rectangular Pattern...	Creating Rectangular Patterns
Circular Pattern...	Creating Circular Patterns
Duplicate Geometrical Set	Duplicating Geometrical Sets

Insert -> Knowledge Templates

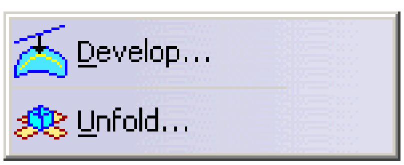
For...	See...
Power Copy...	Creating PowerCopies
UserFeature...	Allows the creation of user features. Refer to the chapter Creating a User Feature in the *Product Knowledge Template User's Guide*.
Document Template...	Allows the creation of part templates. Refer to the chapter Creating a Part Template in the *Product Knowledge Template User's Guide*.
Save in Catalog...	Saving PowerCopies into a Catalog

Insert -> Advanced Surfaces



For...	See...
Bump...	Creating Bumped Surfaces
WrapCurve	Deforming Surfaces Based on Curve Wrapping
WrapSurface	Deforming Surfaces According to Surface Wrapping
ShapeMorphing	Deforming surfaces According to Shape Morphing

Insert -> Developed Shapes



For...	See...
Develop...	Developing Wires and Points
Unfold...	Unfolding a Surface

Insert -> BiW Templates

For...	See...

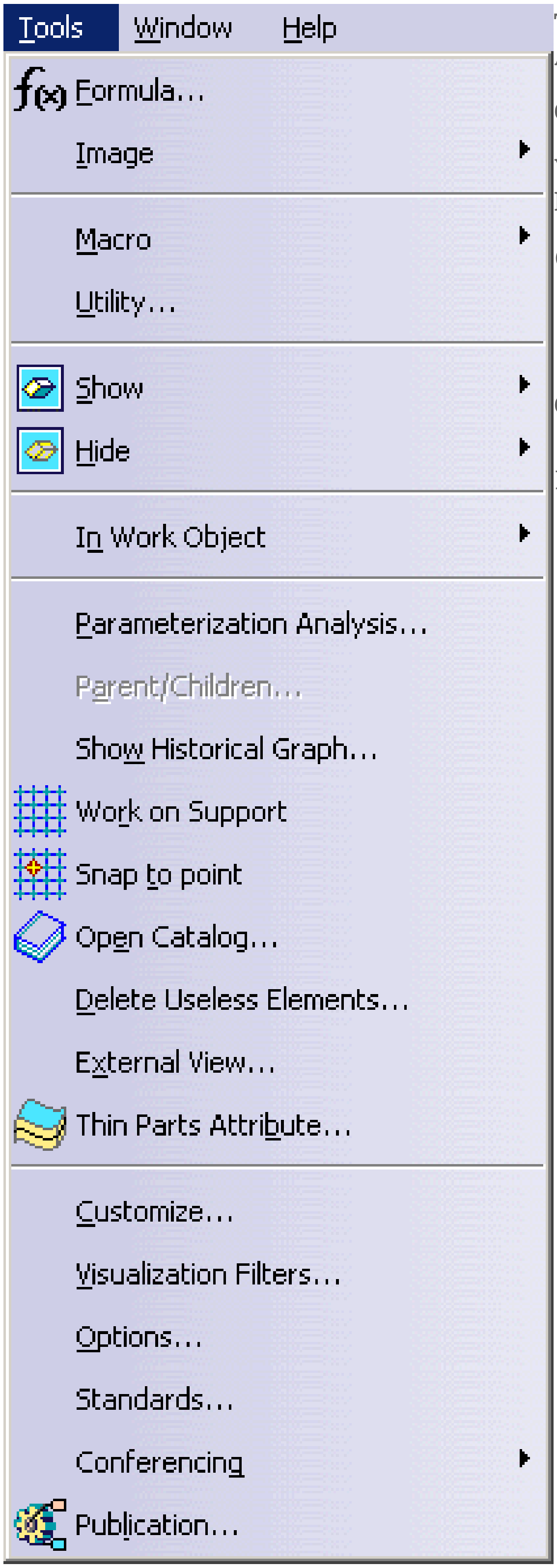
Junction...	Creating Junctions
Diabolo...	Creating a Diabolo
Hole...	Creating a Hole
Mating Flange...	Creating a Mating Flange
Bead...	Creating a Bead

Tools

Please note that most of the Tools commands available here are common facilities offered with the Infrastructure.

Specific Generative Shape Design commands are described in the present document.

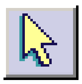
For...	See...
Formula	Allows editing parameters and formula
Image	Allows capturing images
Macro	Allows recording, running and editing macros
Utility	Using the Batch Monitor
Show	Allows to show a set of elements according to their type, or whether they are currently selected or not
Hide	Allows to hide a set of elements according to their type, or whether they are currently selected or not
In Work Object	See Scanning a Part and Defining In Work Objects
Parameterization Analysis...	Analyzing Using Parameterization
Parent/Children...	Allows viewing the parents and children of a selected object
Show Historical graph...	Using the Historical Graph
Work on Support	Working with a Support
Snap to Point	Working with a Support
Open Catalog...	Allows catalog browsing and management
Delete useless elements...	Deleting Geometry
External View...	Allows specifying a feature as a reference for other products/applications

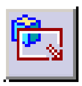


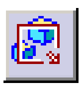
Thin Parts Attribute...	Applying a Thickness
Customize...	Allows customizing the workbench.
Visualization Filters...	Allows layer filters management
Options...	Allows customizing settings
Standards...	See *Managing Standards* in the Interactive Drafting documentation
Conferencing	Allows setting up of communication tools
Publication...	Allows to make documents publicly available.

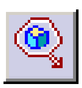
Select Toolbar

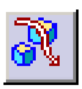
This toolbar contains the following tools to help you selecting and scanning objects.

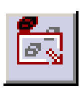


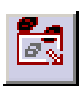




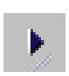
 See Selecting Using Multi-Selection





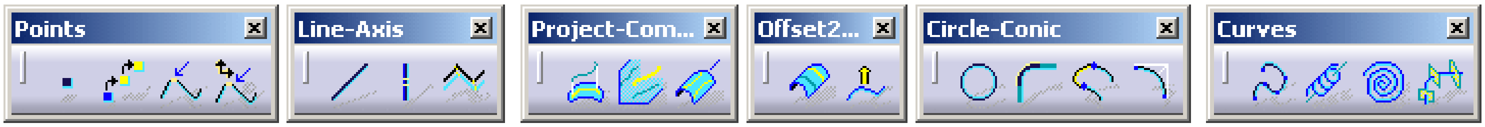


 See Quick Edition of Geometry

 See Scanning the part and Defining In Work Objects

Wireframe Toolbar

This toolbar contains the following tools for creating wireframe elements.



See Creating Points

See Creating Multiple Points

See Creating Extremum Elements

See Creating Polar Extremum Elements

See Creating Lines

See Creating an Axis

See Creating a Polyline

See Creating Planes

See Creating Projections

See Creating Combined Curves

See Creating Reflect Lines

See Creating Intersections

See Creating Parallel Curves

See Create a 3D Curve Offset

See Creating Circles

See Creating Corners

See Creating Connect Curves

See Creating Conic Curves

See Creating Splines

See Creating an Helix

See Creating Spirals

See Creating a Spine

Surfaces Toolbars

This toolbar contains the following tools for creating surfaces.

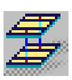
 See Creating Extruded Surfaces

 See Creating Revolution Surfaces

 See Creating Spherical Surfaces

 See Creating Cylindrical Surfaces

 See Creating Offset Surfaces

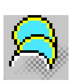
 See Creating Variable Offset Surfaces

 See Rough Rough Offset Surfaces

 See Creating Swept Surfaces

 See Creating Adaptive Swept Surfaces

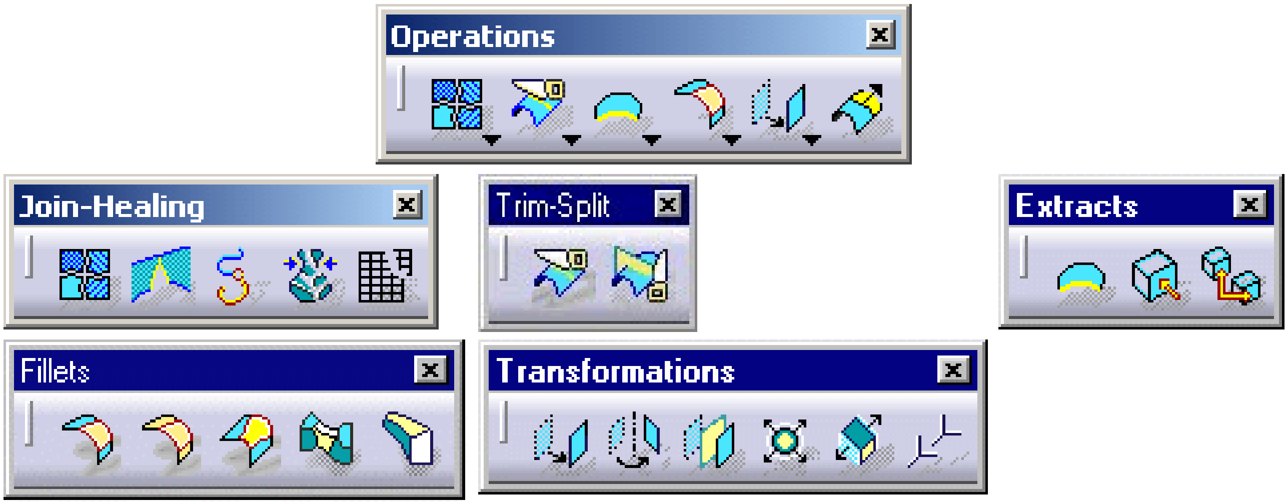
 See Creating Fill Surfaces

 See Creating Multi-section Surfaces

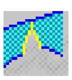
 See Creating Blend Surfaces

Operations Toolbars

These toolbars contain the following tools for performing operations on surface and wireframe elements.

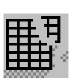


 See Joining Curves and Surfaces

 See Healing Geometry

 See Smoothing Curves

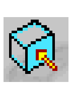
 See Restoring a Surface

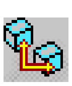
 See Disassembling Elements

 See Splitting Geometry

 See Trimming Geometry

 See Creating Boundary Curves

 See Extracting Geometry

 See Extracting Multiple Edges

See Shape Fillets

See Edge Fillets

See Variable Radius Fillets

See Face-Face Fillets

See Tritangent Fillets

See Translating Geometry

See Rotating Geometry

See Performing a Symmetry on Geometry

See Transforming Geometry by Scaling

See Transforming Geometry by Affinity

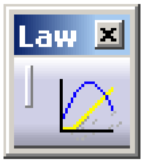
See Transforming Elements from an Axis to Another

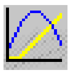
See Extrapolating Surfaces and

See Extrapolating Curves

Law Toolbar

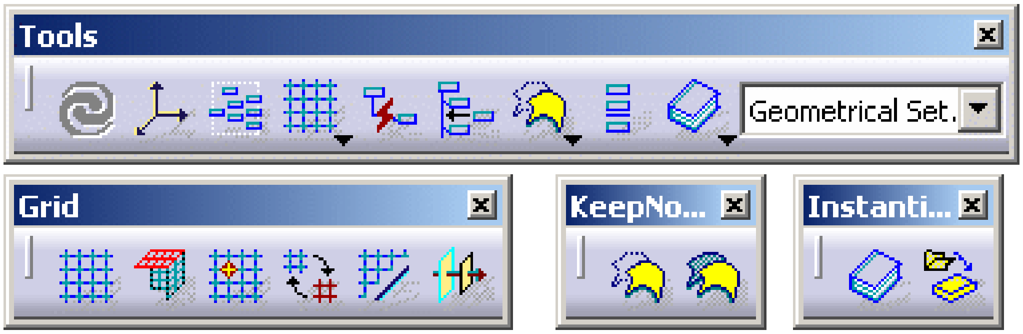
This toolbar contains the following tool for creating laws.

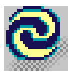


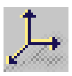
See Creating Laws

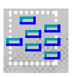
Tools Toolbar

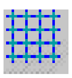
This toolbar contains the following tools to help you model your shape designs.

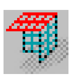


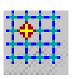
 See Updating Constraints

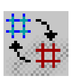
 See Axis system

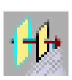
 See Using the Historical Graph

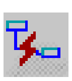
 See Working with a Support

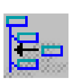
 See Working with a 3D Support

 See Working with a Support

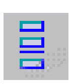
 See Working with a Support

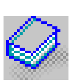
 See Creating Plane Systems

 See Creating Datums

 See Inserting Elements

  See Keeping the Initial Element

 See Displaying the Current Body

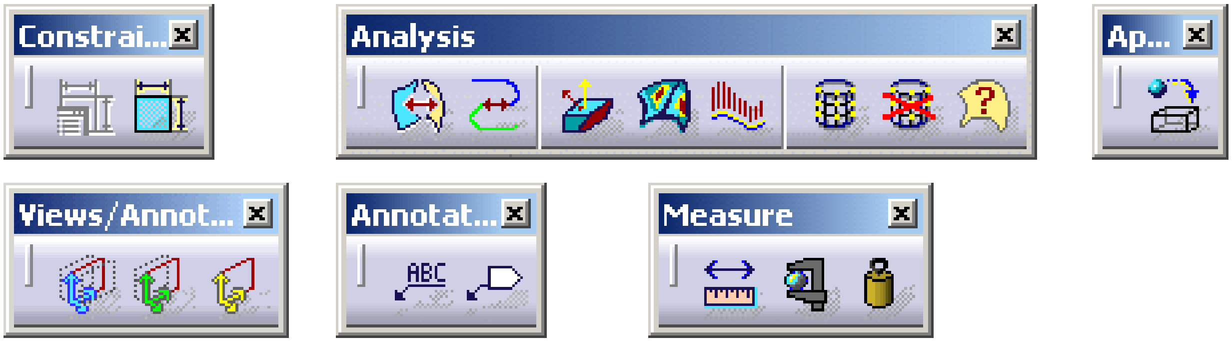
 See Instantiating PowerCopies

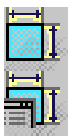
 See Instantiating PowerCopies

See Selecting Bodies

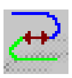
Generic Tools Toolbars

These toolbars contain the following tools to help you manage constraints between geometric elements, perform analyses, and annotate elements in the documents.



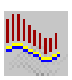
 See Creating Constraints

 See Checking Connections between Surfaces

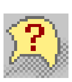
 See Checking Connections between Curves

 See Performing a Draft Analysis

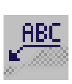
 See Performing a Surfacic Curvature Analysis

 See Performing a Curvature Analysis

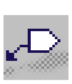
 See Applying a Dress-Up

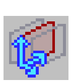
 See Displaying Geometric Information on Elements

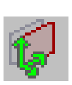
 See Applying a Material Onto Surfaces

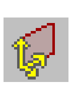


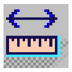
See Creating a Textual Flag With Leader

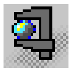
 See Creating a Flag Note With Leader

 See Creating a Projection View

 See Creating a Section View

 See Creating a Section Cut View

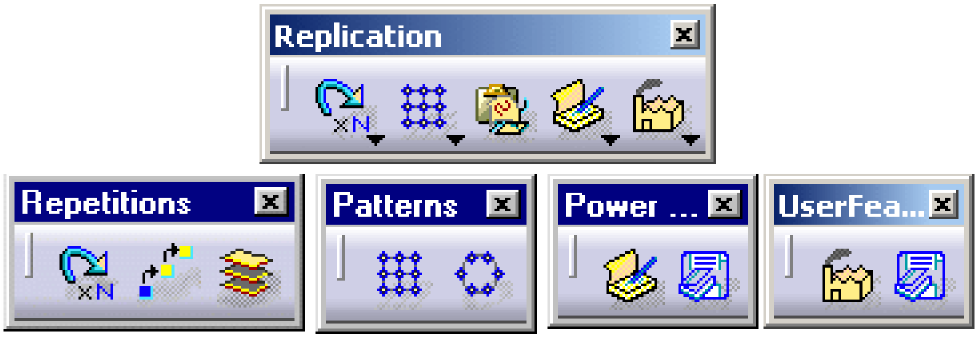
 See Measuring Minimum Distances and Angles

 See Measuring Properties

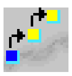
 See Measuring Inertia

Replication Toolbar

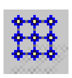
This toolbar contains the tools to performing operations on surface and wireframe elements.

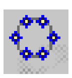


 See Repeating Objects

 See Creating Multiple Points

 See Creating Planes Between Other Planes

 See Creating Rectangular Patterns

 See Creating Circular Patterns

 See Duplicating Geometrical Sets

 See Creating PowerCopies

 See Saving PowerCopies into a Catalog

 See Creating User Features

 See Saving User Features into a Catalog

Selection Filter Toolbar

This toolbar contains the following tools to help you manage sub-geometry selection.

See Selecting using A Filter

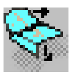
Advanced Surfaces Toolbar

This toolbar, available only with the Generative Shape Optimizer product, contains the tools to create complex surfaces.

 See Creating Bumped Surfaces

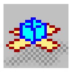
 See Deforming Surfaces Based on Curve Wrapping

 See Deforming Surfaces According to Surface Wrapping

 See Deforming Surfaces According to Shape Morphing

Developed Shapes Toolbar

This toolbar, available only with the Developed Shapes, contains the tools to create developed surfaces.

 See Unfolding a Surface

 See Developing Wires

Volumes Toolbar

This toolbar contain the following tools to help you create volumetric features.

 See Creating Extruded Volumes

 See Creating Revolution Volumes

 See Creating Multi-Sections Volumes

 See Creating Swept Volumes

 See Creating a Thick Surface

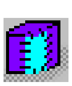
 See Creating a Close Surface

 See Creating a Draft

 See Creating a Variable Angle Draft

 See Creating a Draft From Reflect Lines

 See Creating a Shell

 See Creating a Sew Surface

 See Adding Volumes

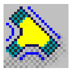
See Removing Volumes

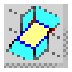
See Intersecting Volumes

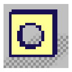
See Trimming Volumes

BiW Templates Toolbar

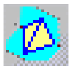
This toolbar, available only with the BiW Templates product, contains the tools to create BiW design.

 See Creating Junctions

 See Creating a Diabolo

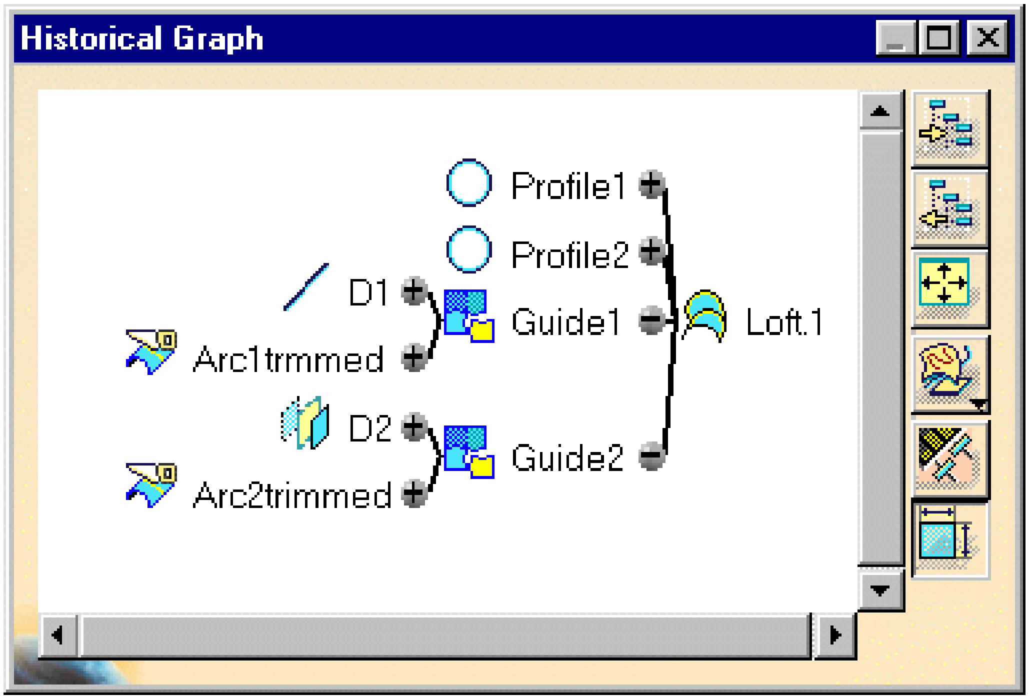
 See Creating a Hole

 See Creating a Mating Flange

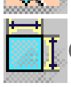
 See Creating a Bead

CATIA - Generative Shape Design Historical Graph

In this chapter we will describe the Historical Graph's commands and contextual commands that are specific to the CATIA - Generative Shape Design workbench.



Historical Graph Commands

	Command...	Description...
	Add Graph	Adds a selected element to the graph.
	Remove Graph	Removes a selected element from the graph.
	Reframe	Centers the graph in the window.
	Surface or Part graph representation	Gives a horizontal or vertical representation.
	Parameters	Displays any parameters associated with the elements in the graph.
	Constraints	Displays any constraints associated with the elements in the graph.

Historical Graph Contextual Commands

Command...	Description...
Reframe	Centers the graph in the window.
Print	Allows you to obtain a print of the graph.
Graph All	Restores the graph to the window.
Clean Graph	Clears the graph from the window.
Refresh	Refreshes the graph display.

Generative Shape Design Specification Tree

Within the Generative Shape Design 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
- Geometrical Set
- Ordered Geometrical Set
- Multi-Output
- Point
- Multiple Points
- Extremum
- Extremum Polar
- Line
- Axis
- Polyline
- Plane
- Multiple Planes
- Circle
- Conic
- Spiral
- Spline
- Helix
- Spine
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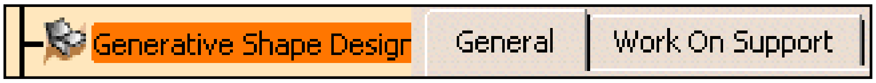
Generative Shape Design



This page deals with the Generative Shape Design options:

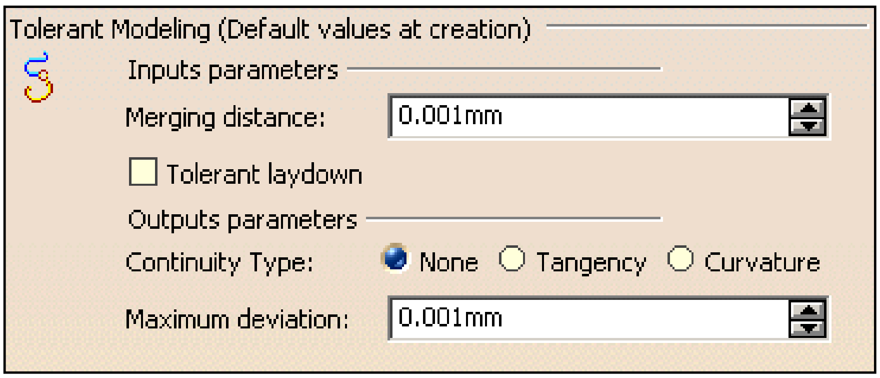
- The General tab lets you define the tolerant modeling, axes visualization and groups options.
- The Work On Support tab lets you define the work on support and the work on support 3D options.

General Settings

This page deals with the following settings:

- Tolerant Modeling
- Axes Visualization
- Groups
- Stacked Analysis

Tolerant Modeling



Input parameters

- **Merging distance**: default value defining the distance below which elements are to be joined or healed.

This option is available with the following commands: Join and Healing.

By default, this option is set to 0.001mm.

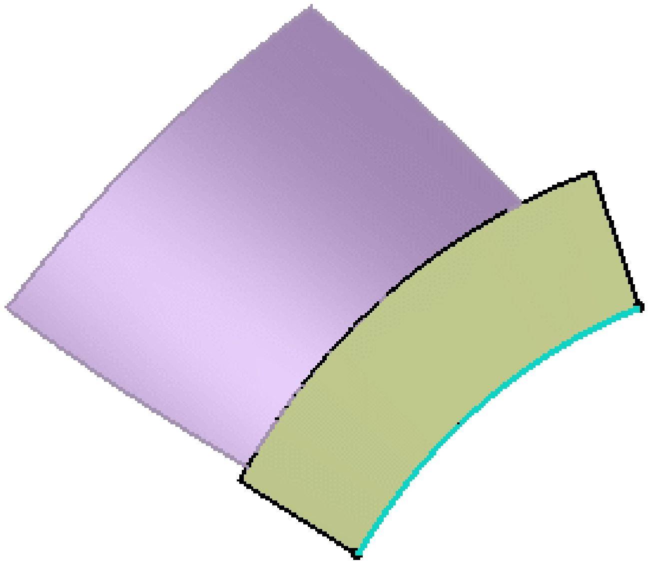
- **Tolerant laydown:** in case the lay down of an input guide on a surface fails, you can check the **Tolerant laydown** button. When activated, a fixed lay down tolerance of 0.1mm is applied.

This option is available with the following commands: Parallel Curve, Sweep, Multi-Sections Surface, Blend, Split, Curve Smooth,

Fill and Extrapol.

By default, this option is unchecked.

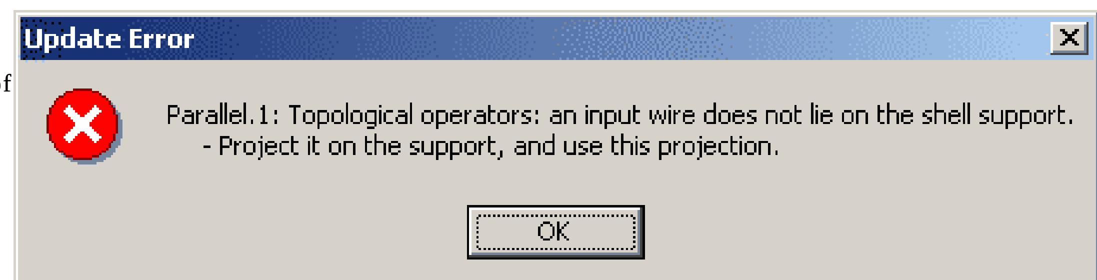
Here is a scenario with the Sweep command.



1. Create a tolerant swept surface and define a **Deviation from guide** of 0.1 mm.

   In our scenario we created a swept surface using an implicit linear profile and the "two limits" sub-type. We created Curve.1 and Curve.2 as the guide curves.

2. Create a parallel curve of Curve.1 using the swept surface as the support.

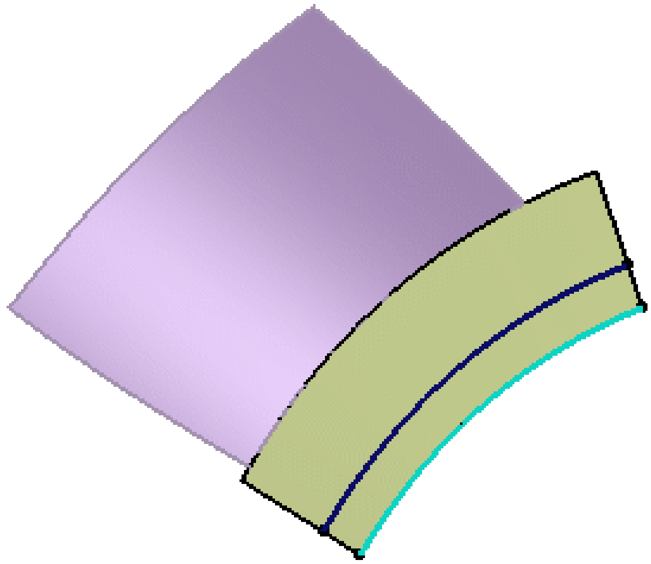


   The creation of the parallel curve fails.

   An error message opens informing you that the guide curve does not lie on the swept surface.

3. Check the **Tolerant laydown** option.
4. Perform step 2 again.

   The creation of the parallel curve is successful.



It is advised not to use a wire that lies on the edge of the sweep when working with the **Tolerant laydown** option.

Output parameters

- Choose the Continuity Type:
  - **None**: deactivates the smoothing result
  - **Tangency**: enhances the current continuity to tangent continuity
  - **Curvature**: enhances the current continuity to curvature continuity

This option is available with the following commands: Project and Parallel Curve.

By default, the option is None.

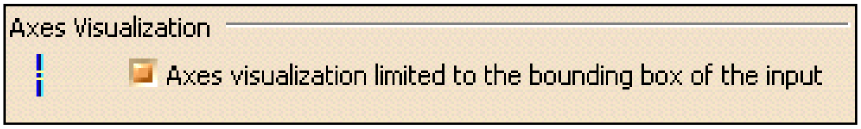
- You can specify a **Maximum deviation** to set the allowed deviation between the initial element and the smoothed element by entering a value or using the spinners.

By default, this option is set to 0.001mm.

This option is available with the following commands: Project, Parallel Curve, Sweep, Multi-Sections Surface, and Curve Smooth.

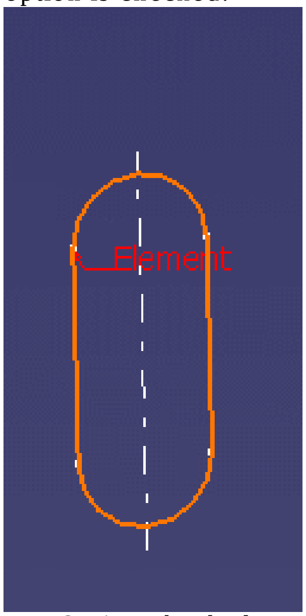
For the Sweep and Multi-Sections Surface commands, only the **Deviation** parameter can be defined from **Tools -> Options** (the **Angular correction** parameter cannot be defined here). The **Deviation** and **Angular correction** will be activated only if the smoothing type is set to **Tangency** or **Curvature**.

Axes Visualization

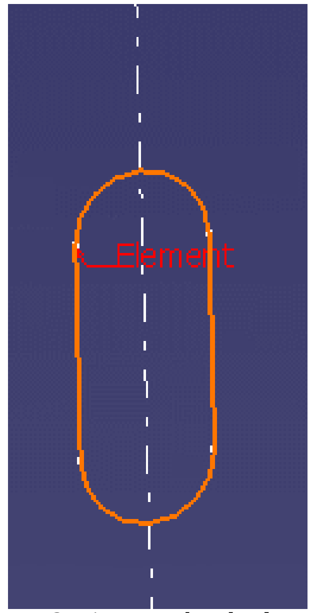


Uncheck the **Axes visualization limited to the bounding box of the input** button to visualize an infinite axis in the 3D geometry.

By default this option is checked.



Option checked



Option unchecked

Groups

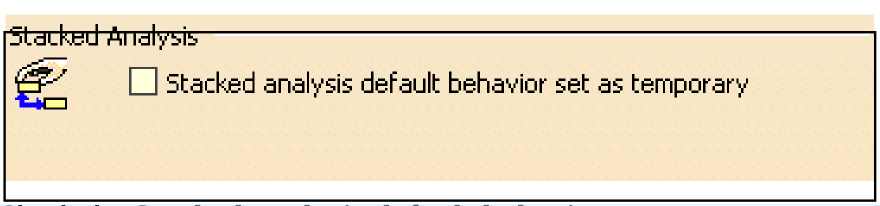
Check the **Integration of created features as group inputs** option if you want each new feature to be included as an input in an existing group and remain visible in the specification tree.
If you uncheck this option, created features will not included in the group and will be hidden in the group tree (expand the group to be able to see it).

By default this option is unchecked.

- This option is only available when creating a new feature.
- It is only available for features accessible in the Generative Shape Design workbench. All other features will not be included in the group even if the option is checked.

For further information, please refer to the *Managing Groups* chapter.

Stacked Analysis

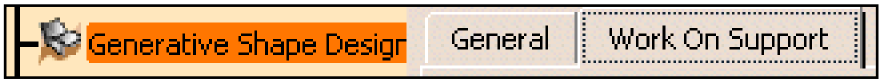


Check the **Stacked analysis default behavior set as temporary** option to automatically create a temporary analysis when checking connections between surfaces or curves.

By default this option is unchecked.

This option is only available with the Offset command.

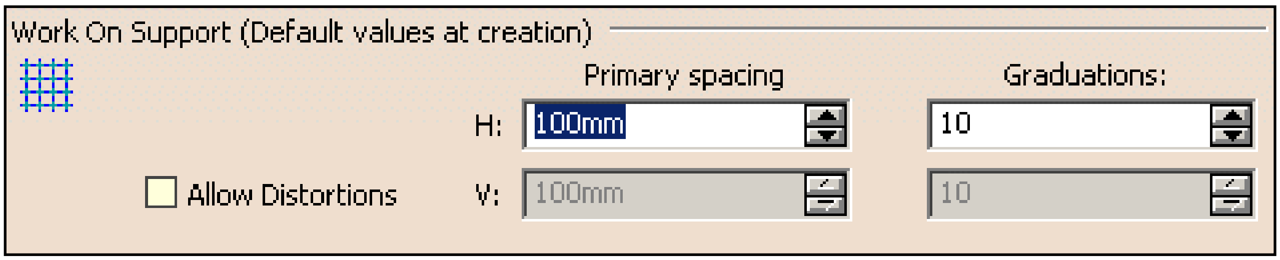
Working with a Support



This page deals with the following settings:

- Work On Support
- Work On Support 3D

Work On Support



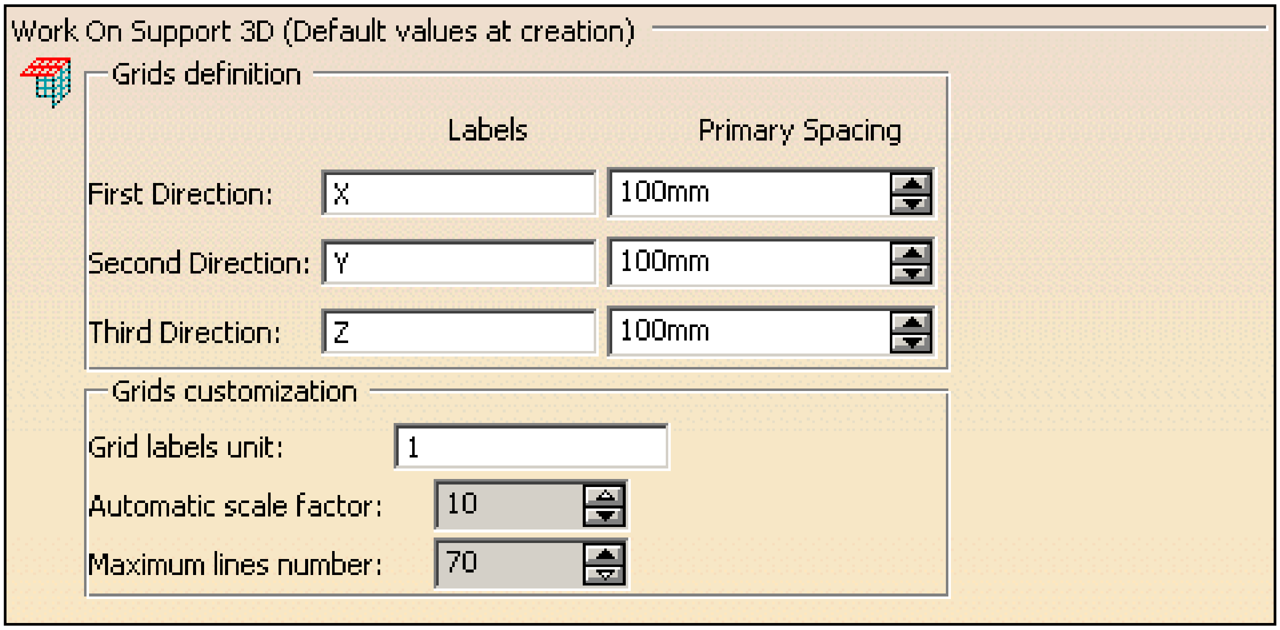
- Define the First direction scale (H for horizontal), by setting **Primary spacing** and **Graduations** values.

By default, this option is set to 100mm for primary spacing and 10 graduations.

- 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.

By default, this option is unchecked.

Work On Support 3D



Grids definition

Define the default values for the grids, by defining **Labels** names and **Primary Spacing** values for each direction.

By default, the Labels are set to X, Y, and Z for the first, second and third directions, and the Primary Spacing is set to 100mm for all directions.

Grids customization

- Define the **Grid labels unit**, that is the coefficient of division for all straight lines units.

  For example, the grid label unit is set to 1 and the X label for one line is set to 500. If you modify the grid label unit to 10, the X label will be 50.

By default, this option is set to 1.

- Define the **Automatic Scale Factor**, that is the scale to be used when the grid becomes too small. The factor is comprised between 2 to 10.

By default, this option is set to 10.

- Define the **Maximum lines number** to be displayed on the screen, from 70 up to 500 lines.

By default, this option is set to 70.

All these values will be used as the default values when creating a support.

For further information, please refer to the *Working With a Support* and *Working With a 3D Support* chapters.

Glossary

A C E F G J M 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

extrapolate	An operation in which an element is extended a specified amount while respecting tangency or curvature conditions. Typically a surface boundary can be selected for in order to extrapolate the surface a specified length.
extruded surface	A surface that is obtained by extruding a profile along a specified direction.

F

feature	A component of a part.
fill surface	A surface that is obtained by filling a closed boundary that is made up from a number of segments.
fillet	A curved surface of a constant or variable radius that is tangent to and joins two surfaces. Together these three surfaces form either an inner or outer corner.

G

guiding curve	A curve, intersecting with a profile, and along which this profile is swept. See also spine.

J

join	An operation in which adjacent curves or adjacent curves can be assembled to make up one element.

M

multi-section 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.

spine A curve which normal planes are used to position a profile when creating a surface (lofted or swept surface for example). The profile does not necessarily intersect with this spine. See also guiding curve.

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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