

Generative Shape Design

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Preface

CATIA Version 5 Generative Shape Design 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 CATIA.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, CATIA.Generative Shape Design can be used with other CATIA Version 5 products such as CATIA.Part Design and CATIA.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.

The *CATIA Generative Shape Design 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.



[Using This Book](#)



[More Information](#)

Using This Book

This book is intended to help you become quickly familiar with CATIA Generative Shape Design. You should already be accustomed with basic CATIA Version 5 concepts such as document windows, standard and view toolbars.

To get the most out of this guide, we suggest you start reading and performing the step-by-step [Getting Started](#) tutorial.

This tutorial shows you how to build a shape design from a basic wireframe model.

The next sections present the main capabilities in the form of basic and advanced user's tasks. It may be a good idea to take a look at the section describing the workbench menus and toolbars.



Where to Find More Information

Prior to reading this book, we recommend that you read the *CATIA.Version 5 Infrastructure User's Guide*.

The *CATIA.Part Design User's Guide* and *CATIA.FreeStyle Shaper and Optimizer User's Guide* may also prove useful.



What's New?

Editing Geometry:

Enhanced [extrapolating surfaces](#) or [curves](#)

New shape geometry capability: [tritangent fillets](#) and [face-face fillets](#)

Modified [splitting](#) and [trimming](#) capabilities

Creating Surfaces:

Enhanced [lofted surfaces creation](#)

Modified [sweep](#) creation capability

Creating Wireframe Geometry:

Enhanced [spline creation](#) capabilities

Modified [projection creation](#) capability

Using Tools

New analysis capabilities: [mapping](#), [draft](#) and [connection checking](#)

Getting Started

Before getting into the detailed instructions for using CATIA 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:

Entering the Workbench	
Lofting and Offsetting	
Splitting, Lofting and Filleting	
Sweeping and Filleting	
Using the Historical Graph	
Transforming the Part	



This tutorial should take about 20 minutes to complete.

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 *CATIA Version 5 Infrastructure User's Guide*.



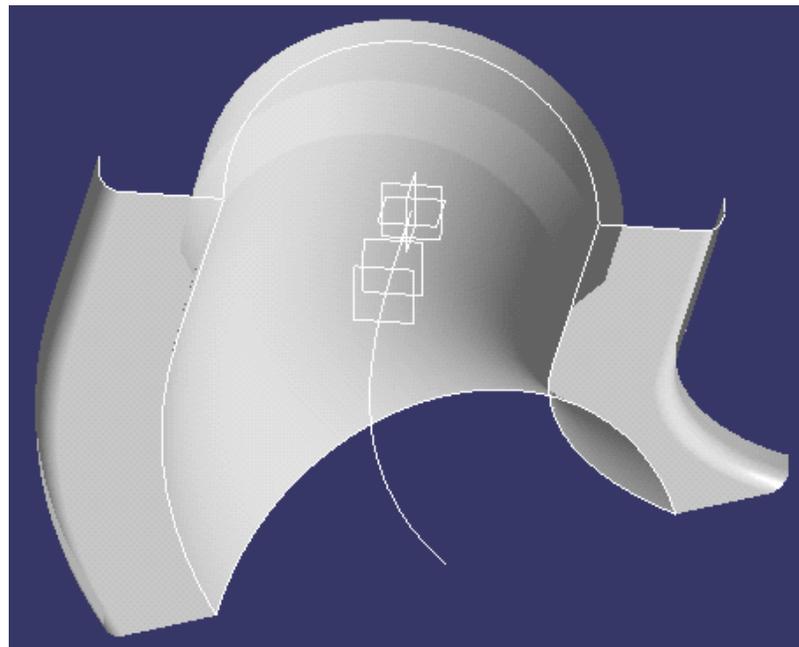
1. Select Shape -> Generative Shape Design from the Start menu.

The Shape Design workbench is displayed.

2. Select File -> Open then select the GettingStartedShapeDesign.CATPart document from the samples/ShapeDesign directory.

A wireframe design part is displayed.

In the rest of this scenario, you will use the construction elements of this part to build up the following shape design.





Up

Splitting, Lofting and Filleting



Entering the Workbench

Sweeping and Filleting



Transforming the Part



Lofting and Offsetting

Using the Historical Graph



Lofting and Offsetting



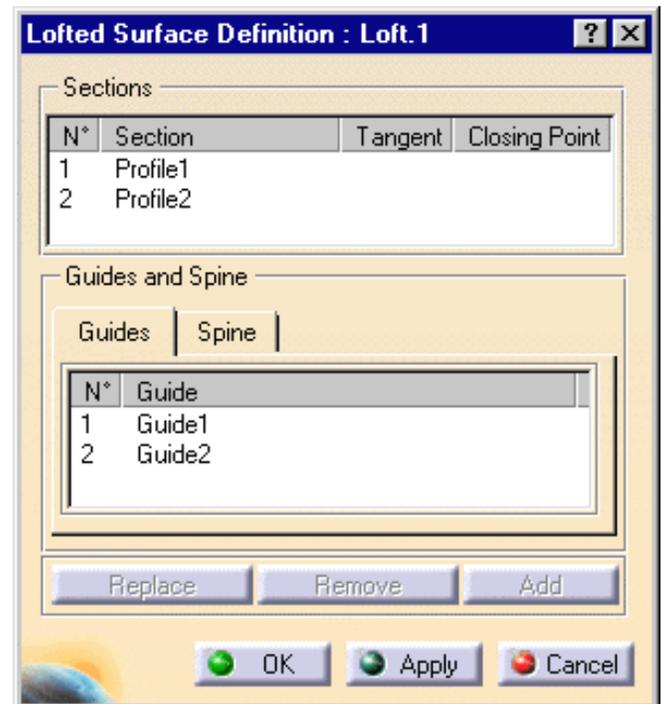
This task shows you how to create a lofted surface and an offset surface.



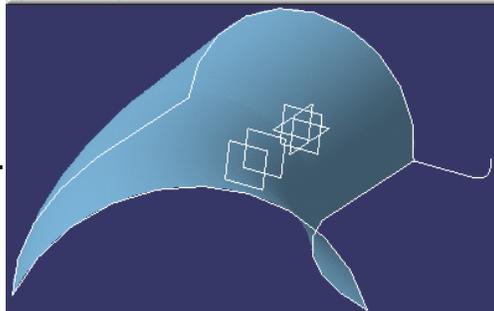
1. Click the Loft icon .

The lofted Surface Definition dialog box appears.

2. Select the two section curves.
3. Select the two guide curves.



4. Click OK to create the lofted surface.



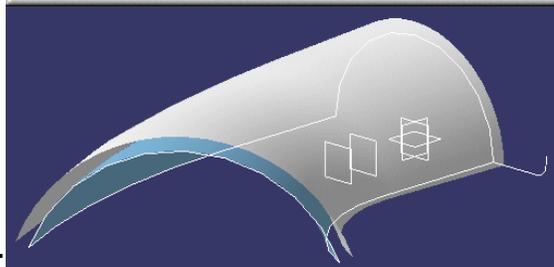
5. Click the Offset icon .



6. Select the lofted surface.
7. Enter an offset value of 2mm.

The offset surface is displayed normal to the lofted surface.

8. Click OK to create the offset surface.





 Up

 Entering the Workbench

 Lofting and Offsetting

 Splitting, Lofting and Filleting

 Sweeping and Filleting

 Using the Historical Graph

 Transforming the Part

Splitting, Lofting and Filleting



This task shows how to split surfaces then create a lofted 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 as cutting element.

4. Click OK to split the surface.

5. Repeat the previous operation by selecting the lofted surface then the second plane as cutting element.

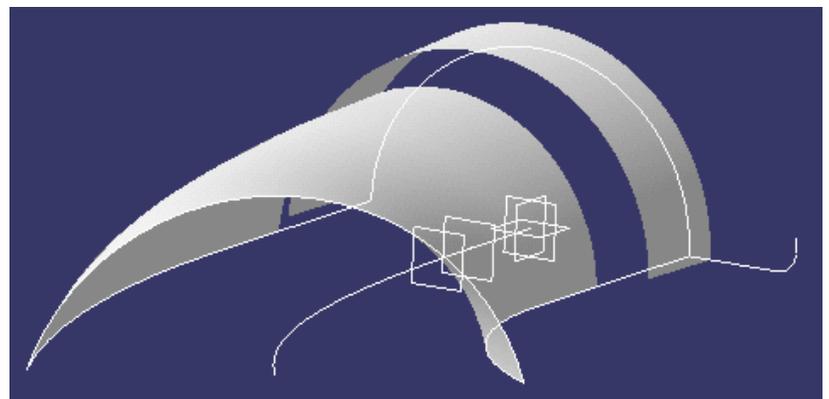
6. Click OK to split the surface.

7. Click the Loft icon .

The Lofted Surface Definition dialog box appears.

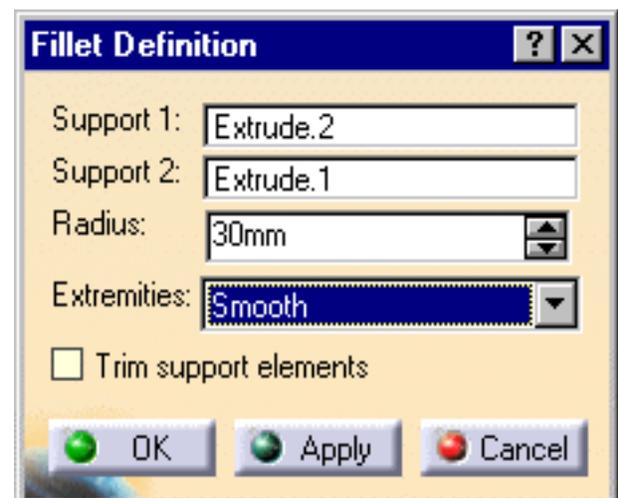
8. Select the edges of the two split surfaces as sections.

9. Click OK to create the lofted surface between the two split surfaces.



10. Click the Shape Fillet icon .

The Fillet Definition dialog box appears.



11. Select the first split surface as the first support element.
12. Select the lofted surface as the second support element.
13. Enter a fillet radius of 3mm.

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

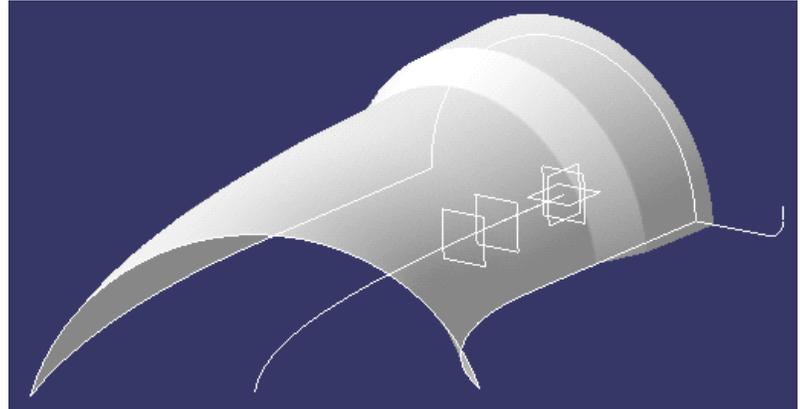
14. Make sure that the surface orientations are correct, then click OK to create the first fillet surface.

15. Select the second split surface as the first support element.

16. Select the lofted surface as the second support element.

17. Enter a fillet radius of 3mm.

18. Make sure that the surface orientations are correct, then click OK to create the second fillet surface.



Sweeping and Filletting



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

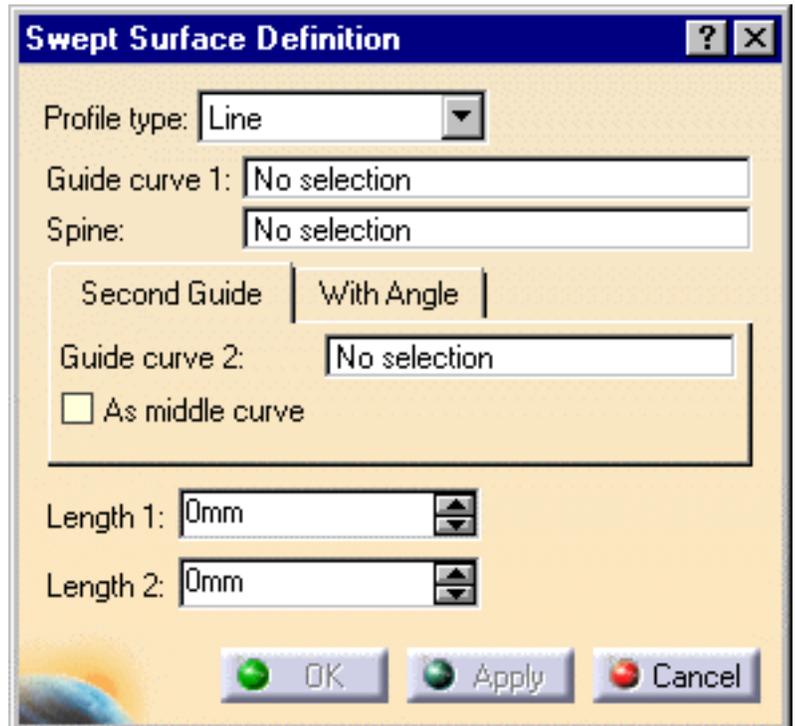
You will use the profile element on the side of the part for this. In this task you will also create a symmetrical profile element on the opposite side of the part.



1. Click the Sweep icon .

The Swept Surface Definition dialog box appears.

2. Use the combo to choose the Explicit profile type.

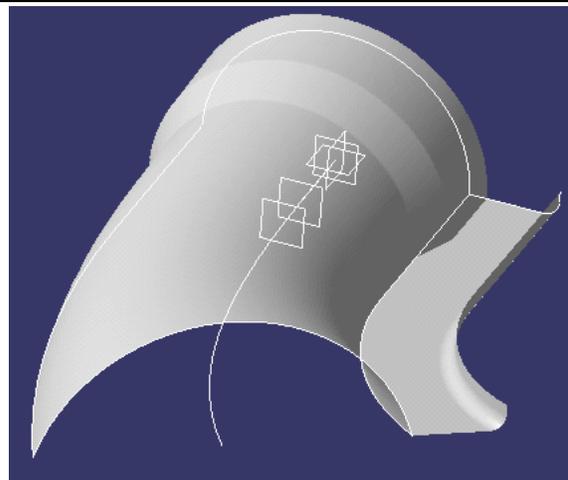


3. Select the guide curve.

4. Select the profile element.

5. Select the central curve as the spine.

6. Click OK to create the swept surface.



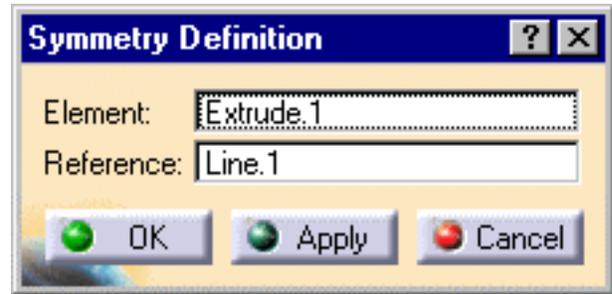
7. Click the Symmetry icon .

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.

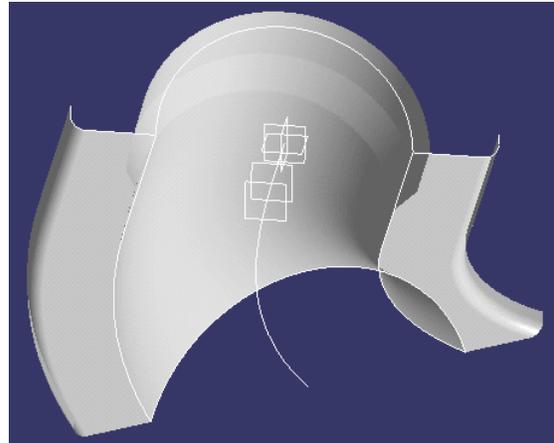


11. Click the Sweep icon  again.

12. Select the guide curve and the profile element.

13. Select the central curve as the spine.

14. Click OK to create the swept surface.



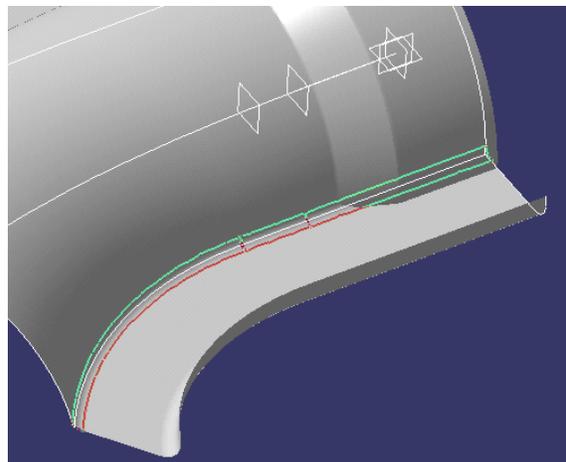
15. To create a fillet between the side portion and the central part

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.

17. Click Apply to preview the fillet.

18. Select the other sweep element and the central portion of the part then enter a fillet radius of 1mm.



19. Click OK to create the fillet.



Up

Splitting, Lofting and Filleting



Entering the Workbench



Sweeping and Filleting



Transforming the Part



Lofting and Offsetting



Using the Historical Graph



Using the Historical Graph



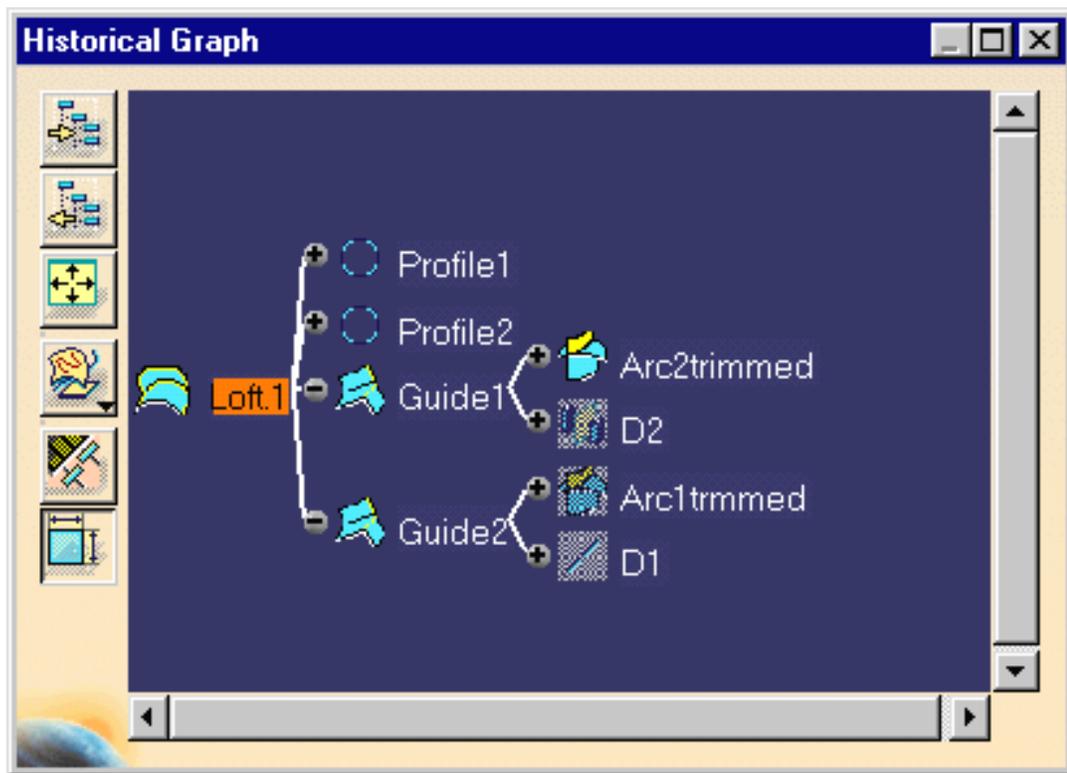
This task shows how to use the historical graph.



1. Select the element for which you want to display the historical graph.

2. Click the Show 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 Loft.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.

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

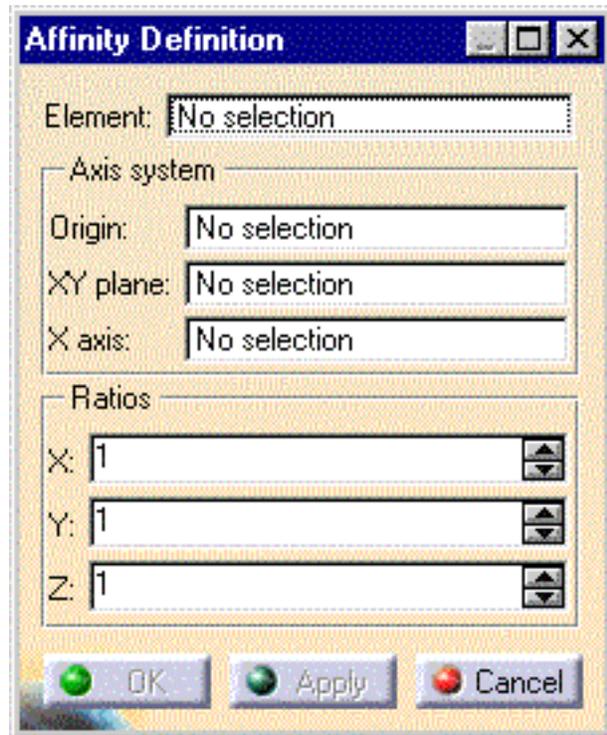


 Up	 Entering the Workbench	 Lofting and Offsetting
 Splitting, Lofting and Filleting	 Sweeping and Filleting	 Using the Historical Graph
	 Transforming the Part	

Transforming the Part



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



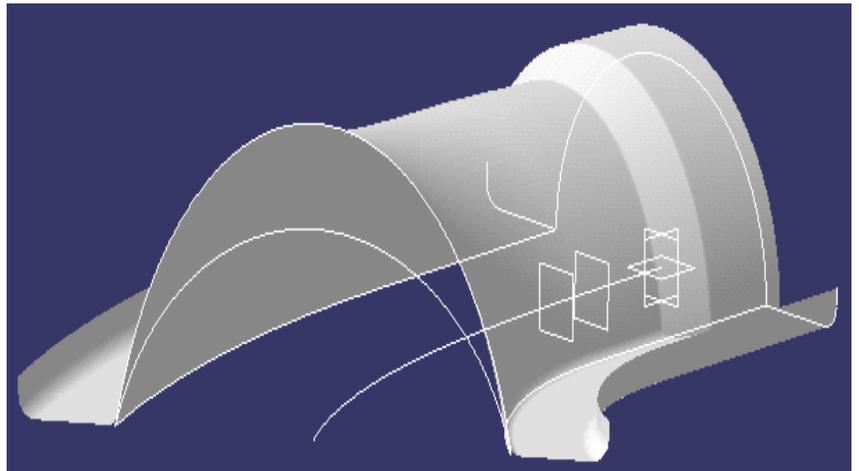
1. Click the Affinity icon .

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 to replace the section profile by the new profile.

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



 Up

 Entering the Workbench

 Lofting and Offsetting

 Splitting, Lofting and Filleting

 Sweeping and Filleting

 Using the Historical Graph

 Transforming the Part

Basic Tasks

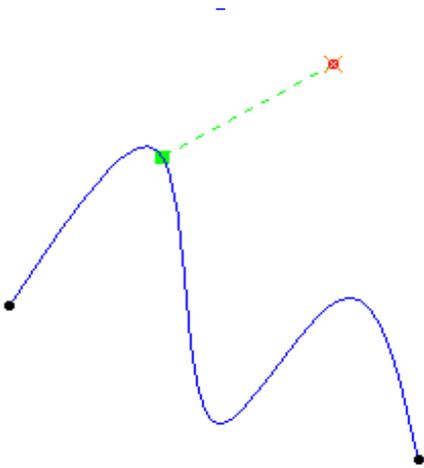
The basic tasks you will perform in the CATIA - Generative Shape Design workbench will involve creating and modifying wireframe and surface geometry that you will use in your part.

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

Theme	
Creating Wireframe Geometry	
Creating Surfaces	
Performing Operations	
Editing Geometry	
Using Tools	

Creating Wireframe Geometry

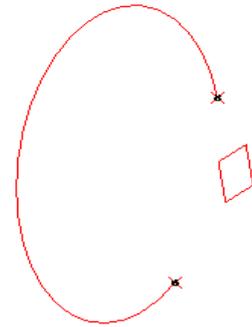
CATIA - Generative Shape Design allows you to create wireframe geometry such as points, lines, planes and curves. You can make use of this elementary geometry when you create more complex surfaces later on.



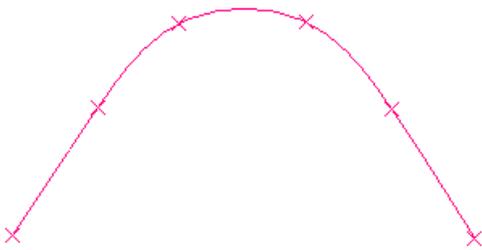
Creating Points



Creating Lines



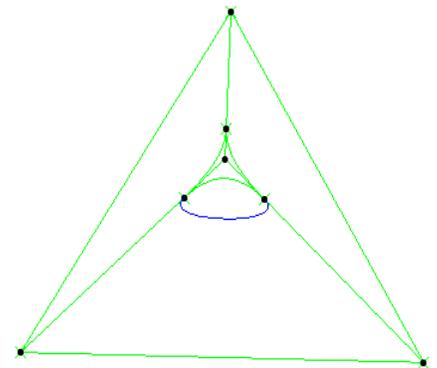
Creating Circles



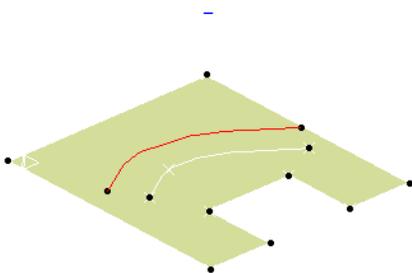
Creating Splines



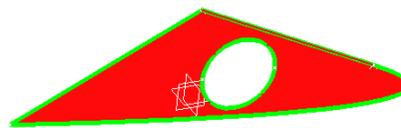
Creating Corners



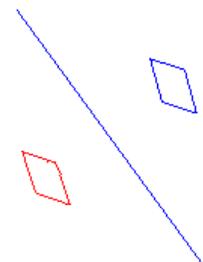
Creating Connect Curves



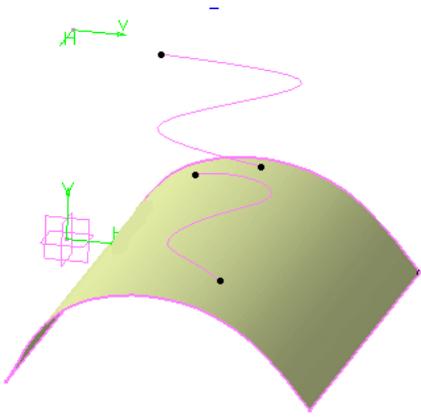
Creating Parallel Curves



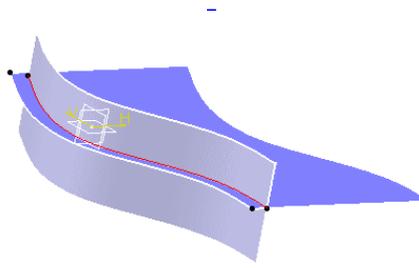
Creating Boundary Curves



Creating Planes



Creating Projections



Creating Intersections



Creating Points



This task shows the various methods for creating points:

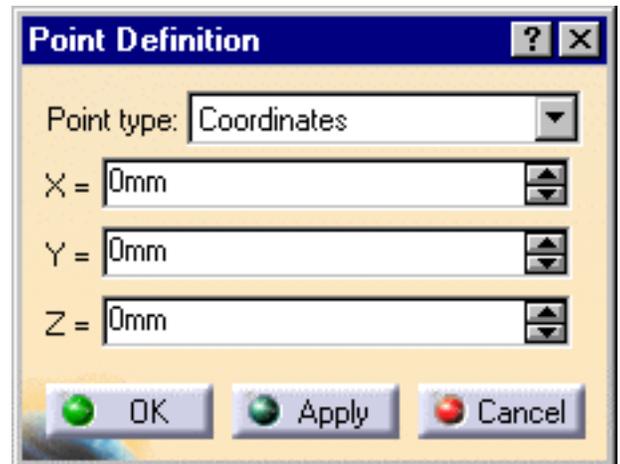
- [by coordinates](#)
- [on a curve](#)
- [on a plane](#)
- [on a surface](#)
- [at a circle center](#)
- [tangent points on a curve.](#)



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.

The corresponding point is displayed.

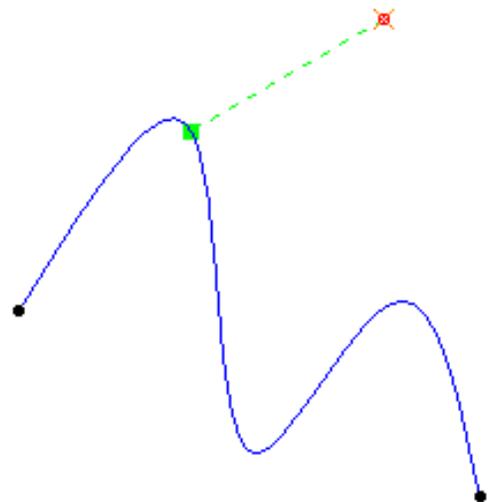
On curve

- 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 button to determine whether the new point is to be created:
 - a given distance along the curve from the reference point



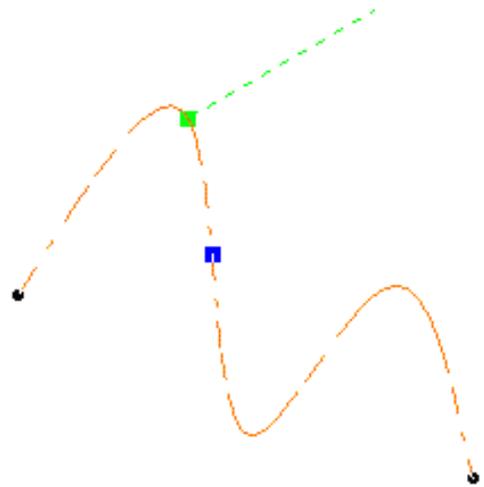
- a given ratio between the reference point and the curve's extremity.
- Enter the distance or ratio value.

The corresponding point is displayed.

You can click the Nearest extremity button to display the point at the nearest extremity of the curve.

You can click the Middle Point button to display the mid-point of the curve.

- You can 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).



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 local axis system's origin on the plane is taken as reference.

- 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.
- Select a line to take its orientation as reference direction or a plane to take its normal as reference direction.
You can also use the contextual menu to specify the X, Y, Z components of the reference direction.
- Enter a distance along the reference direction to display a point.

Circle center

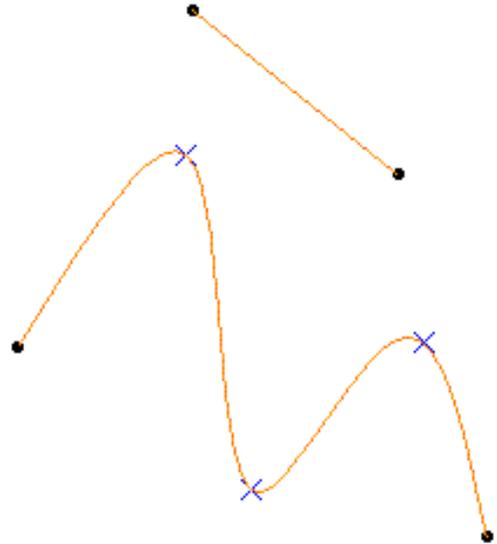
- Select a circle or circular arc.

A point is displayed at the circle center.

Tangent on curve

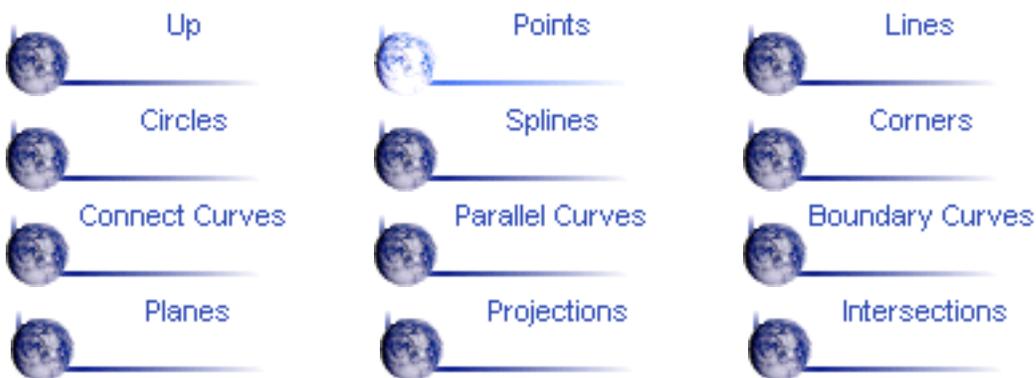
- Select a curve and a direction line.

A point is displayed at each tangent.



3. Click OK to create the point.

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



Creating Lines



This task shows the various methods for creating lines:

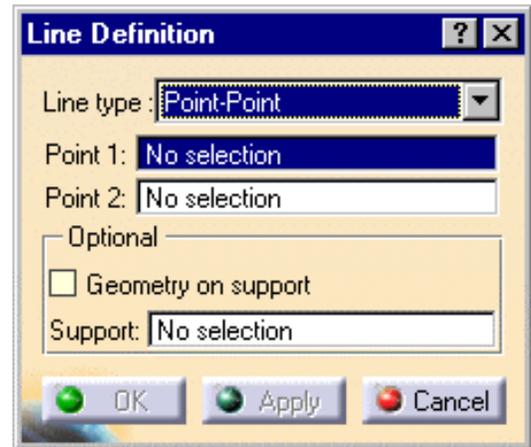
- [point to point](#)
- [point and direction](#)
- [angle or normal to curve](#)
- [tangent to curve](#)
- [normal to surface](#).



1. Click the Line icon .

The Line Definition dialog box appears.

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



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

Point - Point

- Select two points.

The corresponding line is displayed.

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.



Start and End points are specified by entering distance values or by using the graphic manipulators.

You can reverse the direction of the line by either clicking the displayed vector or selecting the Reverse Direction button.

Angle or normal to curve

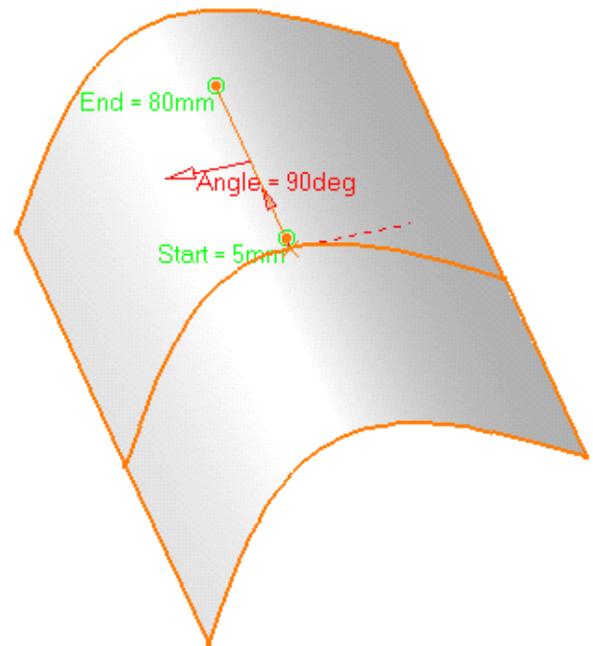
- Select a reference Curve and a Support surface containing that curve.
- Select a Point on the curve.
- Enter an Angle value.

A line is displayed at the given angle with respect to the tangent to 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.



Tangent to curve

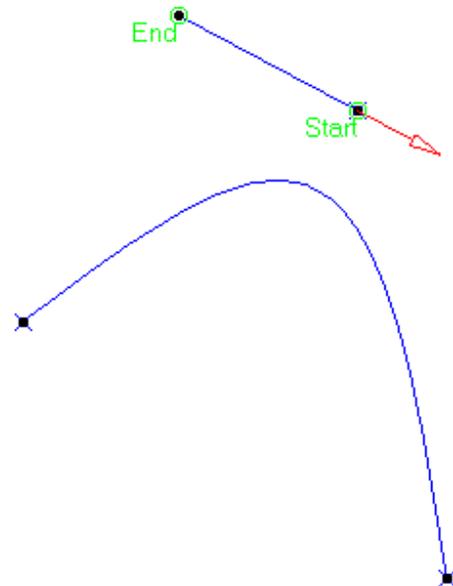
- Select a reference Point and a Curve.

A vector tangent to the curve is displayed at the reference point.

Proposed Start and End points of the new line are shown.

- Specify Start and End points to define the new line.

The corresponding line is displayed.



Normal to surface

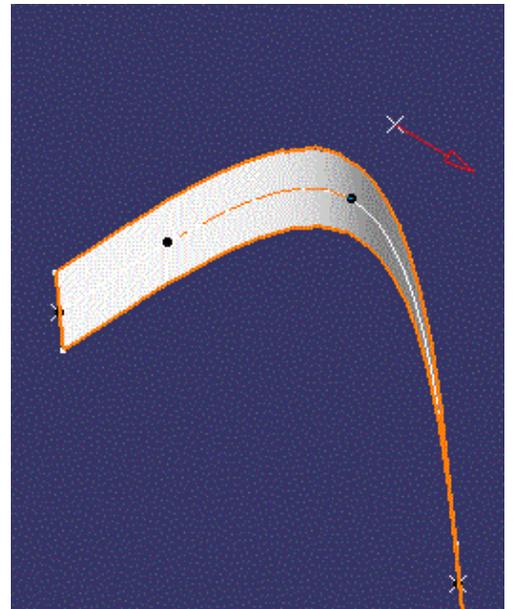
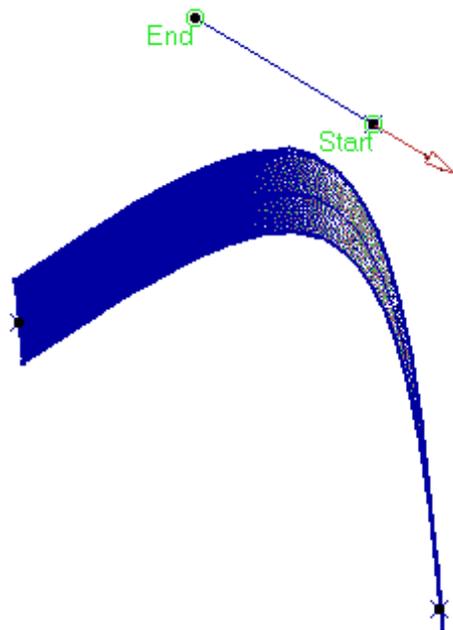
- Select a reference Surface and a Point.

A vector normal to the surface is displayed at the reference point. Proposed Start and End points of the new line are shown.

- Specify Start and End points to define the new line.

The corresponding line is displayed.

3. For most line types you can select the Geometry on Support check box if you want the line to be projected onto a support surface. In this case just select a support surface. The figure below illustrates this case.



4. Click OK to create the line.

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





Up



Circles



Connect Curves



Planes



Points



Splines



Parallel Curves



Projections



Lines



Corners



Boundary Curves



Intersections

Creating Circles



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

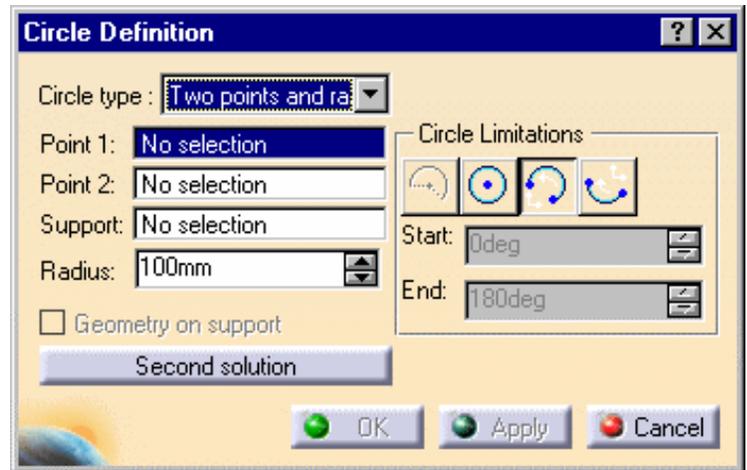
- [center and radius](#)
- [center and point](#)
- [two points and radius](#)
- [three points](#)
- [bitangent and radius](#)
- [bitangent and point](#)
- [tritangent.](#)



1. Click the Circle icon .

The Circle Definition dialog box appears.

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

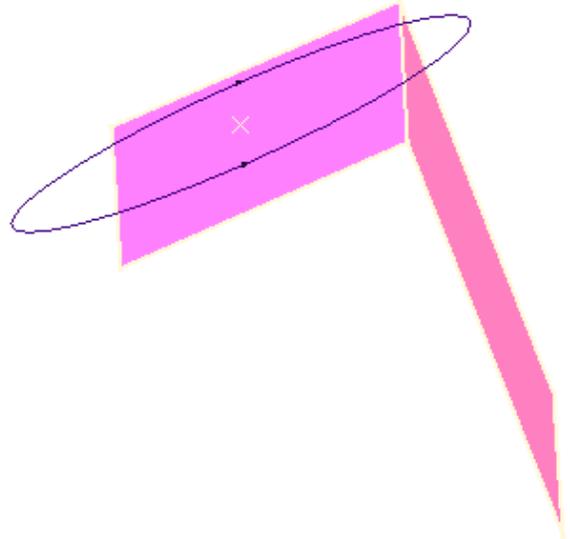


Center and radius

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

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

For a circular arc, you can specify the Start and End angles of the arc.



If a support surface is selected, the plane tangent to the surface at the selected point is used.

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.

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.

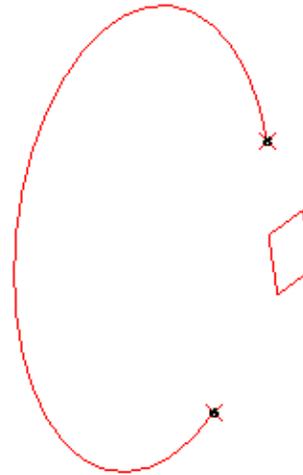
Two points and radius

- Select two points where the circle is to be created.
- 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 trimmed or complementary arc using the two selected points as end points.

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



Three points

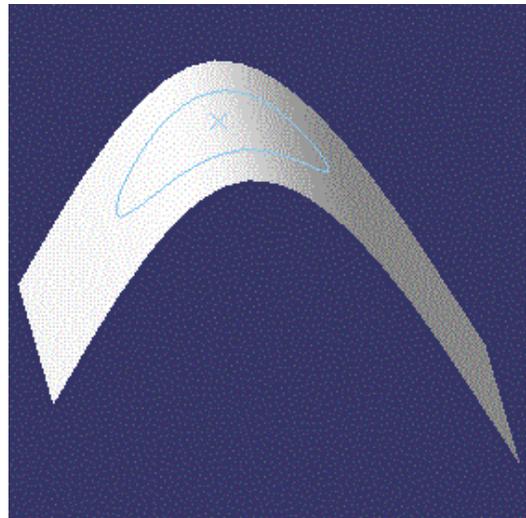
- Select three points where the circle is to be created.

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



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

In this case just select a support surface.



Bitangent and radius

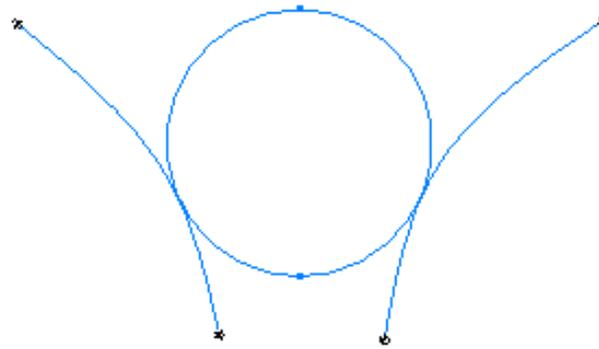
- Select two curves (in Curve 1 and Element 2 fields) to which the circle is to be tangent.
- Select a Support surface.
- Enter a Radius value.
- Several solutions may be possible, so click in the region where you want the circle to be.

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.

Bitangent and point

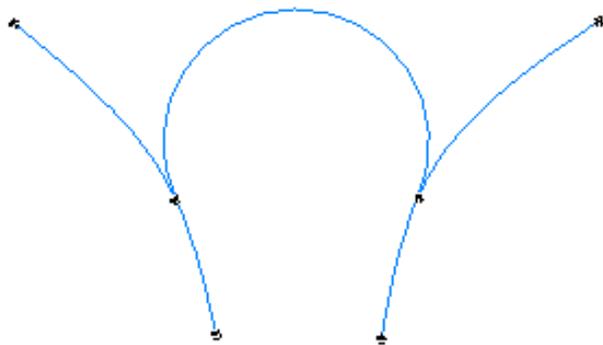
- Select two curves to which the circle is to be tangent.
- Select a Point on the second curve.
- Select a Support plane or surface.
- Several solutions may be possible, so click in the region where you want the circle to be.

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

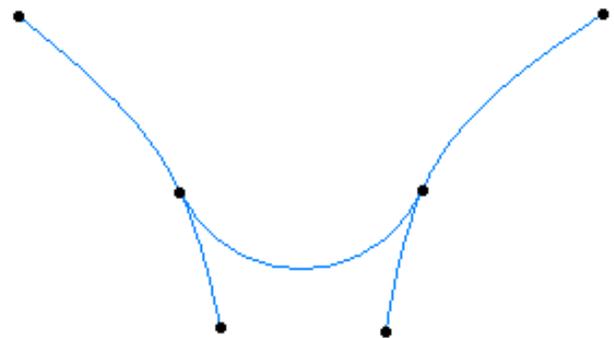


Complete circle

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

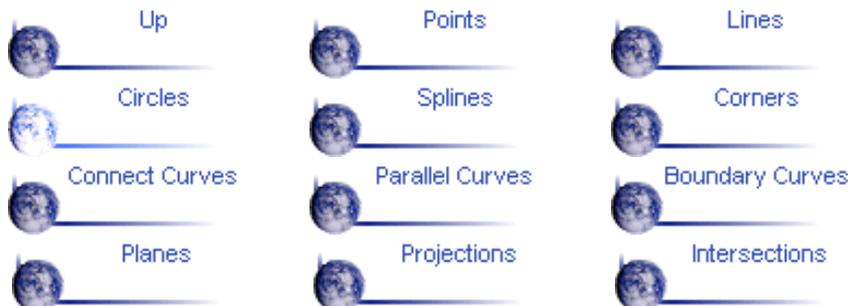
Tritangent

- Select three curves to which the circle is to be tangent.
- Select a Support surface.
- 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.

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

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



Creating Splines



This task shows the various methods for creating spline curves.

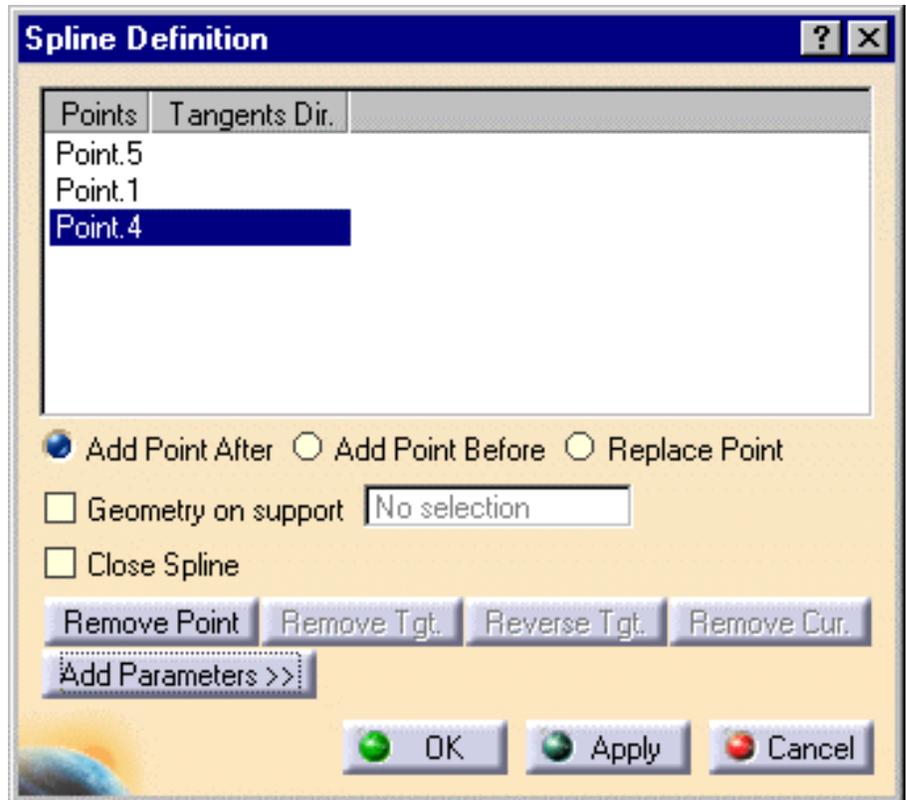


1. Click the Spline icon .

The Spline Definition dialog box appears.

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

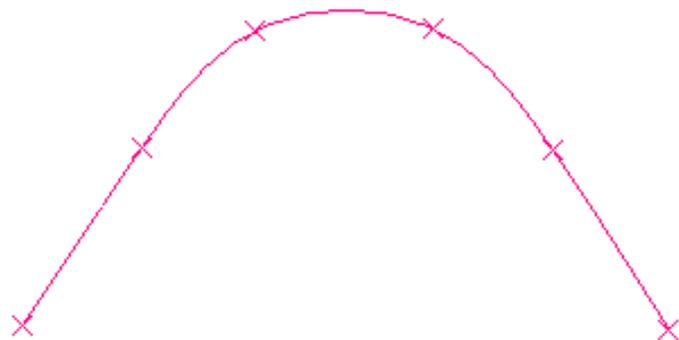
An updated spline is visualized each time a point is selected.



3. You can select the Geometry on support check box 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.

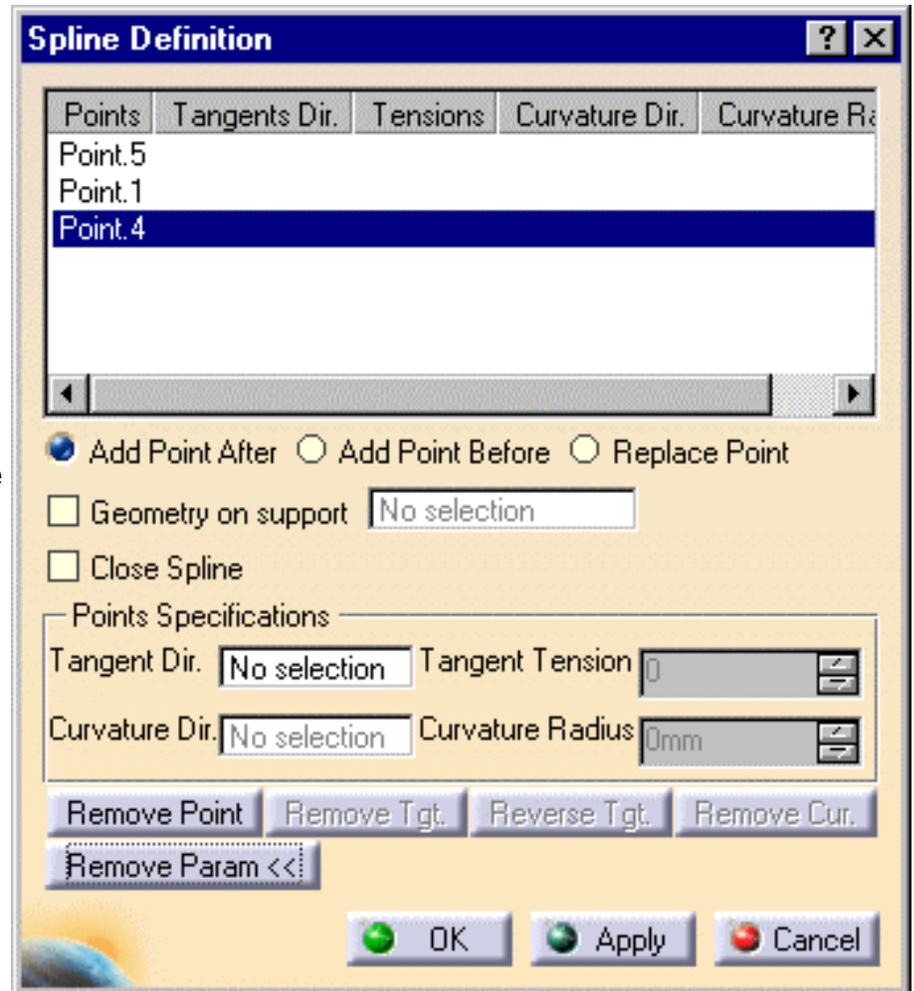


In the figure above, the spline was created on a planar support grid.

4. If you want to set tangency conditions at the spline's extremities, you can right-click on the Tangent Dir. field to display a contextual menu. Using this menu, you can:

- Edit components (specify the tangent direction at the start and end points of the spline)
- Specify the line direction by choosing the X, Y or Z axis.

You only have to select a plane or a line to create a tangent.



5. 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.



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.

6. Click OK to create the spline.

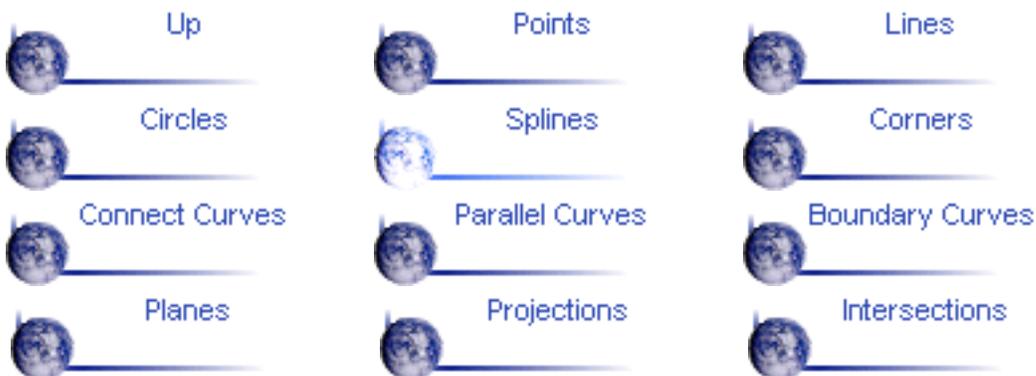
The spline (identified as Spline.xxx) is added to the specification tree.



To add a parameter to a point, select a line in the Points list. This list is highlighted.

You have two possibilities:

- extended parameters
- select any line or plane for the direction.



Creating Corners

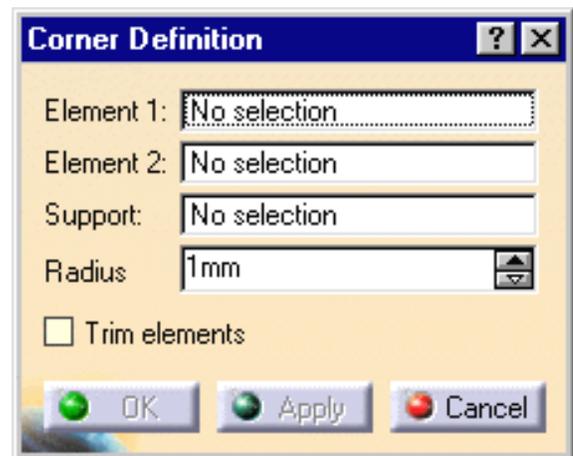


This task shows you how to create a corner between two curves or between a point and a curve.



1. Click the Corner icon .

The Corner Definition dialog box appears.



2. Select a curve or a point as first reference element.

3. Select a curve as second reference element.

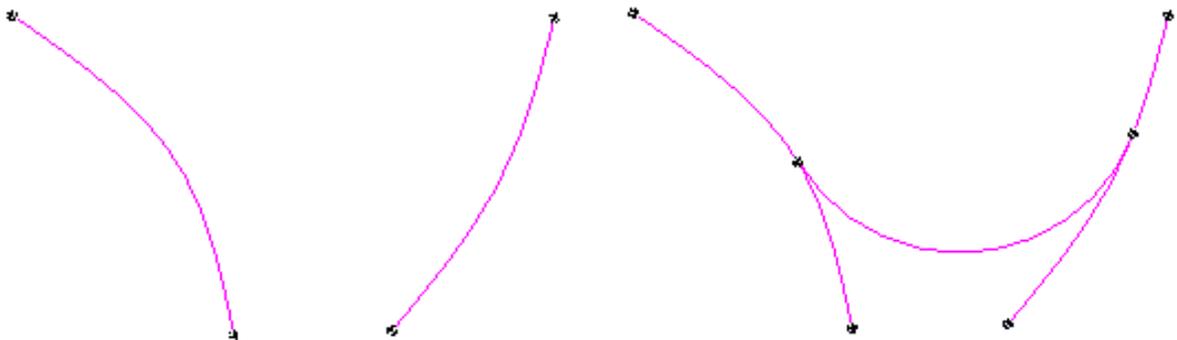
The corner will be created between these two references.

4. Select the Support plane or planar surface.

The reference elements must lie on this support.

5. Enter a Radius value.

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

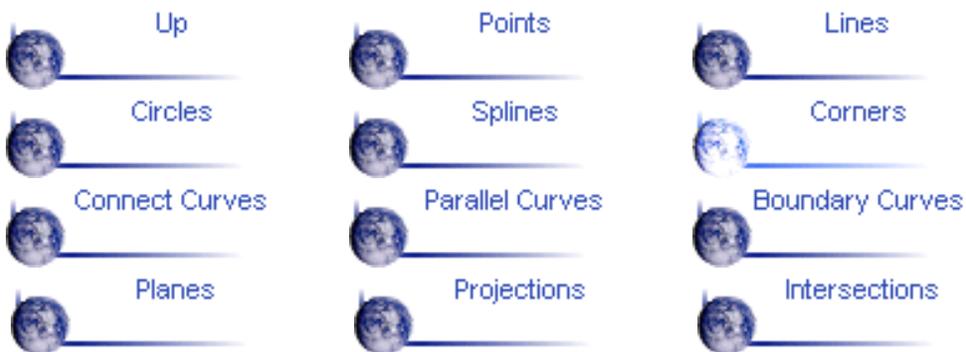


7. You can select the Trim elements check box if you want to trim and assemble the two reference elements to the corner.



8. Click OK to create the corner.

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



Creating Connect Curves



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



Open the Connect.CATPart document from the online/Samples/ShapeDesign directory.



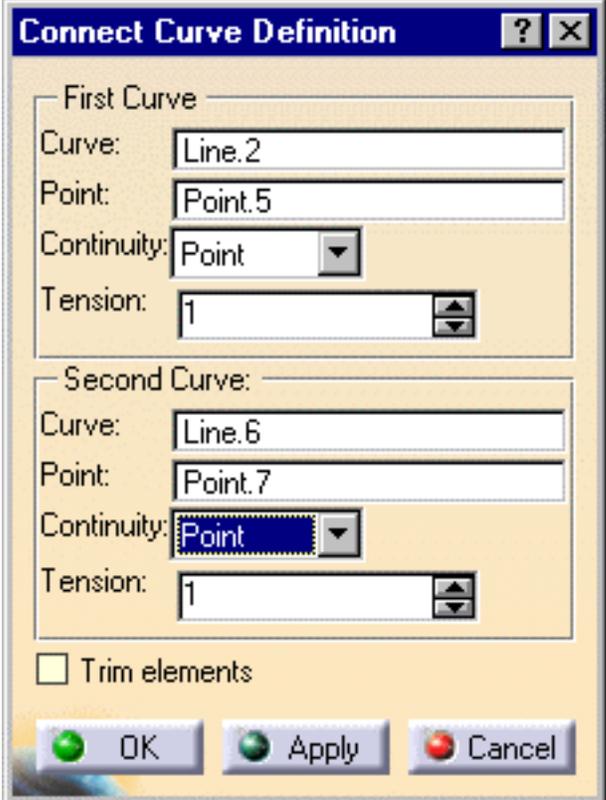
1. Click the Connect Curve icon .

The Connect Curve Definition dialog box appears.

2. Select the first Curve and a Point on the curve.

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

4. If needed, enter a tension value.



The dialog box is titled "Connect Curve Definition" and contains the following fields and controls:

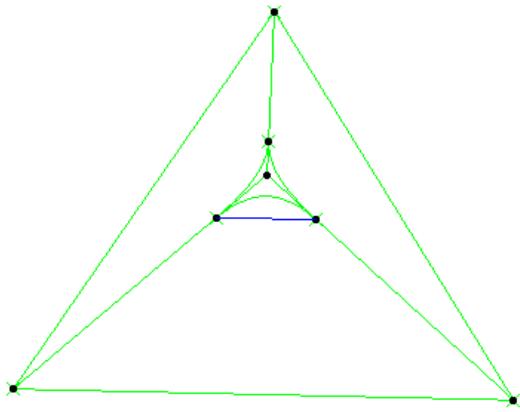
- First Curve:**
 - Curve: Line.2
 - Point: Point.5
 - Continuity: Point (dropdown menu)
 - Tension: 1 (spin box)
- Second Curve:**
 - Curve: Line.6
 - Point: Point.7
 - Continuity: Point (dropdown menu)
 - Tension: 1 (spin box)
- Trim elements
- Buttons: OK, Apply, Cancel

5. Select the second Curve and a Point on the curve.

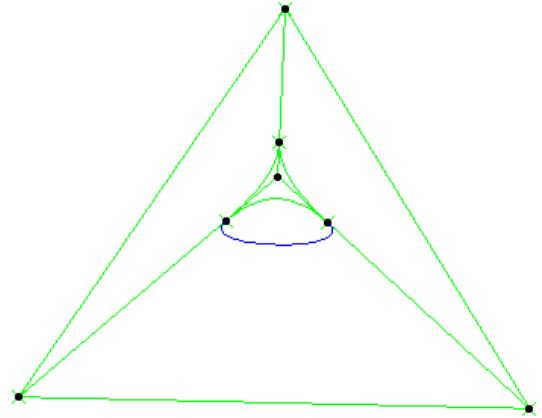
6. Use the combo to specify the desired Continuity type:
Point, Tangency or Curvature.

7. If needed, enter a Tension value.

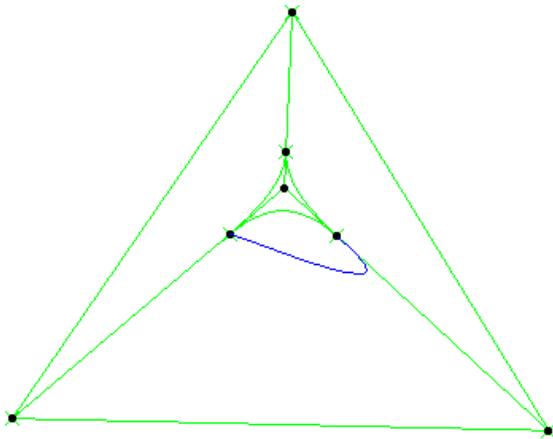
The connect curve is displayed between the two selected points according to the specified continuity and tension values.



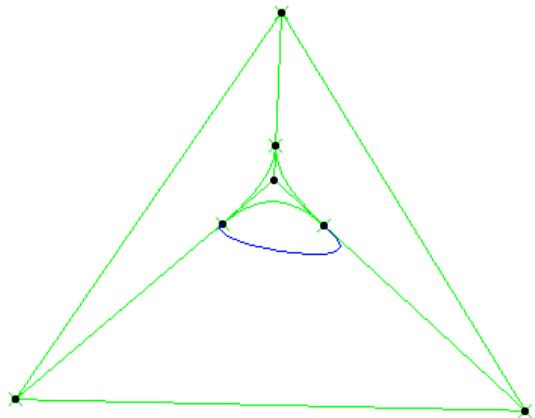
Connect curve with point continuity at both points.



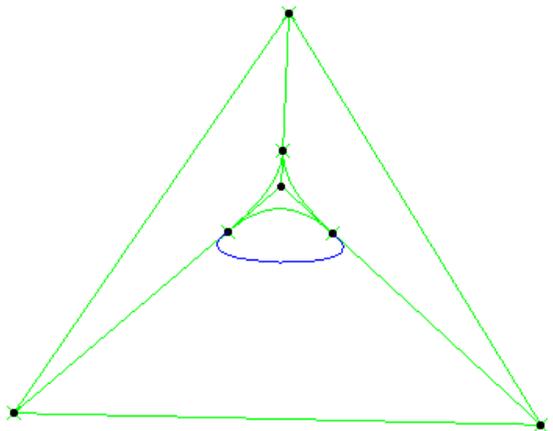
Connect curve with point continuity at one point and tangent continuity at the other.



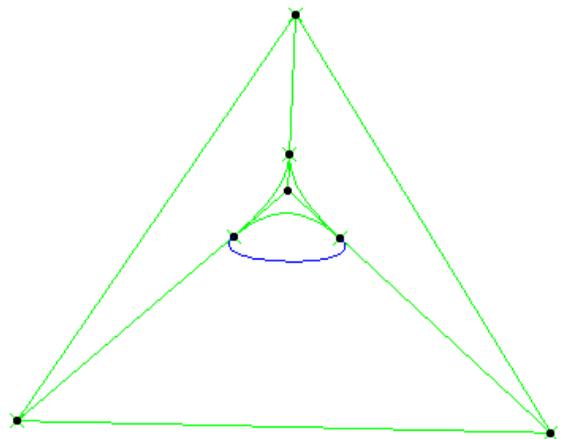
Connect curve with point continuity at one point and curvature continuity at the other.



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



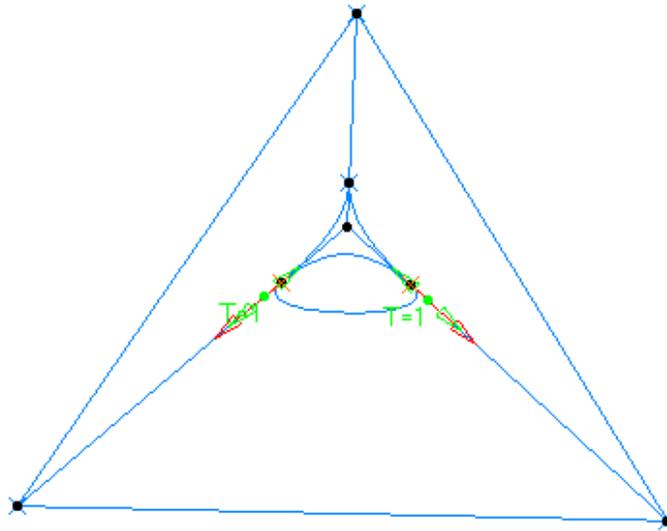
Connect curve with curvature continuity at both points.



Connect curve with tangent continuity at both points.

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

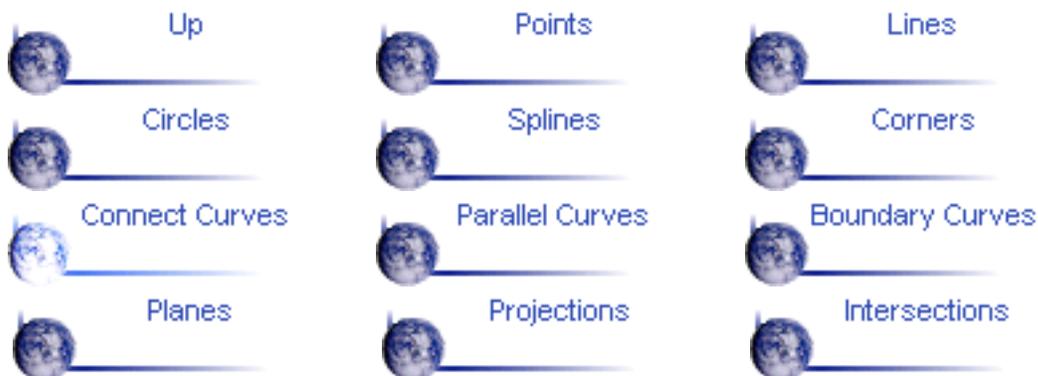
A graphic manipulator also allows you to modify the tension at the extremity of the connect curve.



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

10. Click OK to create the connect curve.

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



Creating Parallel Curves



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

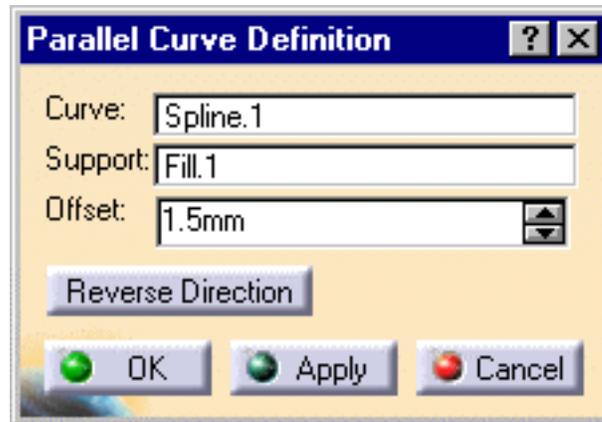


Open the Parallelcurves.CATPart document from the online/Samples/ShapeDesign directory.



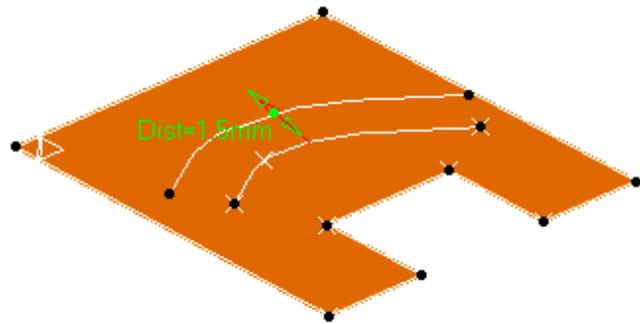
1. Click the Parallel Curve icon .

The Parallel Curve Definition dialog box appears.

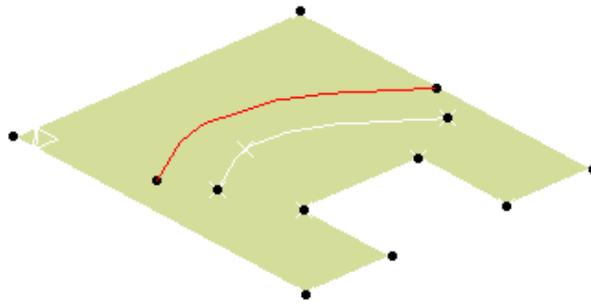


2. Select the reference Curve to be offset.
3. Select the Support plane or surface on which the reference curve lies.
4. Specify the Offset by entering a value or using the graphic manipulator.

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

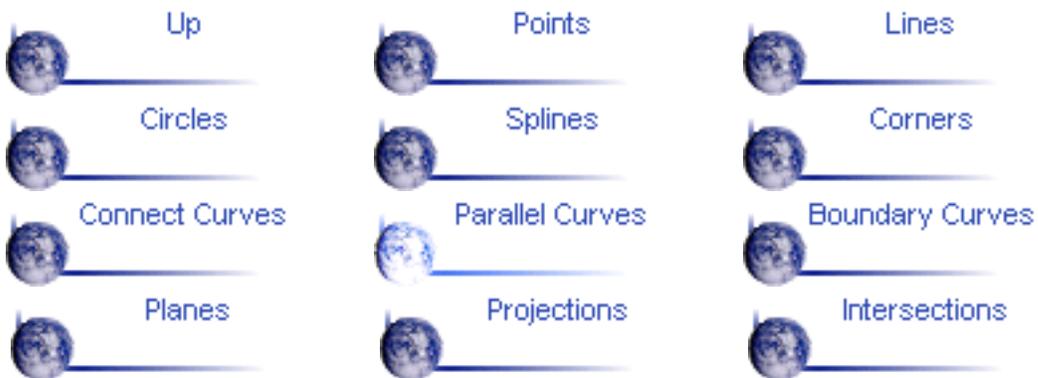


5. Click OK to create the parallel curve.



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

 You can use the Reverse Direction button to display the parallel curve on the other side of the reference curve.



Creating Boundary Curves



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

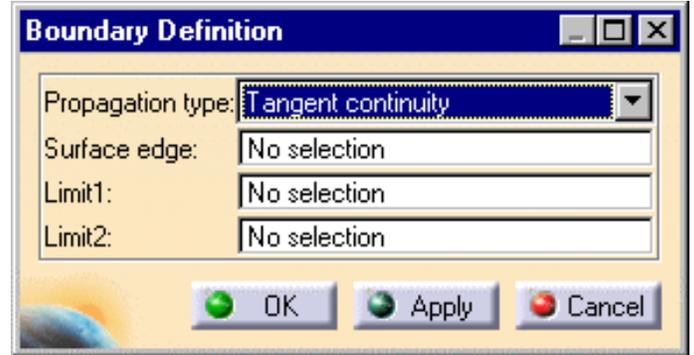


Open the Boundaries.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Boundary icon .

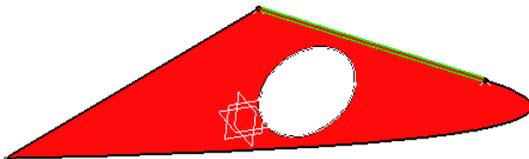
The Boundary Definition dialog box appears.



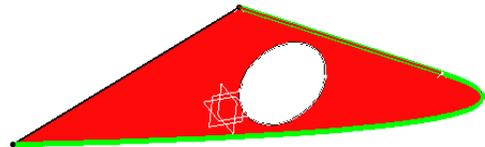
new

2. Use the combo to choose the Propagation type:

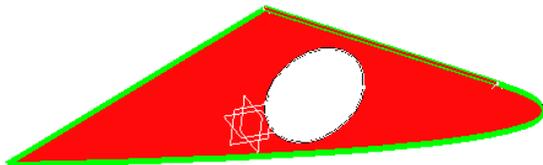
- Complete boundary: the selected edge is propagated around the entire surface boundary.
- Point continuity: the selected edge is propagated around the surface boundary until a point discontinuity is met.
- Tangency continuity: the selected edge is propagated around the surface boundary until a tangent discontinuity is met.
- None: no propagation or continuity condition is imposed, only the selected edge is kept.



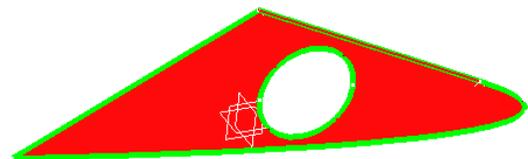
None



Tangent continuity



Point continuity



All contours

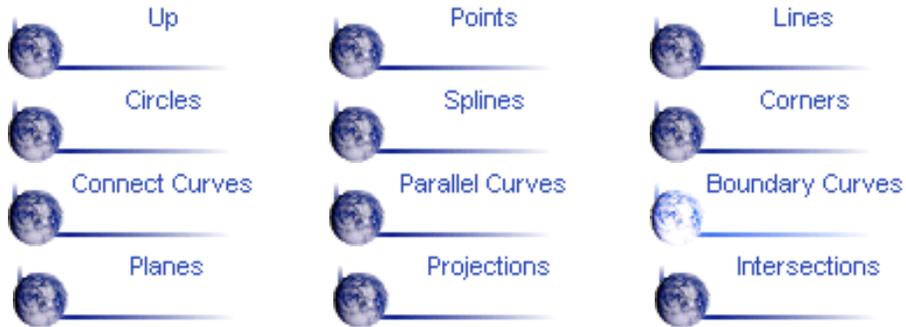
3. Select a Surface edge.

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

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

5. Click OK to create the boundary curve.

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



Creating Planes



This task shows the various methods for creating planes:

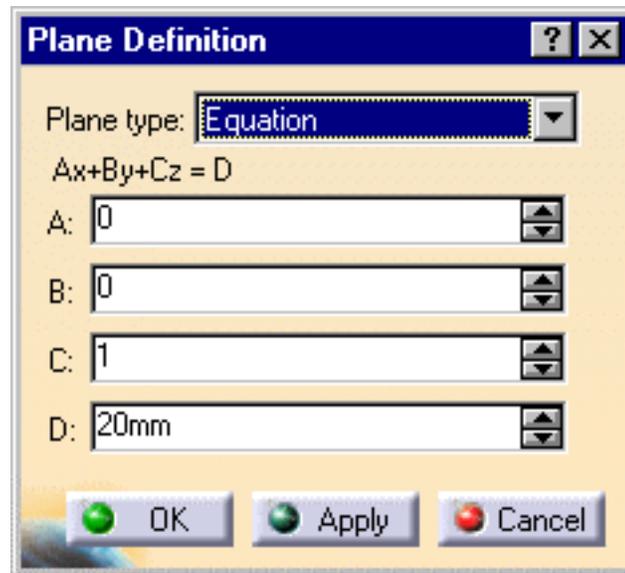
- [from its equation](#)
- [through three points](#)
- [through two lines](#)
- [through a point and a line](#)
- [through a planar curve](#)
- [tangent to a surface](#)
- [normal to a curve](#)
- [offset from a plane](#)
- [offset through point](#)
- [at an angle to a plane](#)
- [mean plane through several points.](#)



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.

Equation

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

Through three points

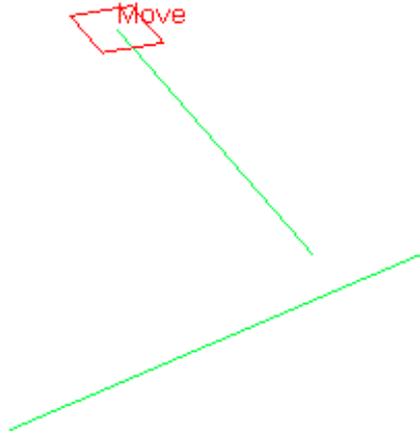
- Select three points.

The plane passing through the three points is displayed.

Through two lines

- Select two lines.

The plane passing through the two line directions is displayed.



Through point and line

- Select a Point and a Line.

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

Through planar curve

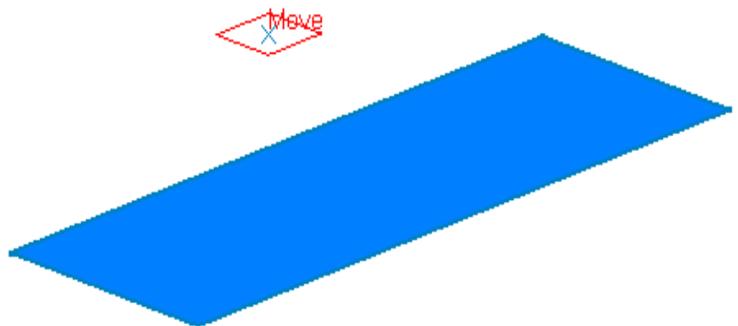
- Select a planar Curve.

The plane containing the curve is displayed.

Tangent to surface

- Select a reference Surface and a Point.

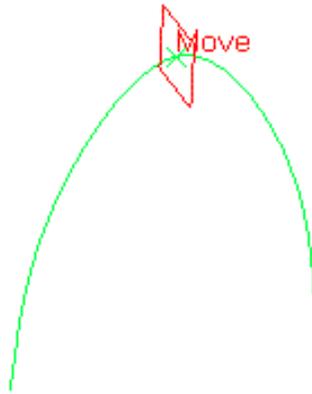
A plane is displayed tangent to the surface at the specified point.



Normal to curve

- Select a reference Curve and a Point.

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



Offset from plane

- Select a reference Plane then enter an Offset value.

A plane is displayed offset from the reference plane.

Offset through point

- Select a reference Plane and a Point.

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

Angle or normal to plane

- Select a reference Plane and a Rotation axis.
- Enter an Angle value.

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

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.

3. Click OK to create the plane.

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



 Up

 Circles

 Connect Curves

 Planes

 Points

 Splines

 Parallel Curves

 Projections

 Lines

 Corners

 Boundary Curves

 Intersections

Creating Projections



This task shows you how to create geometry by projecting an element onto a support element.

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.



Open the Projection.CATPart document from the online/Samples/ShapeDesign directory.



If you select Normal as Projection type:

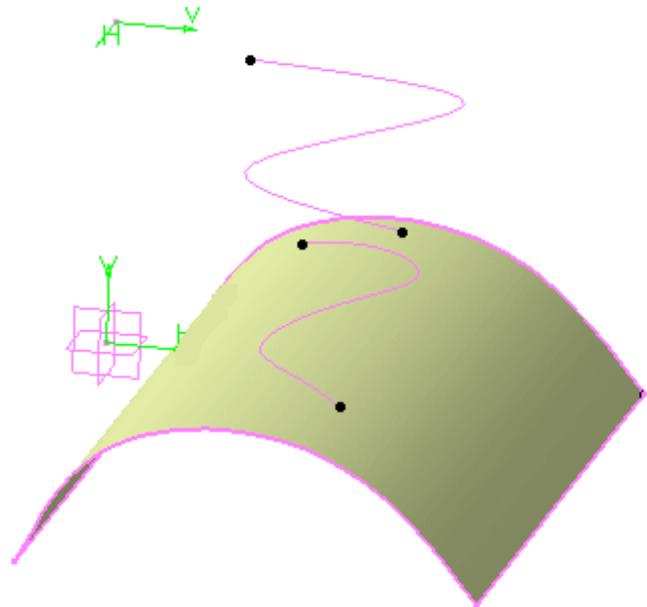
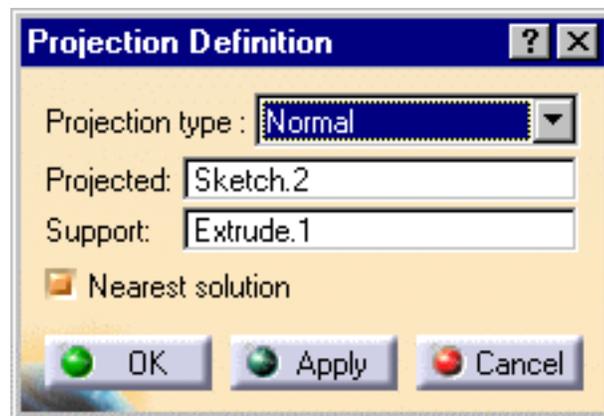
1. Click the Projection icon



The Projection Definition dialog box appears.

2. Select the element to be Projected.
3. Select the Support element.
4. Use the combo to specify the direction type for the projection:

- Normal.
In this case, projection is done normal to the support element.



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

6. Click OK to create the projection element.

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

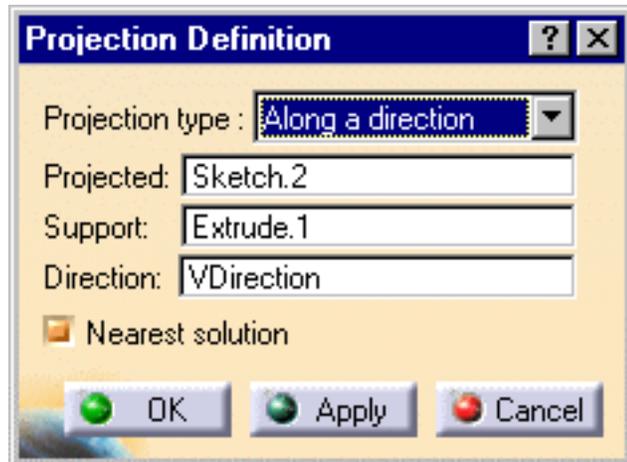


If you select Along a direction as projection type:

1. Click the Projection icon



The Projection Definition dialog box appears.



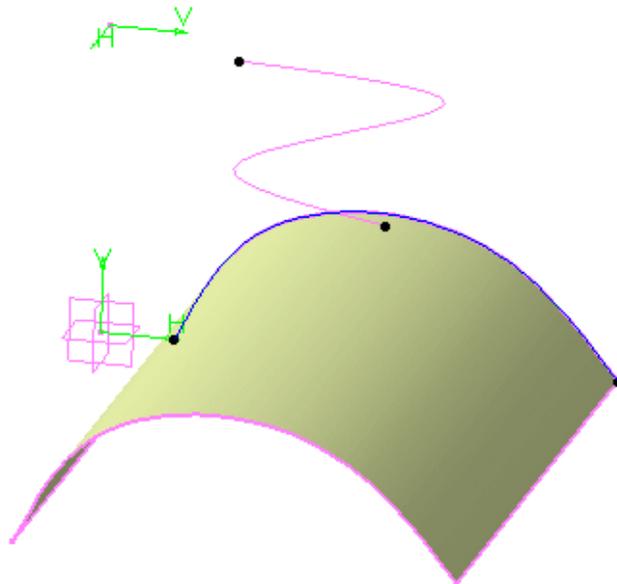
2. Select the element to be Projected.

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

- Along a direction. In this case, the projection is done along the selected direction.

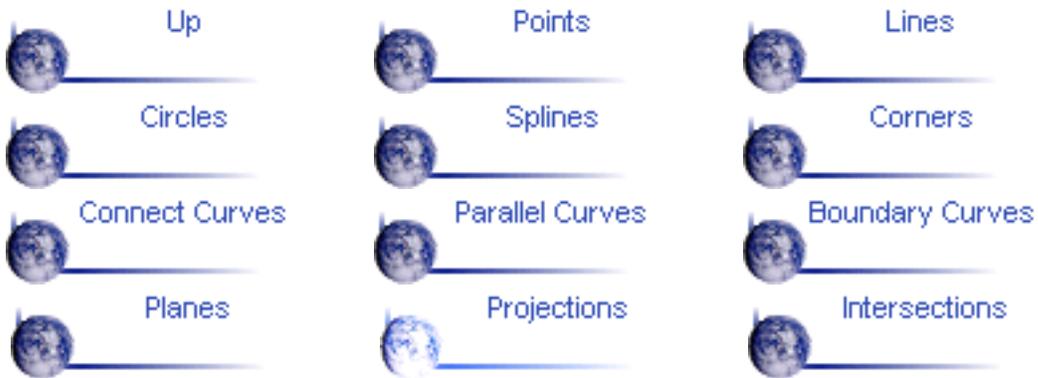
4. Select the Direction, that is a line to take its orientation as the translation direction or a plane to take its normal as the translation direction.

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



5. Whenever several projections are possible, you can select the Nearest Solution check box to keep the nearest projection.
6. Click OK to create the projection element.

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



Creating Intersections



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

You can intersect:

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

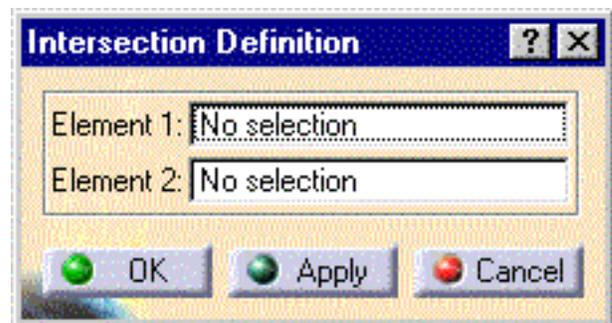


Open the Intersectsurface.CATPart and the Intersectsurf.CATPart documents from the online/Samples/ShapeDesign directory.



1. Click the Intersection icon .

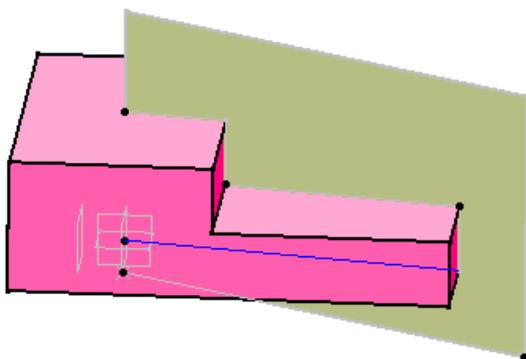
The Intersection Definition dialog box appears.



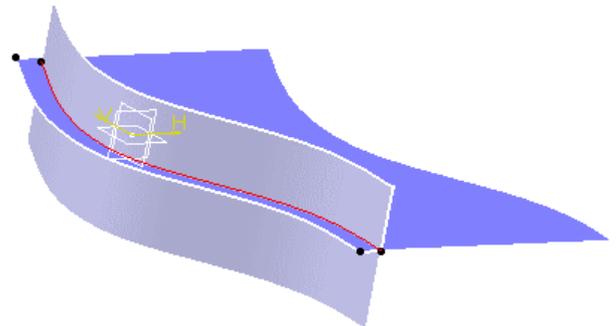
2. Select the two elements to be intersected.

The intersection is displayed.

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.



3. Click OK to create the intersection element.

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



 Up

 Circles

 Connect Curves

 Planes

 Points

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 Parallel Curves

 Projections

 Lines

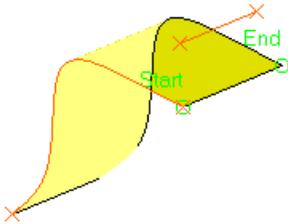
 Corners

 Boundary Curves

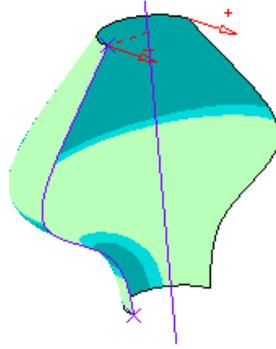
 Intersections

Creating Surfaces

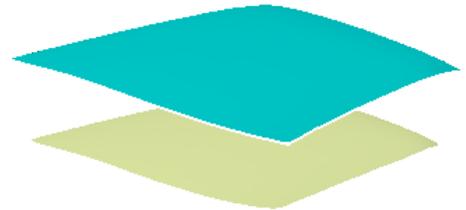
CATIA - Generative Shape Design allows you to model both simple and complex surfaces using techniques such as lofting, sweeping and filling.



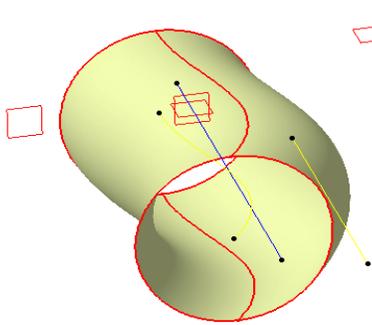
Creating Extruded Surfaces



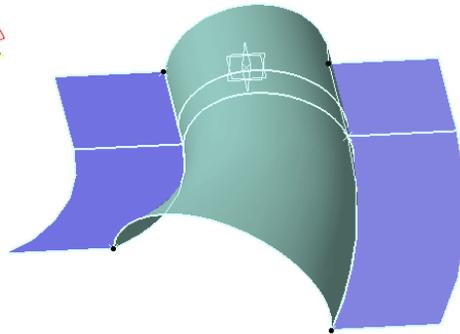
Creating Revolution Surfaces



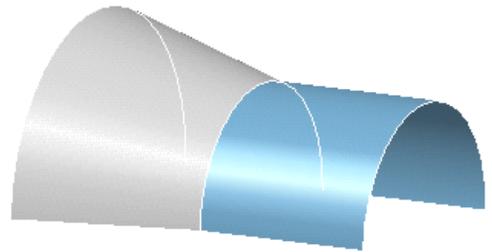
Creating Offset Surfaces



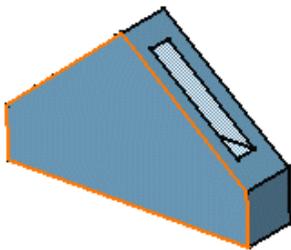
Creating Swept Surfaces



Creating Fill Surfaces



Creating Lofted Surfaces



Extracting Geometry

 Up
 Performing Operations

 Creating Wireframe Geometr
 Editing Geometry

 Creating Surfaces
 Using Tools

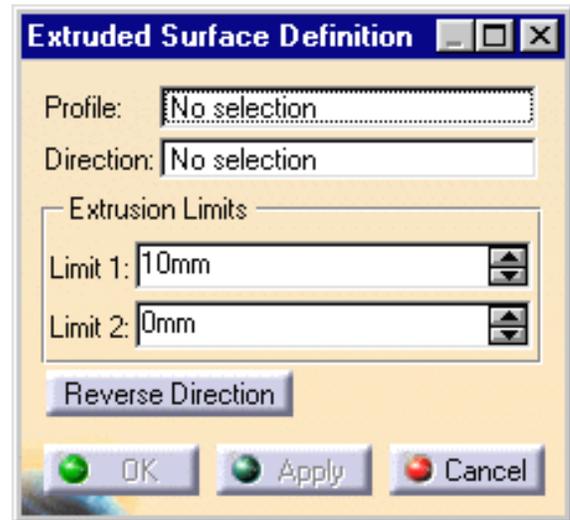
Creating Extruded Surfaces



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

1. Click the Extrude icon .

The Extruded Surface Definition dialog box appears.

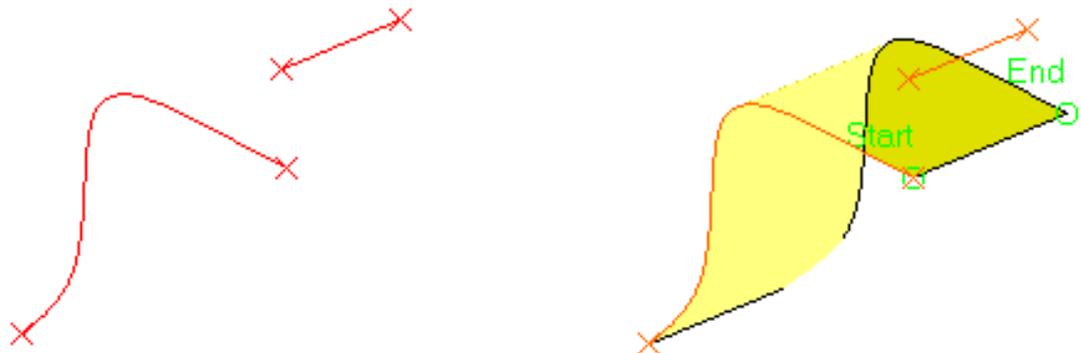


2. Select the Profile to be extruded and specify the desired extrusion Direction.

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.

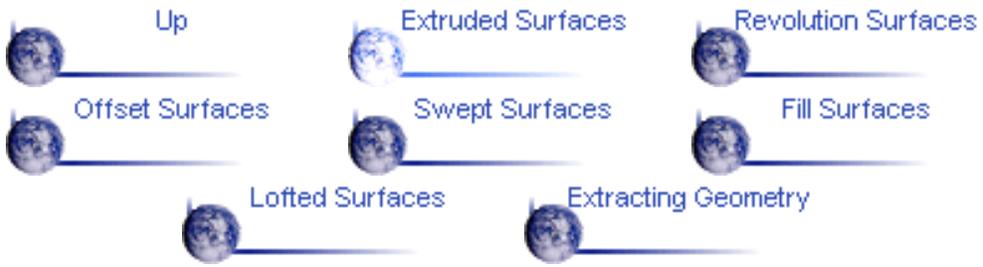
3. Enter values or use the graphic manipulators to define the start and end limits of the extrusion.



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

5. Click OK to create the surface.

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



Creating Revolution Surfaces

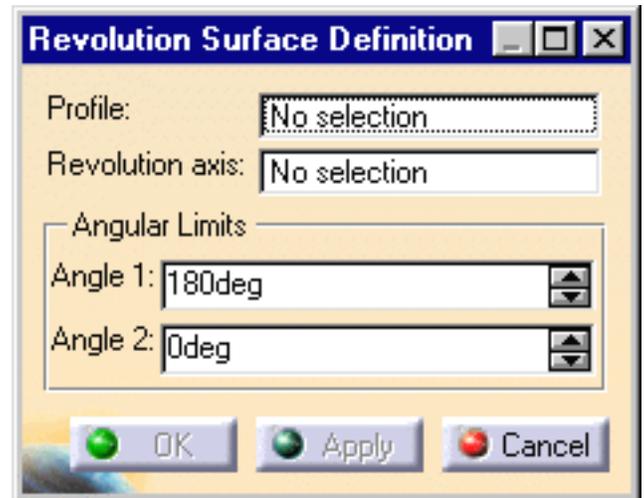


This task shows how to create a surface by revolving a profile around an axis.



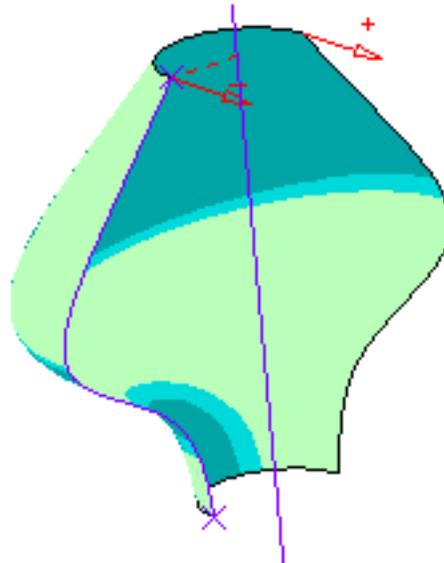
1. Click the Revolve icon .

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.

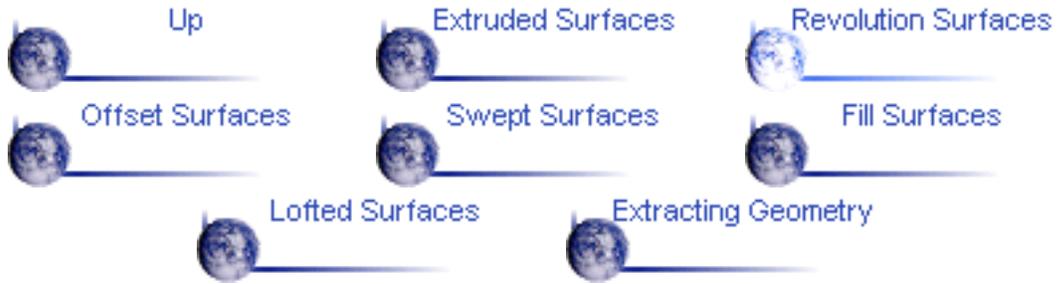


4. Click OK to create the surface.

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



There must be no intersection between the axis and the profile.



Creating Offset Surfaces



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

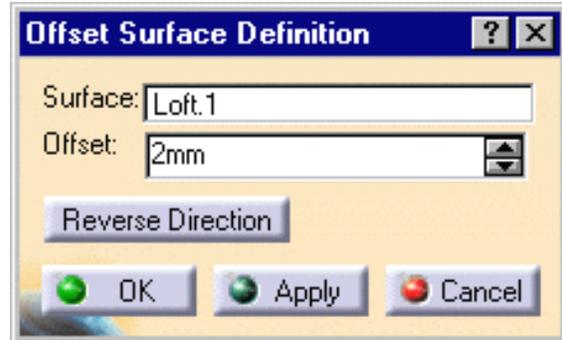


Open the Offset.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Offset icon .

The Offset Surface Definition dialog box appears.

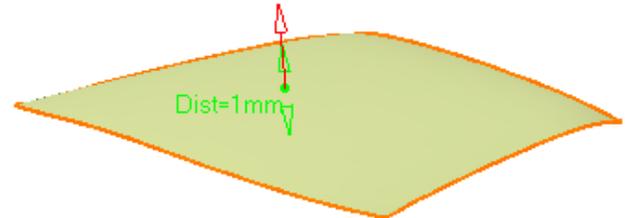


2. Select the Surface to be offset.

3. Specify the Offset by entering a value or using the graphic manipulator.

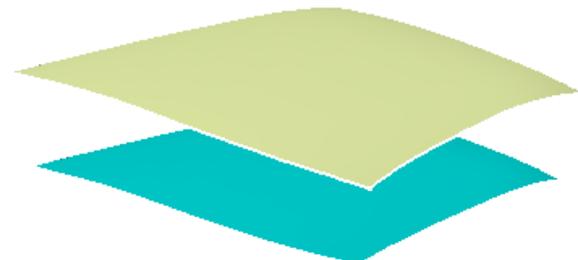
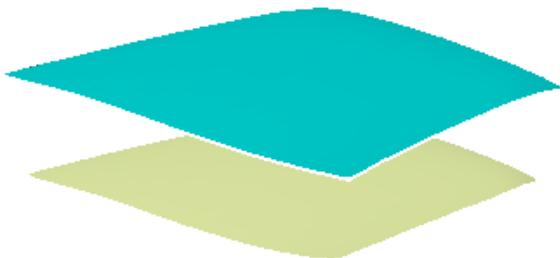
The offset surface is displayed normal to the reference surface.

4. An arrow indicates the proposed direction for the offset.



5. Click OK to create the surface.

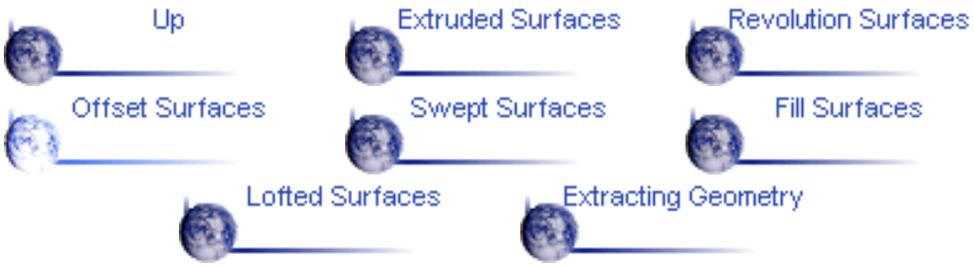
You can display the offset surface on the other side of the reference surface by clicking either the arrow or the Reverse Direction button.



The figure above shows the offset after clicking the Reverse Direction button

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





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)
- along one or two guide curves while respecting a spine.

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

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

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

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.

You can sweep an [implicit circular profile](#) along a spine. This profile is defined by:

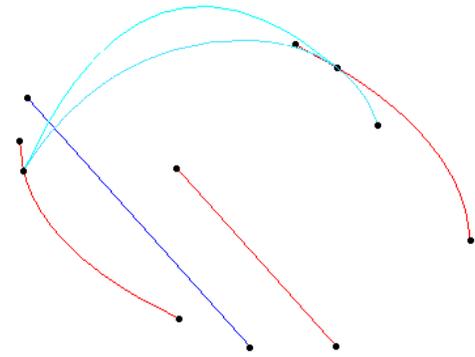
- three guide curves
- two guide curves and a radius value
- a center curve and two angle values defined from a reference curve (that also defines the radius)
- a center curve and a radius.

Swept Surfaces using an Explicit Profile



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

You can use the wireframe elements shown in this figure.



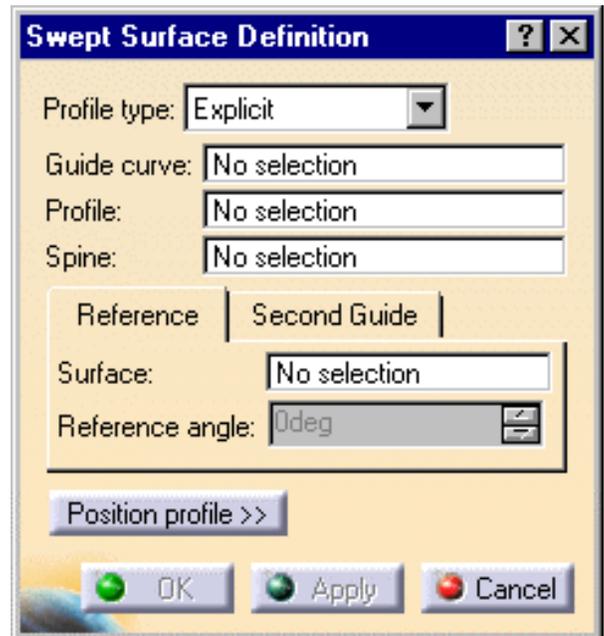
Open the Sweep.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Sweep icon .

The Swept Surface Definition dialog box appears.

2. Use the combo to choose the Explicit profile type.



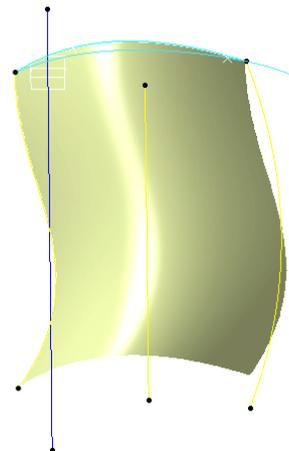
3. Select a Guide curve.

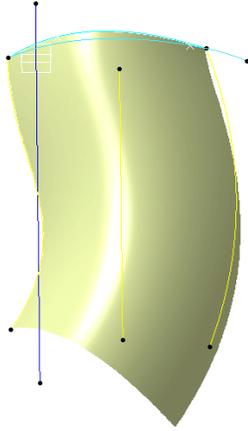
4. Select the planar Profile to be swept out.

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

The figure below shows the result obtained when you include a linear spine element in the definition.

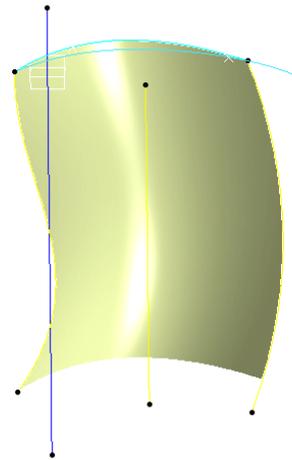
The figure below shows the result obtained after selecting the first guide curve and the profile.





The figure below shows the result obtained when you include a linear spine element in the definition.

6. If needed, select a Second guide curve.



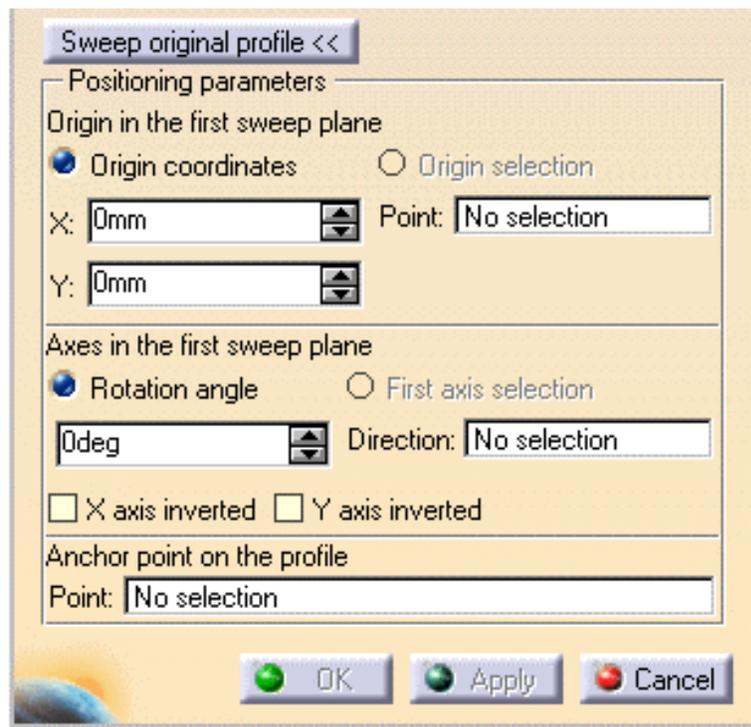
7. If you want to control the position of the profile during the sweep, you can select a reference Surface. You can impose a Reference angle on this surface.

new

By default, the sweep follows the mean plane of the spine, otherwise it follows the reference.

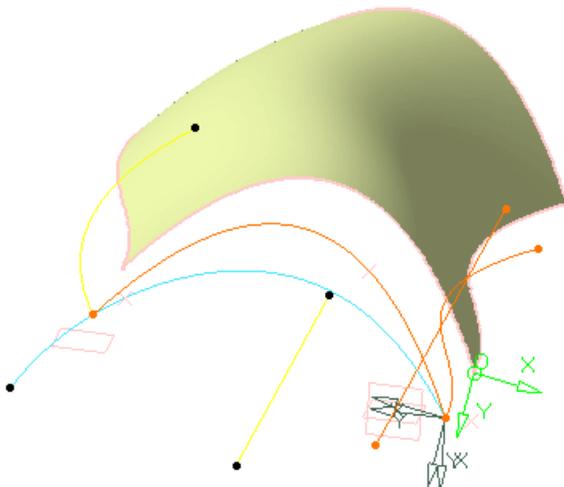
8. If you want to manually position the profile, click the Position profile >> button to access the following positioning parameters.

These parameters and the graphic manipulators will 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 x-axis orientation (while keeping the y-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 to the profile.

If you want to go back to the original profile, click the Sweep original profile << button to access the original positioning parameters.



9. Click OK to create the swept surface.

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



Swept Surfaces using a Linear Profile



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

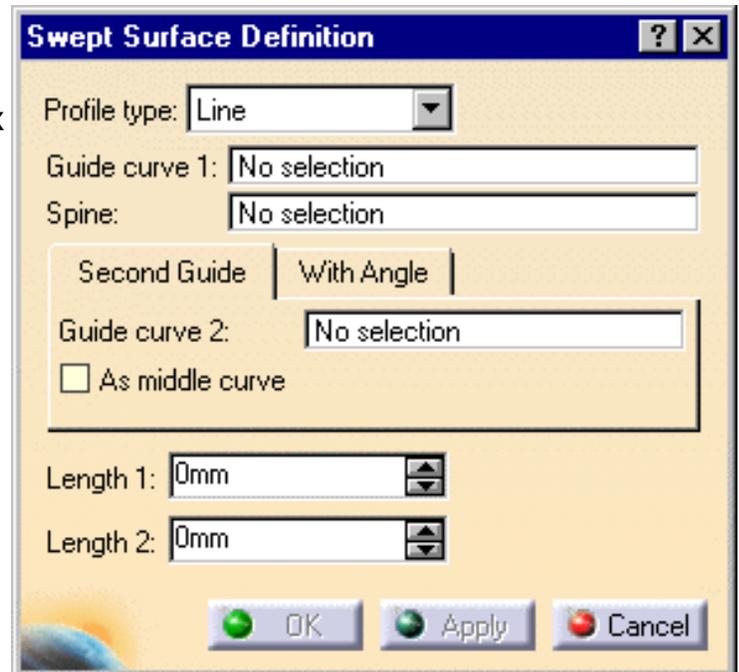


1. Click the Sweep icon .

The Swept Surface Definition dialog box appears.

2. Use the combo to choose the Line profile type.

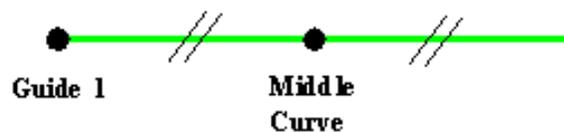
The possible cases are described below.



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

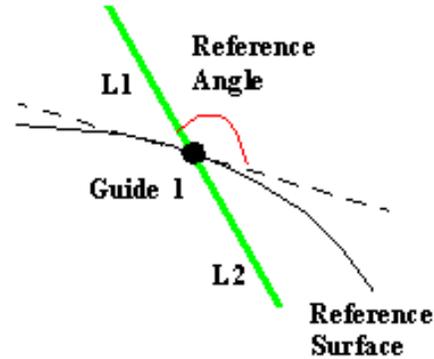
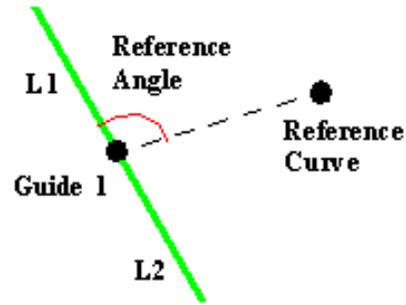


- Select two guide curves.
- Select the As middle curve check box to use the second guide curve as middle curve.



- Select a guide curve, then select the With Angle tab to specify a reference curve and a reference angle.
- You can enter one or two length values to define the width of the swept surface.

- Select a guide curve, then select the With Angle tab to specify a reference surface and a reference angle.
- You can enter one or two length values to define the width of the swept surface.



In any of the above cases, you can select a spine if you want to specify a spine different from the first guide curve.

3. Click OK to create the swept surface.

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

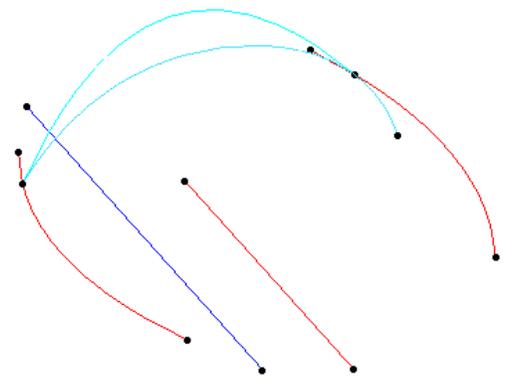


Swept Surfaces using a Circular Profile



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

You can use the wireframe elements shown in this figure.



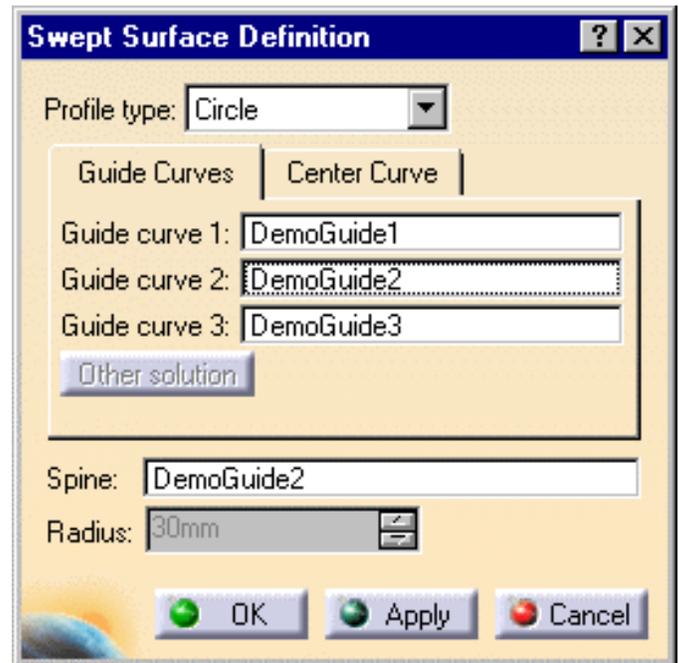
Open the Sweep.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Sweep icon .

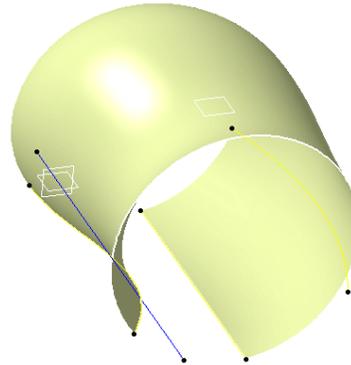
The Swept Surface Definition dialog box appears.

2. Use the combo to choose the Circle profile type.



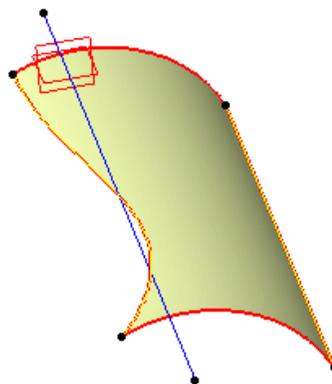
The two following cases are possible using guide curves.

- Select three guide curves.



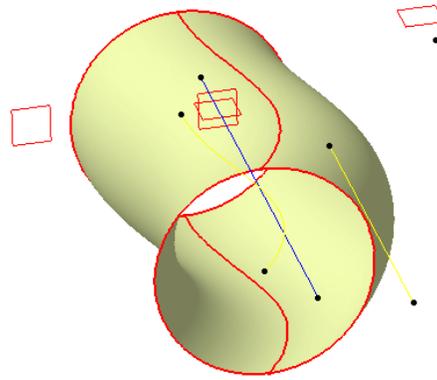
In the figure below, the radius value is 30.

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



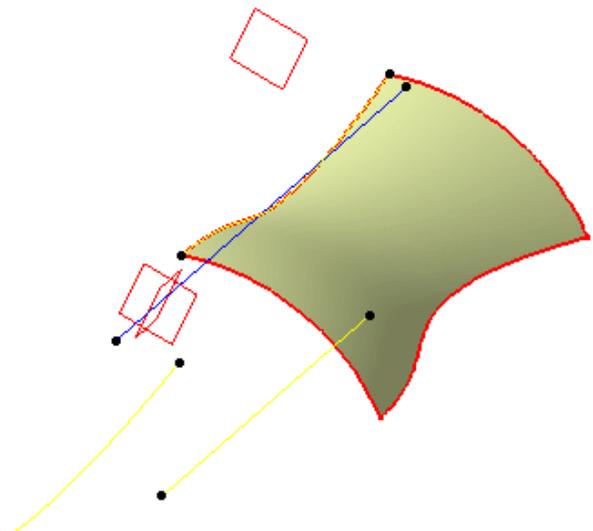
The two following cases are possible using a center curve.

- Select a Center Curve and enter a Radius value.



In the example above, we selected a spine

- 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 the following values:

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

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

3. Click OK to create the swept surface.

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



Creating Fill Surfaces



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



Open the Fill.CATPart document from the online/Samples/ShapeDesign directory.

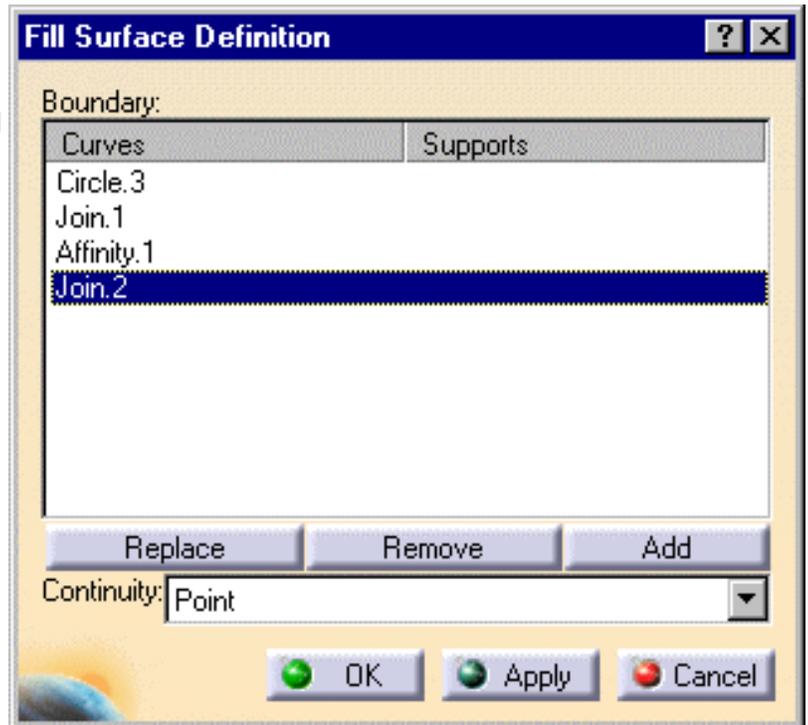


1. Click the Fill icon .

The Fill Surface Definition dialog box appears.

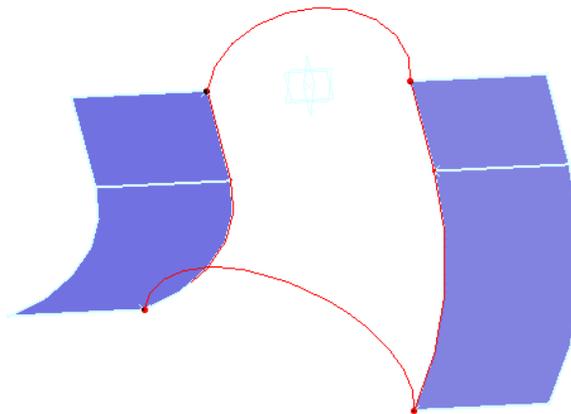
2. Select curves or surface edges to form a closed boundary.

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



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

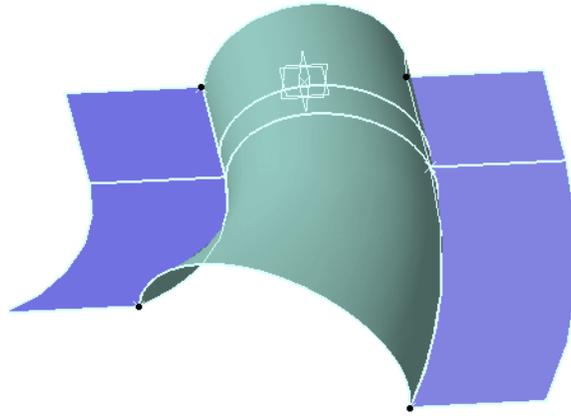
The fill surface is displayed within the boundary.



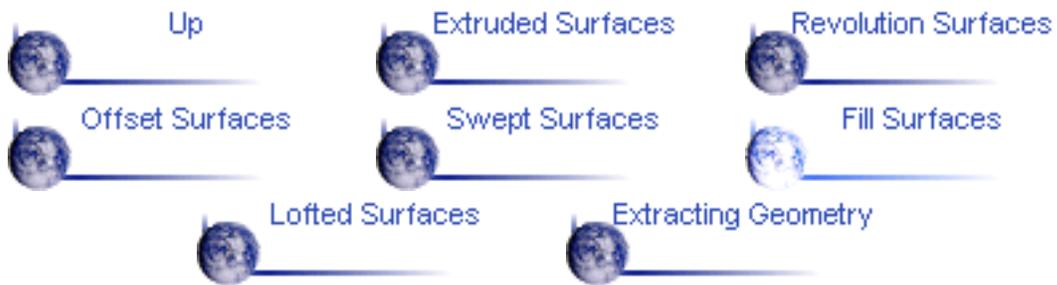
4. You can edit the boundary by first selecting an element in the dialog box list then choosing a button to either:

- Remove the selected element
- Replace the selected element by another curve or support surface
- Add a curve at the end of the list.

5. Click OK to create the fill surface.



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



Creating Lofted Surfaces



This task shows how to create a lofted surface.

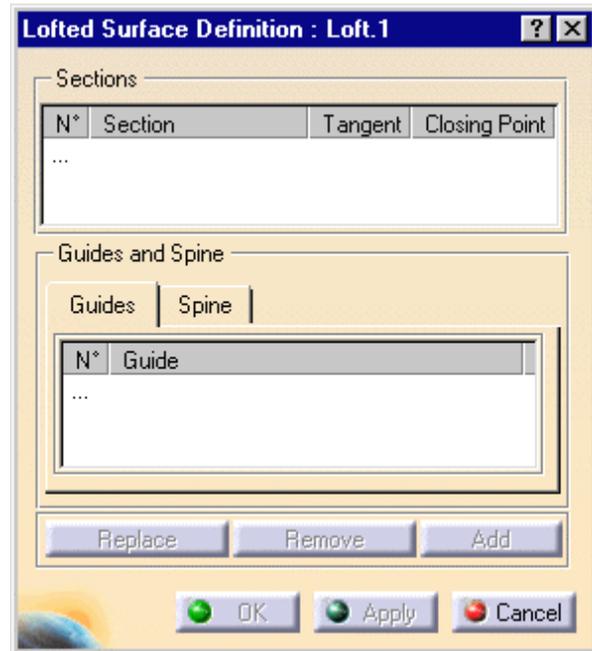
You can generate a lofted surface by sweeping two or more planar section curves along an automatically computed or user-defined spine. The surface can be made to respect one or more guide curves.

Open the Loft.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Loft icon  .

The Lofted Surface Definition dialog box appears.



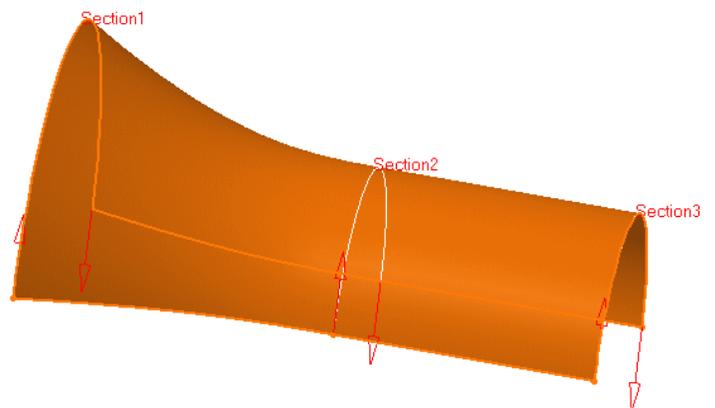
2. Select two or more planar section curves.

The curves must be continuous in point.

You can select tangent surfaces for the start and end section curves.

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

Example of a loft defined by three planar sections:



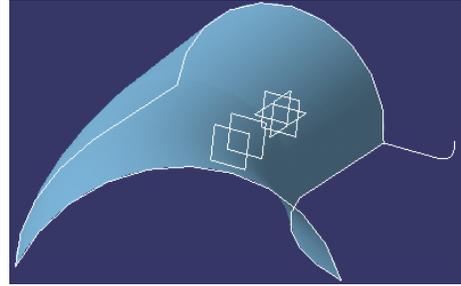
Example of a loft defined by 2 planar sections and 2 guide curves:

3. If needed, select one or more 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 loft if it intersects the first extremity of each section curve.

Similarly, the last guide curve will be a boundary of the loft if it intersects the last extremity of each section curve.



You can make a loft tangent to an adjacent surface by selecting an end section that lies on the adjacent surface.

In Figure 2 a loft tangent to the existing surface has been created:

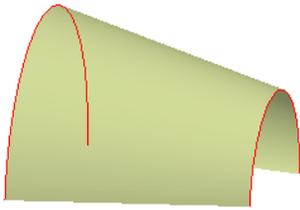


Figure 1

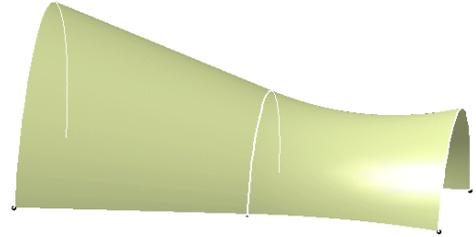


Figure 2

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.

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.

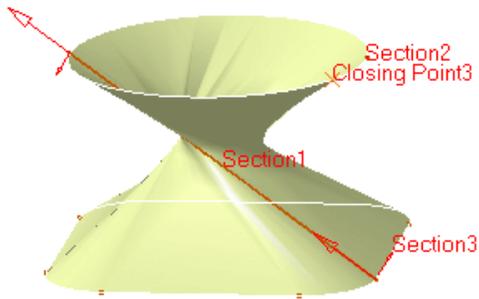
Note that the spine curve must be normal to each section plane and must be continuous in tangency.



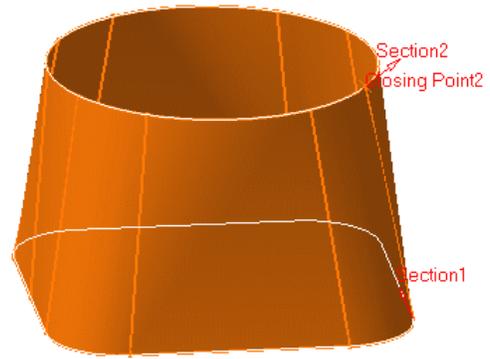
You can create lofted surfaces between closed section curves. These curves have point continuity at their closing point.

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



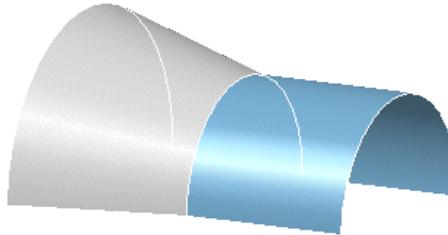
A new closing point has been imposed to get a non-twisted surface

5. It is possible to edit the loft 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 loft and the adjacent surface.



6. Click OK to create the lofted surface.

The surface (identified as Loft.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.





This task presents the two kinds of coupling during the creation of the lofted surface:

- coupling between two consecutive sections
- coupling between guides

These couplings compute the distribution of isoparameters on the surface.

Open the Loftcoupling.CATPart document from the online/Samples/ShapeDesign directory.

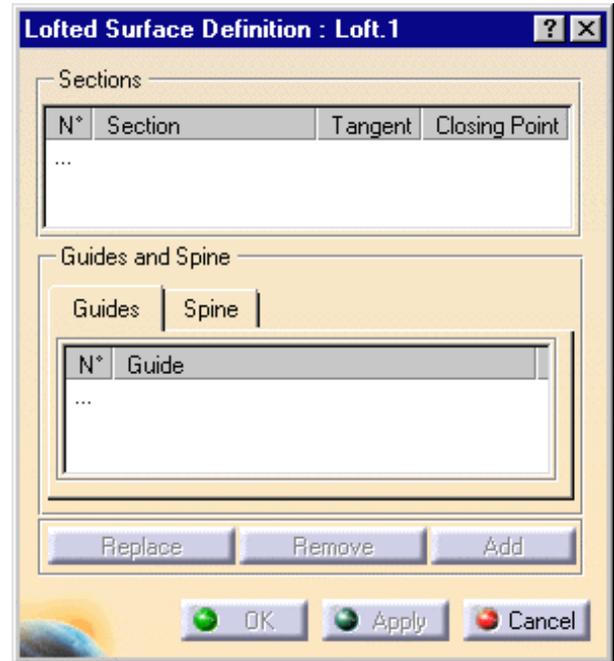
Coupling between two consecutive sections

This coupling is based on the curvilinear abscissa.

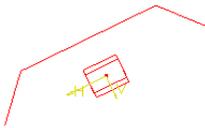


1. Click the Loft icon .

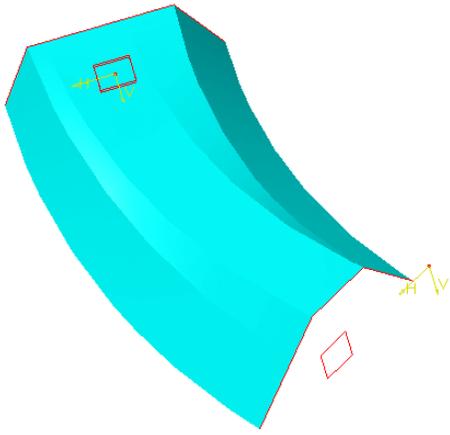
The Lofted Surface Definition dialog box appears.



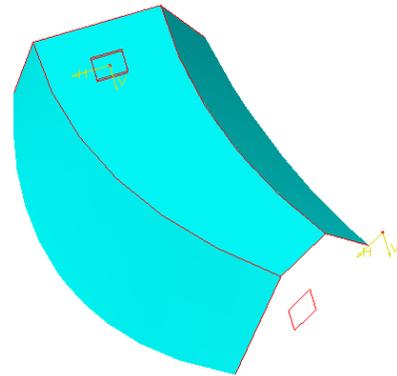
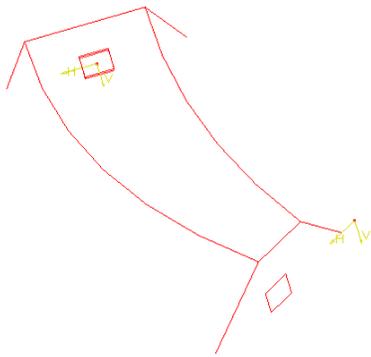
2. Select the two consecutive sections.



3. Click OK to create the loft.



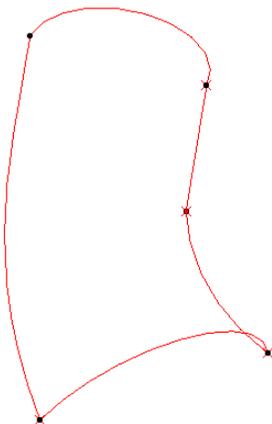
If you want to create a coupling between particular points, you must add guides.

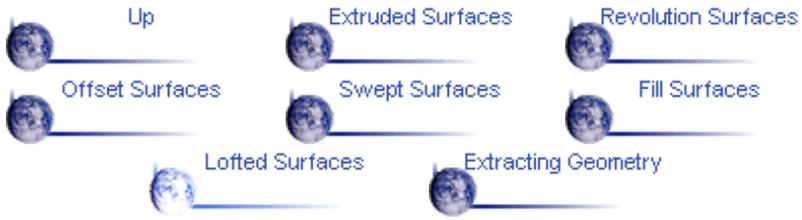
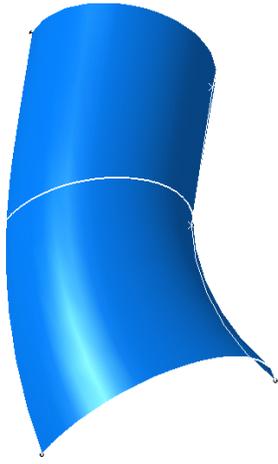


Coupling between guides

This coupling is performed by the spine.

If a guide is the concatenation of several curves, the resulting loft will contain as many surfaces as curves within the guide.



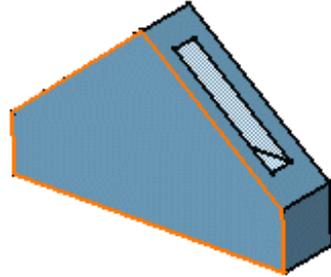


Extracting Geometry

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

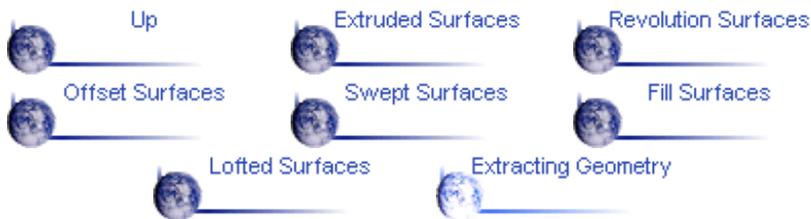
 1. Select an edge or the face of an element.

The selected element is highlighted.



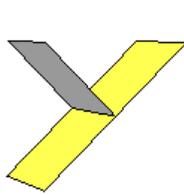
2. Click the Extract icon .

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



Performing Operations on Shape Geometry

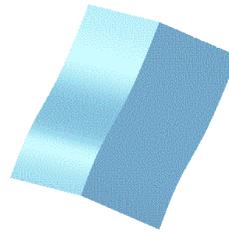
CATIA - Generative Shape Design allows you to modify your design using techniques such as trimming, extrapolating and filleting.



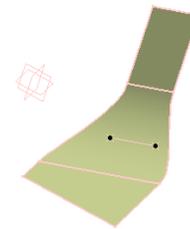
[Splitting Geometry](#)



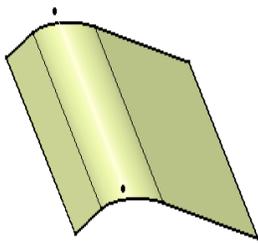
[Trimming Geometry](#)



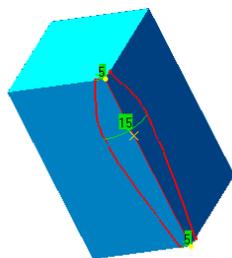
[Joining Geometry](#)



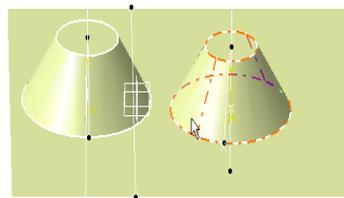
[Shape Fillets](#)



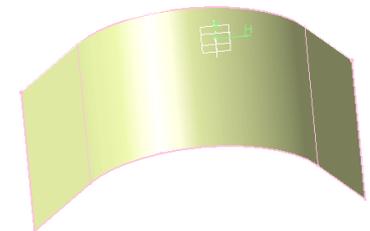
[Edge Fillets](#)



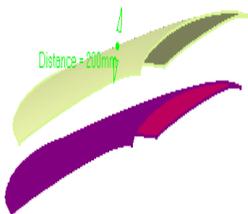
[Variable Radius Fillets](#)



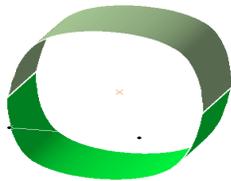
[Face-Face Fillets](#)



[Tritangent Fillets](#)



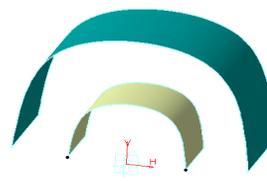
[Translating Geometry](#)



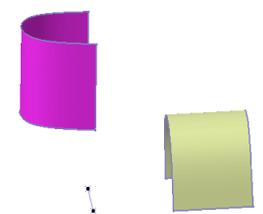
[Performing Symmetry on Geometry](#)



[Transforming Geometry by Scaling](#)



[Transforming Geometry by Affinity](#)



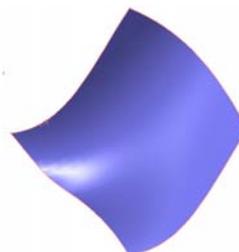
[Rotating Geometry](#)



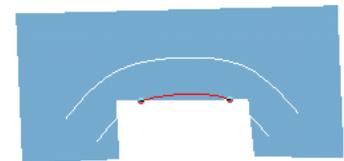
[Extrapolating Surfaces](#)



[Extrapolating Curves](#)



[Inverting the Orientation of Geometry](#)



[Creating Nearest Sub-element](#)

 Up
 Performing Operations

 Creating Wireframe Geometr
 Editing Geometry

 Creating Surfaces
 Using Tools

Splitting Geometry



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

You can split:

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



1. Click the Split icon .

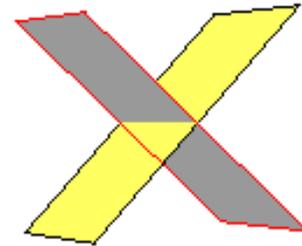
The Split Definition dialog box appears.



2. Select the element to be split.



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

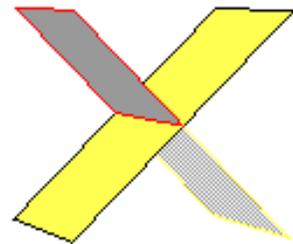


3. Select the cutting element.

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

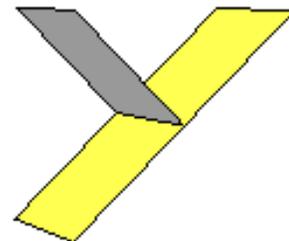
new

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



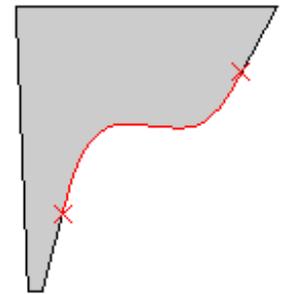
4. Click OK to split the element.

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





When necessary, the cutting element will be extrapolated in order to split a surface correctly (as shown in following figure).



- | | | |
|---|--|---|
|  Up |  Splitting |  Trimming |
|  Joining |  Shape Fillets |  Edge Fillets |
|  Variable Radius Fillets |  Tritangent Fillets |  Face-Face Fillets |
|  Translating |  Symmetry |  Scaling |
|  Affinity |  Rotating |  Extrapolating |
|  Extrapolating Curves |  Inverting |  Nearest Sub-element |

Trimming Geometry

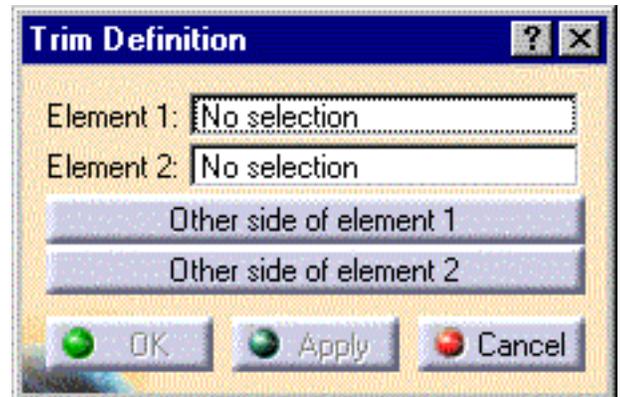


This task shows how to trim two surfaces or two wireframe elements.



1. Click the Trim icon .

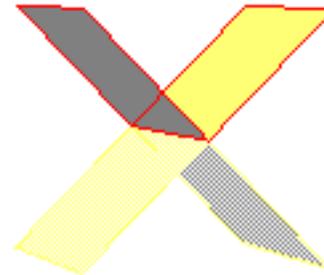
The Trim Definition dialog box appears.



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.

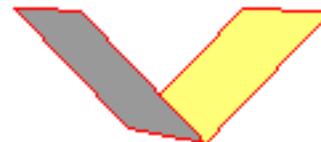


You should make your selections by clicking on the portions that you want to keep after the trim.

When necessary, the cutting elements will be extrapolated in order to trim surfaces correctly.

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

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





Up



Joining



Variable Radius Fillets



Translating



Affinity



Extrapolating Curves



Splitting



Shape Fillets



Tritangent Fillets



Symmetry



Rotating



Inverting



Trimming



Edge Fillets



Face-Face Fillets



Scaling



Extrapolating



Nearest Sub-element

Joining Surfaces or Curves



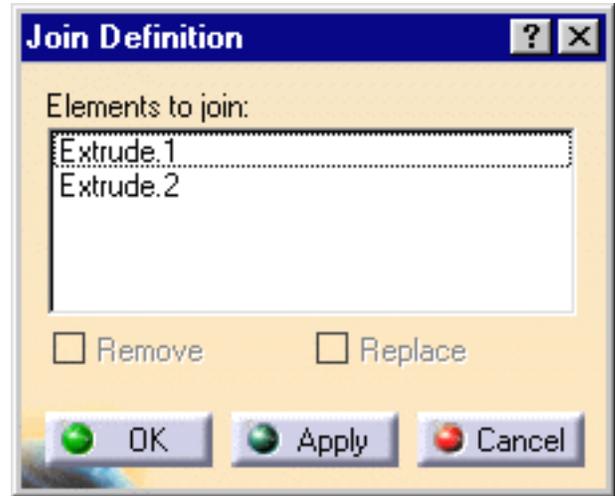
This task shows how to join surfaces or curves.

The surfaces or curves to be joined must be adjacent.



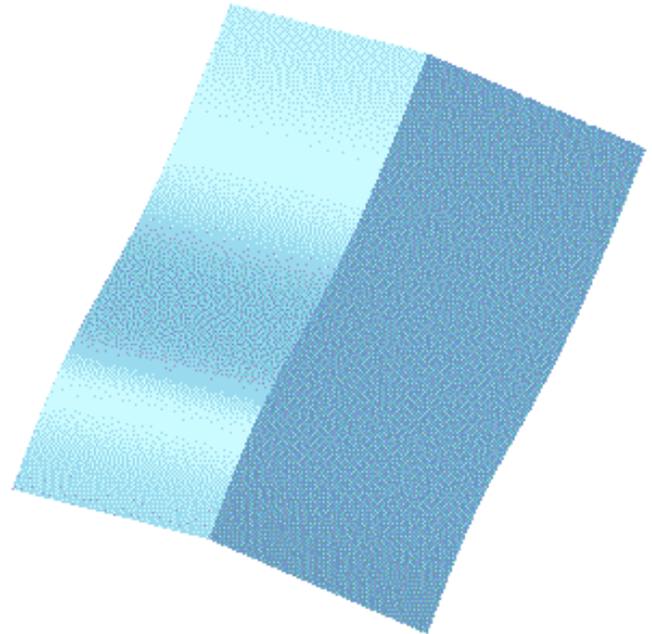
1. Click the Join icon.

The Join Definition dialog box appears.



2. Select the surfaces or curves to be joined.

3. You can edit the list of elements in the definition list by means of the Remove and Replace buttons.



4. Click OK to create the joined surface or curve.

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





Up



Splitting



Trimming



Joining



Shape Fillets



Edge Fillets



Variable Radius Fillets



Tritangent Fillets



Face-Face Fillets



Translating



Symmetry



Scaling



Affinity



Rotating



Extrapolating



Extrapolating Curves



Inverting



Nearest Sub-element

Shape Fillets



This task shows how to create a shape fillet between two other surfaces.

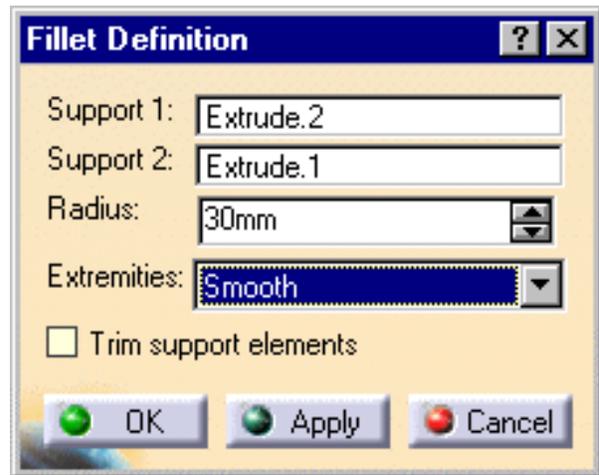


Open the Shape-fillet.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Shape Fillet icon .

The Fillet Definition dialog box appears.



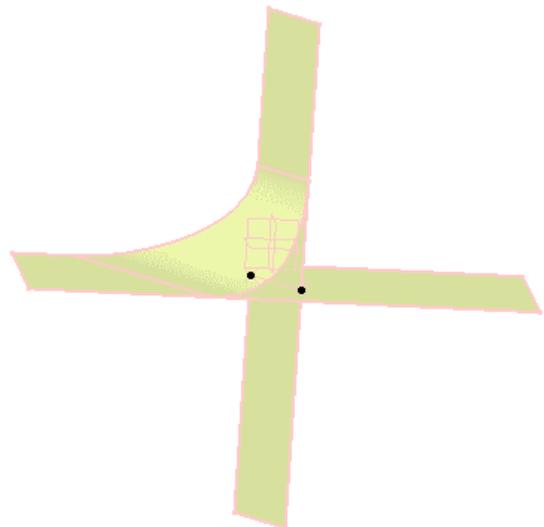
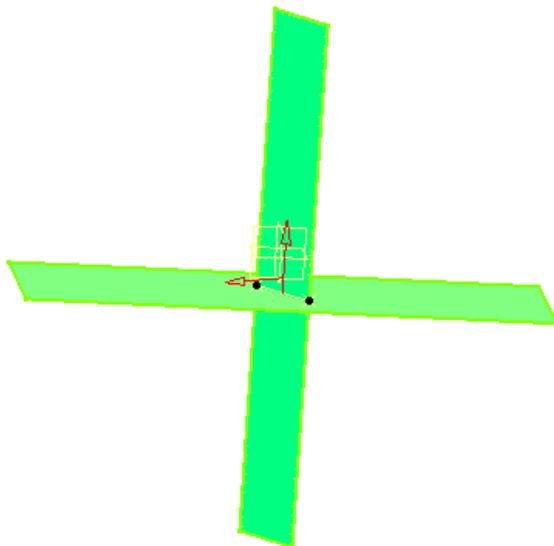
3. Select a surface as the first support element.

4. Select another surface as the second support element.

5. Enter the value of the fillet Radius.

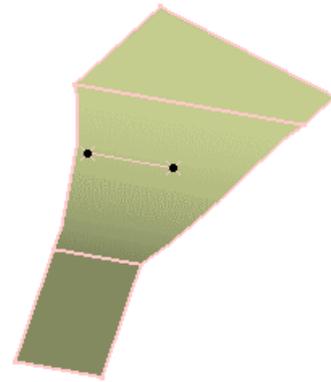
Up to four fillet locations may be possible.

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.

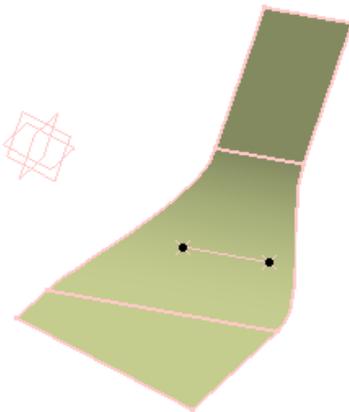


6. Use the combo to choose the desired type of extremity for the fillet:

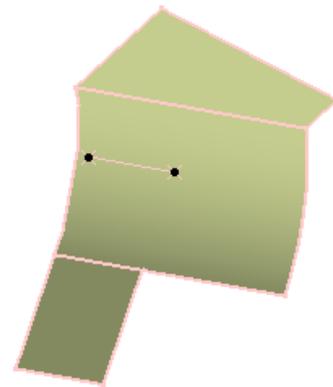
- Straight
- Smooth
- Maximum
- Minimum.



Straight fillet



Smooth fillet



Maximum fillet

7. Click the Trim support elements check box to trim the support elements and assemble them to the fillet.

8. Click OK to create the shape fillet.

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





Up



Joining



Variable Radius Fillets



Translating



Affinity



Extrapolating Curves



Splitting



Shape Fillets



Tritangent Fillets



Symmetry



Rotating



Inverting



Trimming



Edge Fillets



Face-Face Fillets



Scaling



Extrapolating



Nearest Sub-element

Edge Fillets



Edge fillets are useful for providing 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.

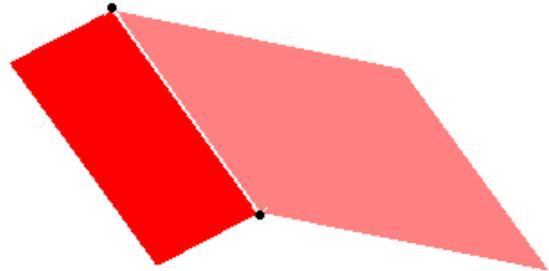


Open the Fillet.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Edge Fillet icon .

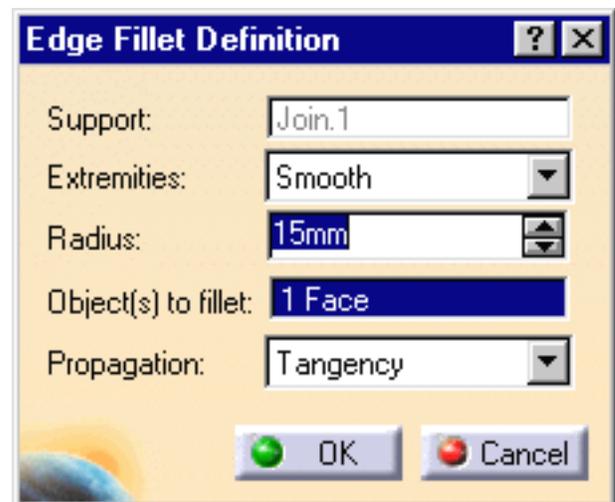
2. Select the joined surface.



The Edge Fillet Definition dialog box appears.

3. Use the combo to select the desired type of extremity for the fillet:

- Smooth
- Straight
- Maximum
- Minimum.

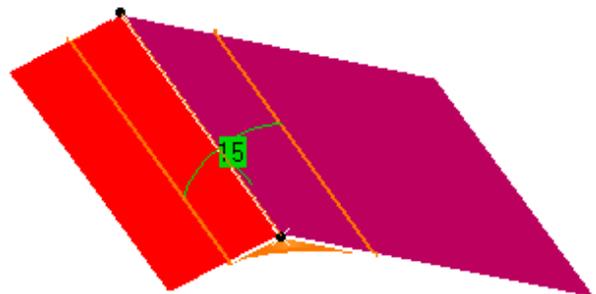


4. Enter the value of the fillet Radius.

A preview of the fillet appears.

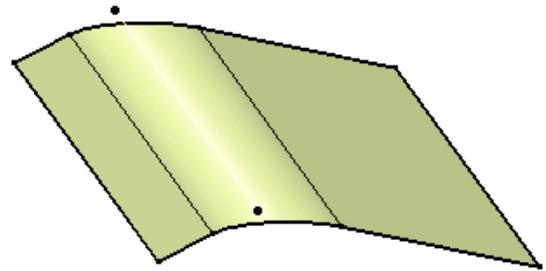
5. You can choose the Propagation type:

- 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.



6. Click OK to create the fillet surface.

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



- | | | |
|---|--|--|
|  Up |  Splitting |  Trimming |
|  Joining |  Shape Fillets |  Edge Fillets |
|  Variable Radius Fillets |  Tritangent Fillets |  Face-Face Fillets |
|  Translating |  Symmetry |  Scaling |
|  Affinity |  Rotating |  Extrapolating |
|  Extrapolating Curves |  Inverting |  Nearest Sub-element |

Variable Radius Fillets

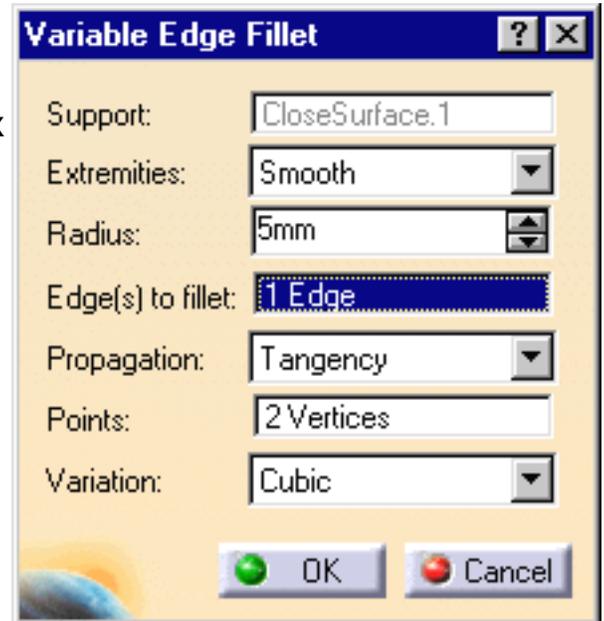


This task shows how to create a variable radius fillet. In this type of fillet the radius varies at selected points along a selected edge.



1. Click the Variable Radius Fillet icon .

The Variable Edge Fillet Definition dialog box appears.

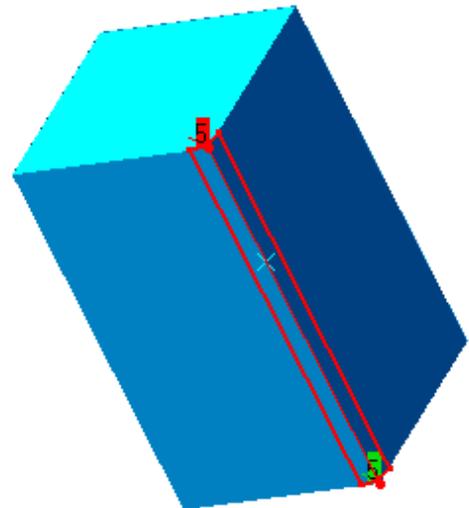


2. Select the edge to be filleted.

CATIA detects the two vertices and displays the default radius value.

3. Use the combo to select the desired type of extremity for the fillet:

- Smooth
- Straight
- Maximum
- Minimum.



- 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.

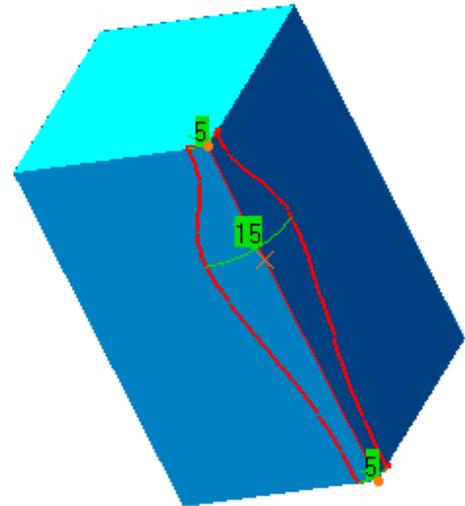
- To add an additional point on the edge to make the variable radius fillet, click the Points field and select a point on the edge.

- Enter a new Radius value for this point.

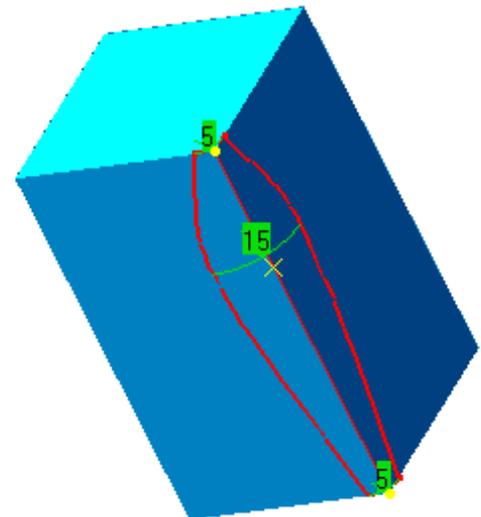
- Set the Propagation mode to Cubic to obtain a smooth transition from one radius to another.

- Click OK to confirm the operation.

The edge is filleted. The specification tree indicates this creation.



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.





Up



Joining



Variable Radius Fillets



Translating



Affinity



Extrapolating Curves



Splitting



Shape Fillets



Tritangent Fillets



Symmetry



Rotating



Inverting



Trimming



Edge Fillets



Face-Face Fillets



Scaling



Extrapolating



Nearest Sub-element

new Tritangent Fillets



This task shows how to create a tritangent fillet.

The creation of tritangent fillets involves the removal of one of the three faces selected.



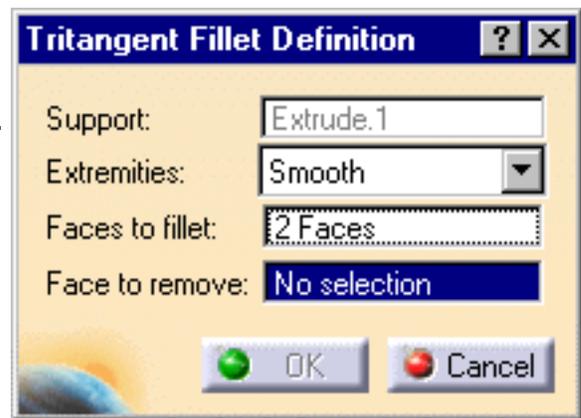
Open the Tritangent.CATPart document from the online/Samples/ShapeDesign directory.



1. Select the support.

2. Click the Tritangent Fillet  icon.

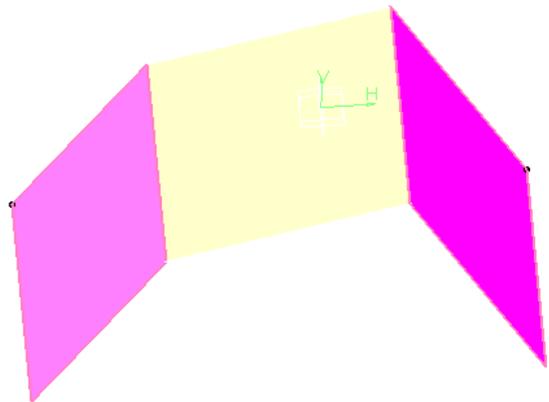
The Tritangent Fillet Definition dialog box appears.



3. Select the Extremities that is the relimitation mode:

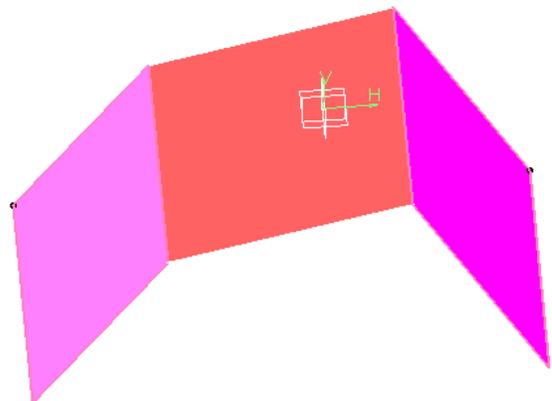
- Smooth
- Straight
- Maximum
- Minimum

4. Select the two Faces to fillet.



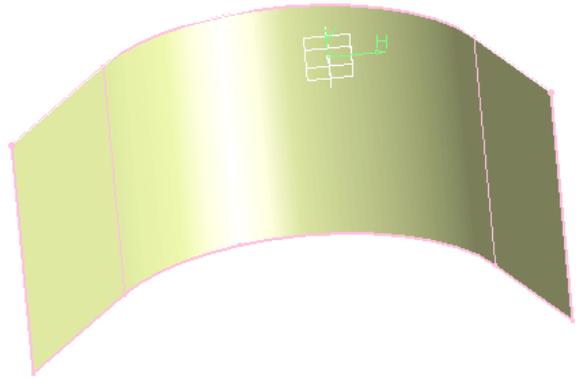
5. Select the Face to remove.

The fillet will be tangent to this face.



6. Click OK.

The faces are filleted. The creation of this fillet is indicated in the specification tree.



new Face-Face Fillets



This task shows how to create a face-face fillet.

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.



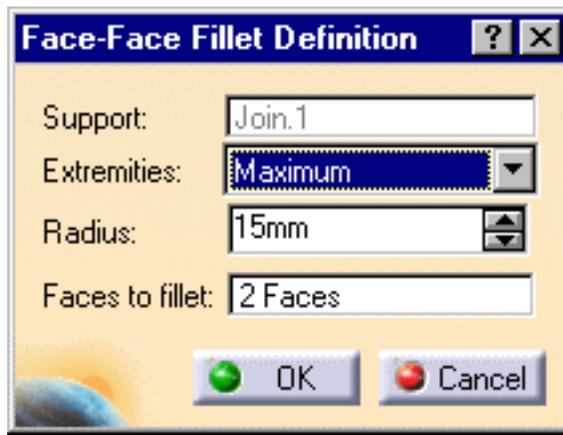
Open the Facefillet.CATPart document from the online/Samples/ShapeDesign directory.



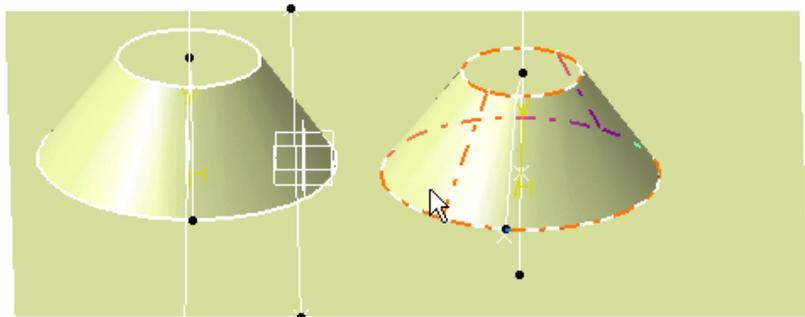
1. Select the support
2. Click the Face-Face Fillet

icon 

The Face-Face Fillet Definition dialog box appears.

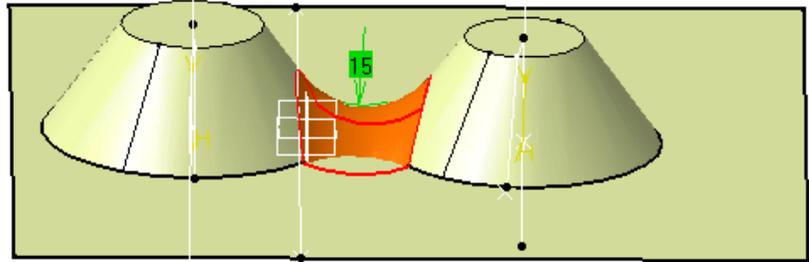


3. Select the Extremities that is the relimitation mode.
4. Select the two Faces to fillet.

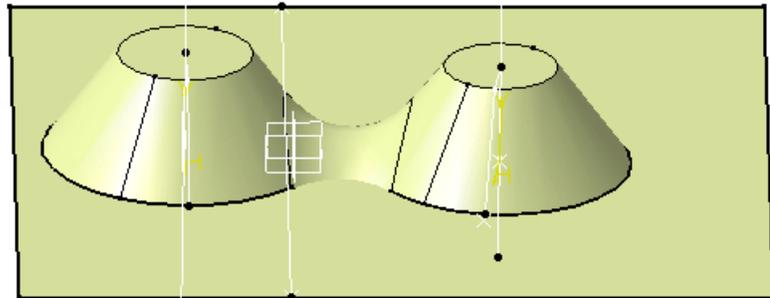


The application previews the fillet to be created.

5. Enter a radius value in the Radius field if you are not satisfied with the default one. This value must be greater than 0.
6. Click OK.



The faces are filleted. This fillet is indicated in the specification tree.



- Up
- Joining
- Variable Radius Fillets
- Translating
- Affinity
- Extrapolating Curves

- Splitting
- Shape Fillets
- Tritangent Fillets
- Symmetry
- Rotating
- Inverting

- Trimming
- Edge Fillets
- Face-Face Fillets
- Scaling
- Extrapolating
- Nearest Sub-element

Translating Geometry



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

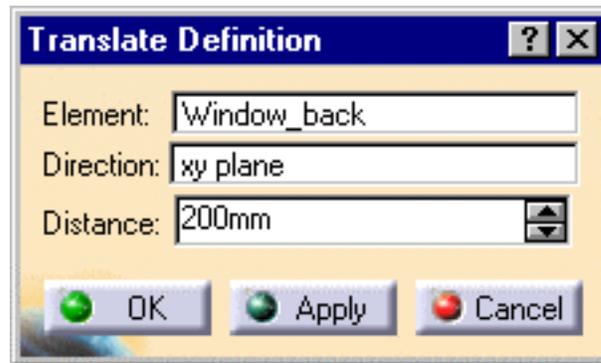


Open the Translate.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Translate icon

The Translate Definition dialog box appears.



2. Select the Element to be translated.

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

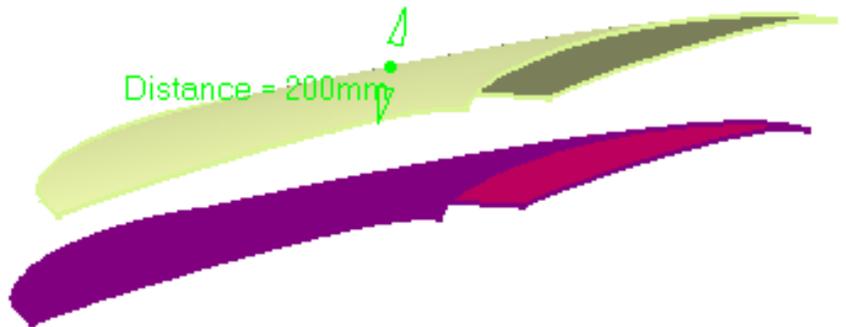
You can also specify the direction by means of X, Y, Z vector components by using the contextual menu on the Direction field.

4. Specify the translation Distance by entering a value or using the drag manipulator.

5. Click OK to create the translated element.

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

The original element is unchanged.





Up



Joining



Variable Radius Fillets



Translating



Affinity



Extrapolating Curves



Splitting



Shape Fillets



Tritangent Fillets



Symmetry



Rotating



Inverting



Trimming



Edge Fillets



Face-Face Fillets



Scaling



Extrapolating



Nearest Sub-element

Performing a Symmetry on Geometry



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

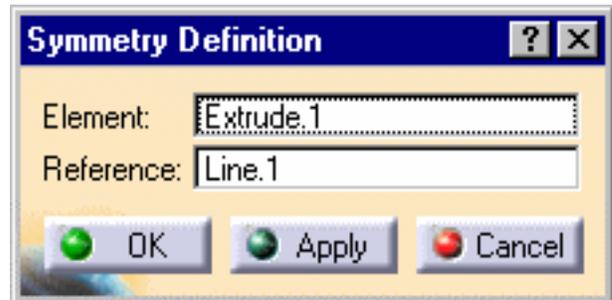


Open the Transform.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Symmetry icon .

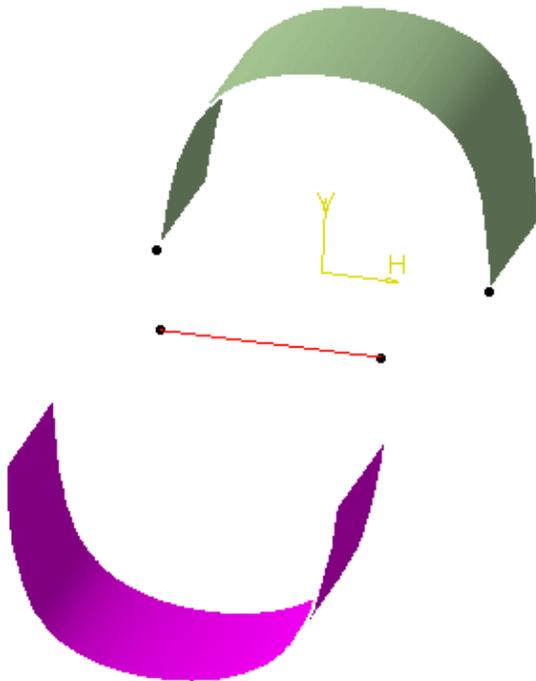
The Symmetry Definition dialog box appears.



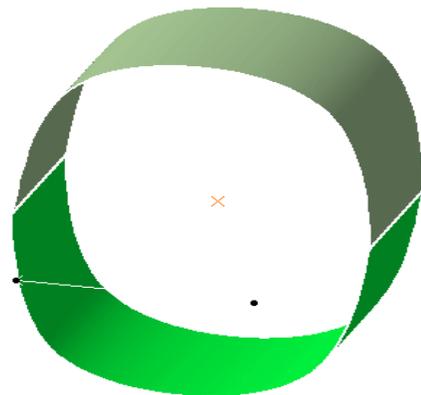
2. Select the Element to be transformed by symmetry.

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.



The original element is unchanged.



Up



Joining



Variable Radius Fillets



Translating



Affinity



Extrapolating Curves



Splitting



Shape Fillets



Tritangent Fillets



Symmetry



Rotating



Inverting



Trimming



Edge Fillets



Face-Face Fillets



Scaling



Extrapolating



Nearest Sub-element

Transforming Geometry by Scaling



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

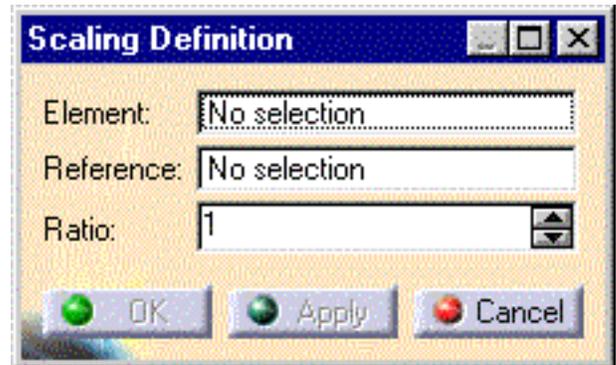


Open the Transform.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Scaling icon .

The Scaling Definition dialog box appears.

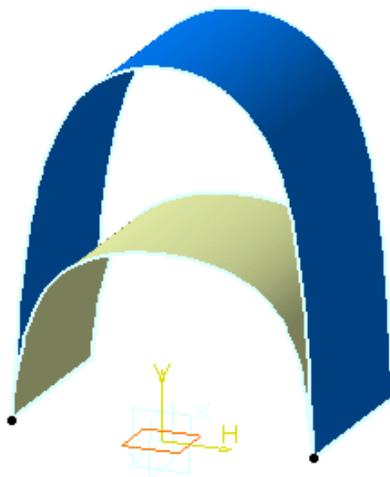


2. Select the Element to be transformed by scaling.

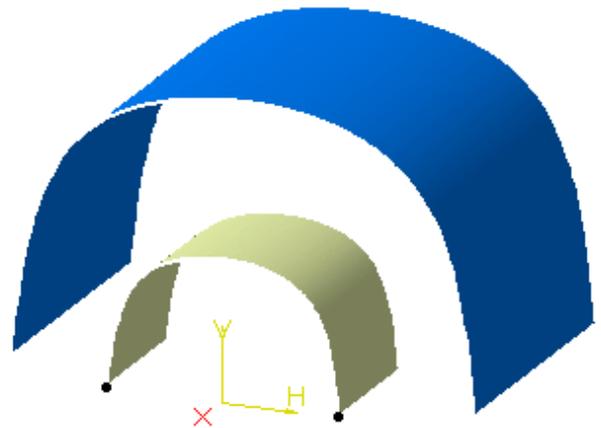
3. Select the scaling Reference point, plane or planar surface.

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.

The original element is unchanged.





Up



Joining



Variable Radius Fillets



Translating



Affinity



Extrapolating Curves



Splitting



Shape Fillets



Tritangent Fillets



Symmetry



Rotating



Inverting



Trimming



Edge Fillets



Face-Face Fillets



Scaling



Extrapolating



Nearest Sub-element

Transforming Geometry by Affinity



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

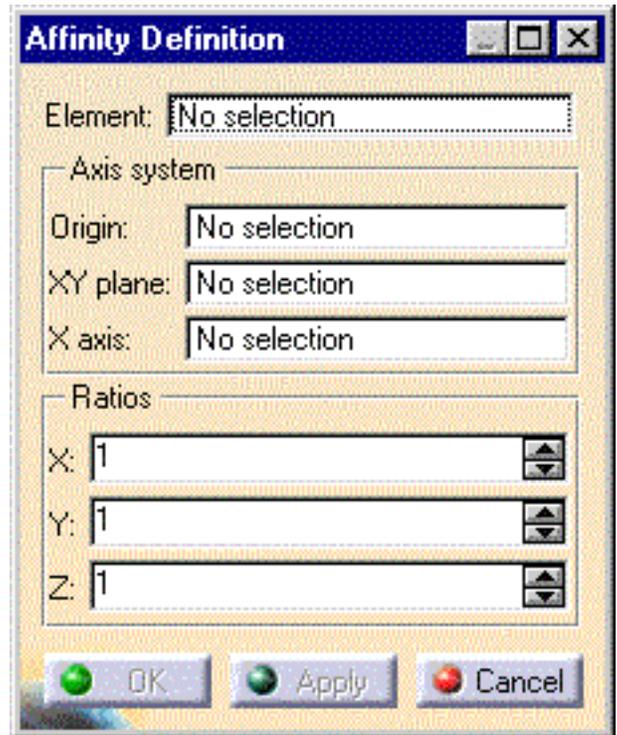


Open the Transform.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Affinity icon .

The Affinity Definition dialog box appears.



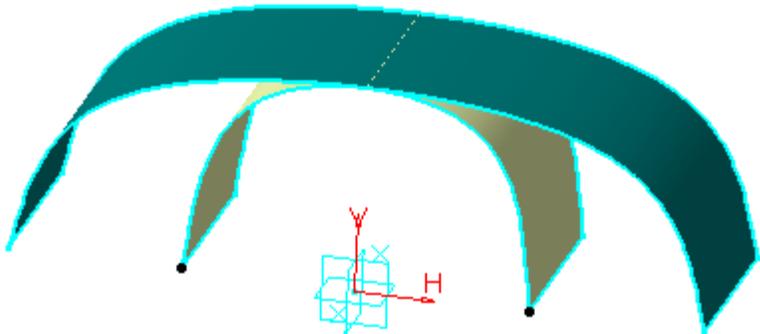
2. Select the Element to be transformed by affinity.

3. Specify the characteristics of the Axis system to be used for the affinity operation:

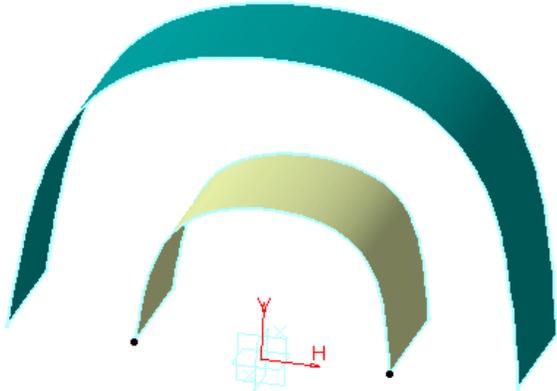
- the Origin
- the XY plane
- the X axis.

4. Specify the affinity Ratios by entering the desired X, Y, Z values.

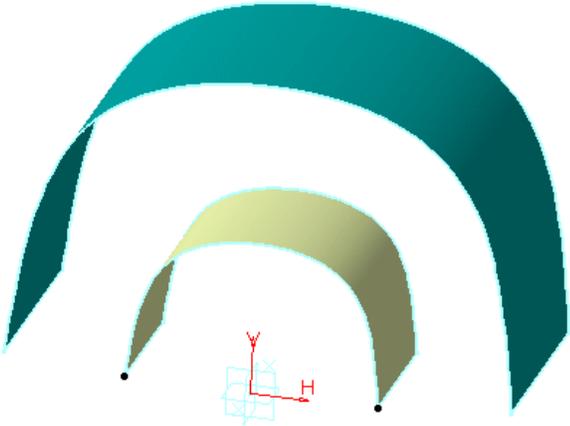
The figure below illustrates the resulting affinity with ratios $X = 2$, $Y = 1$ and $Z = 1$.



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



The figure below illustrates the resulting affinity with ratios $X = 2$, $Y = 2$ and $Z = 1.5$



5. Click OK to create the affinity element.

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

 The original element is unchanged.



Rotating Geometry



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

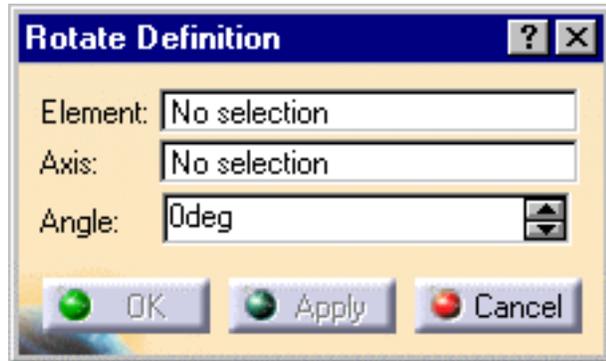


Open the Transform.CATPart document from the online/Samples/ShapeDesign directory.



1. Click the Rotate icon .

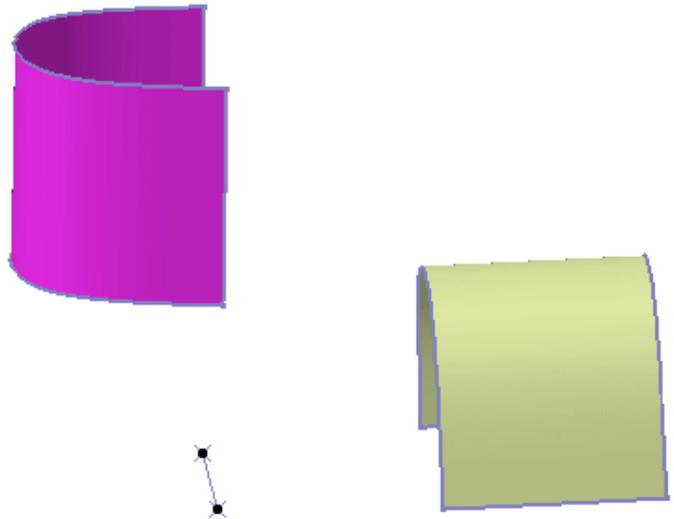
The Rotate Definition dialog box appears.



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.

The original surface is unchanged.





Up



Joining



Variable Radius Fillets



Translating



Affinity



Extrapolating Curves



Splitting



Shape Fillets



Tritangent Fillets



Symmetry



Rotating



Inverting



Trimming



Edge Fillets



Face-Face Fillets



Scaling



Extrapolating



Nearest Sub-element

Extrapolating Surfaces



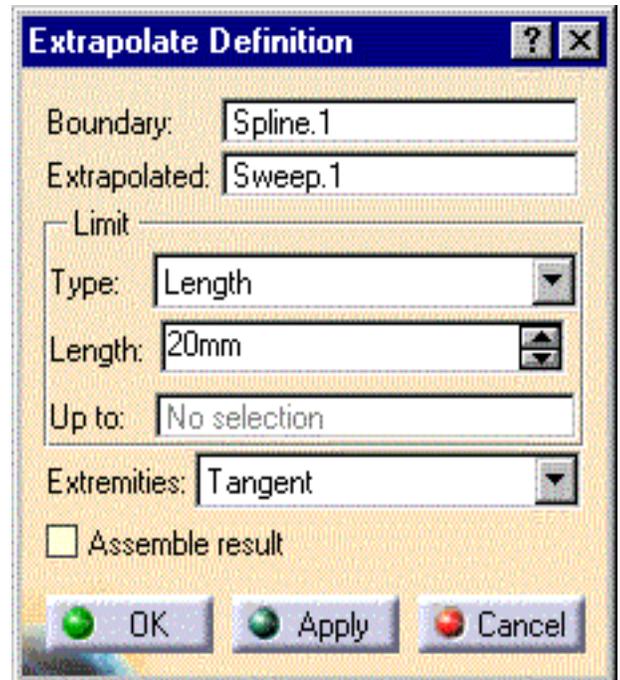
This task shows you how to extrapolate a surface boundary .



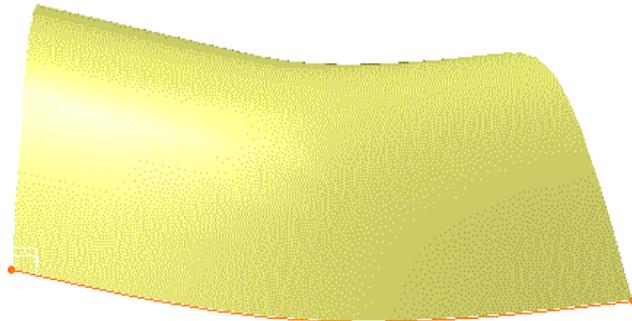
1. Click the Extrapolate icon



The Extrapolate Definition dialog box appears.



2. Select a surface Boundary.



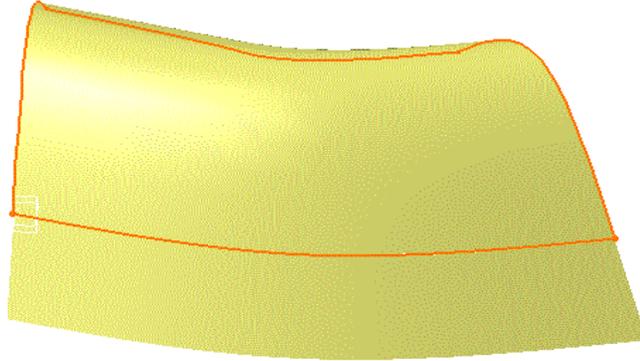
3. Select the surface to be Extrapolated.

4. Specify the Limit of the extrapolation by either:

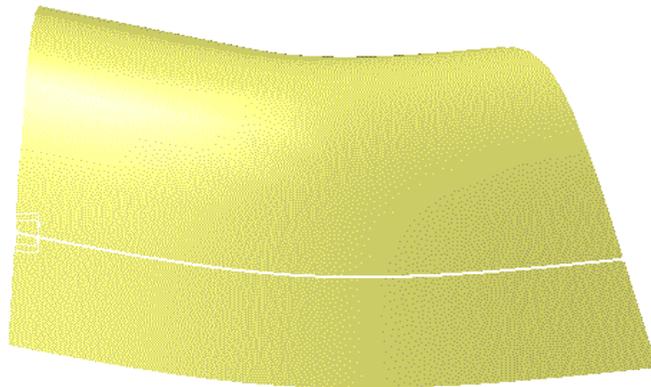
- entering the value of the extrapolation length
- selecting a limit surface or plane.

5. Specify Extremities conditions:

- 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.



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



6. Click OK to create the extrapolated surface.

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





Up



Joining



Variable Radius Fillets



Translating



Affinity



Extrapolating Curves



Splitting



Shape Fillets



Tritangent Fillets



Symmetry



Rotating



Inverting



Trimming



Edge Fillets



Face-Face Fillets



Scaling



Extrapolating



Nearest Sub-element

new Extrapolating Curves

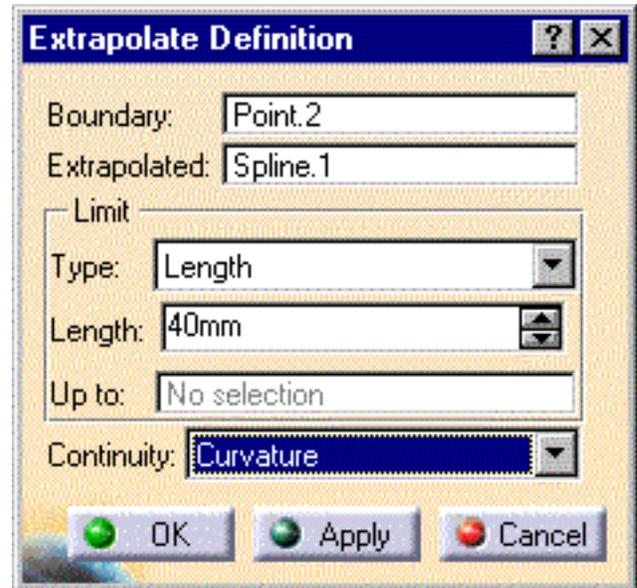


This task shows you how to extrapolate a curve.



1. Click the Extrapolate icon .

The Extrapolate Definition dialog box appears.



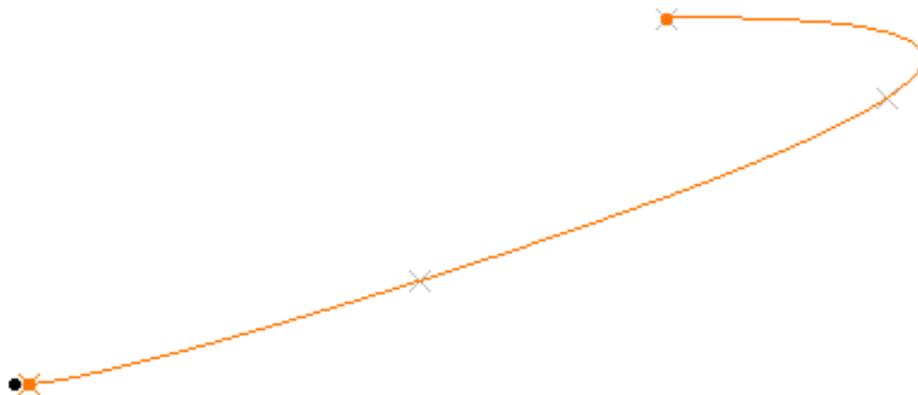
2. Select a point or a curve.

3. Select the curve to be Extrapolated:

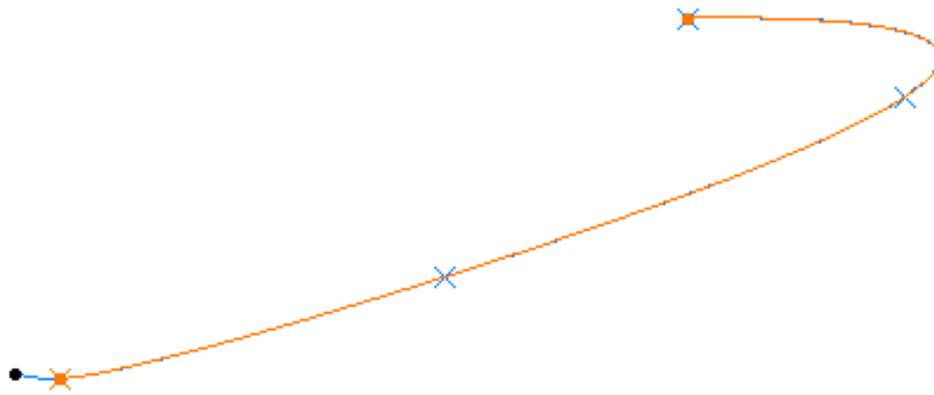
- entering the value of the extrapolation length
- selecting a limit surface or plane.

4. Specify Continuity conditions:

- Tangent: the extrapolation sides are tangent to the edges adjacent to the surface boundary.
- Curvature



5. Click OK to create the extrapolated curve.



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



Inverting the Orientation of Geometry

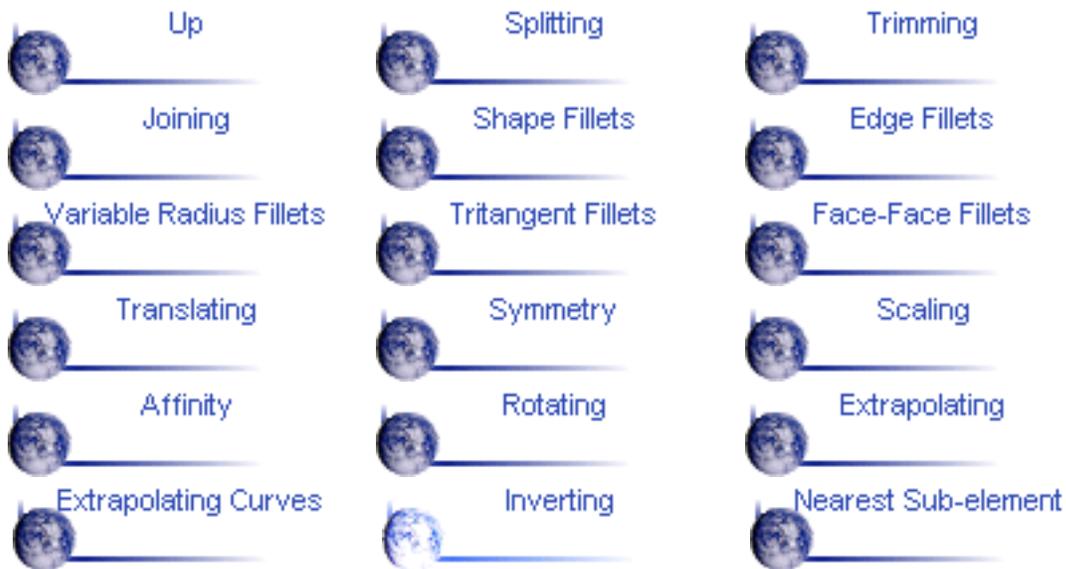


This task shows you how to easily invert the orientation of a surface or curve.



1. Select the Insert > Operations > Invert Orientation command.
2. Select the surface or curve whose orientation is to be inverted. An arrow is displayed indicating the orientation of the element.
3. Click the arrow to invert the orientation of the element.
4. Click Invert Orientation again to accept the inverted element.

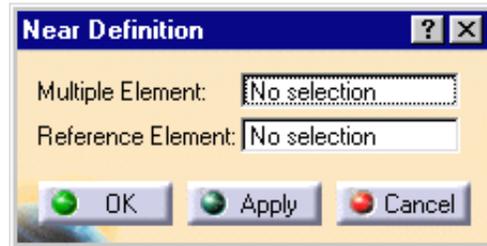
The element (identified as Inverse.xxx) is added to the specification tree.



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.

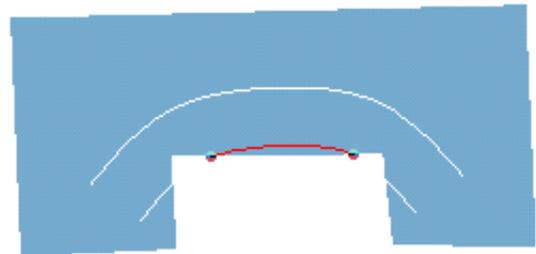
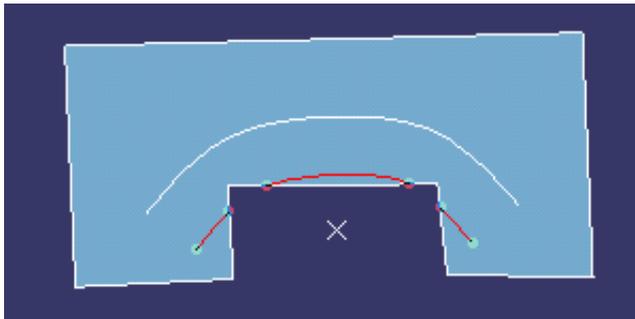
1. Select the Insert > Operations > Near command.
The Near Definition dialog box appears.



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

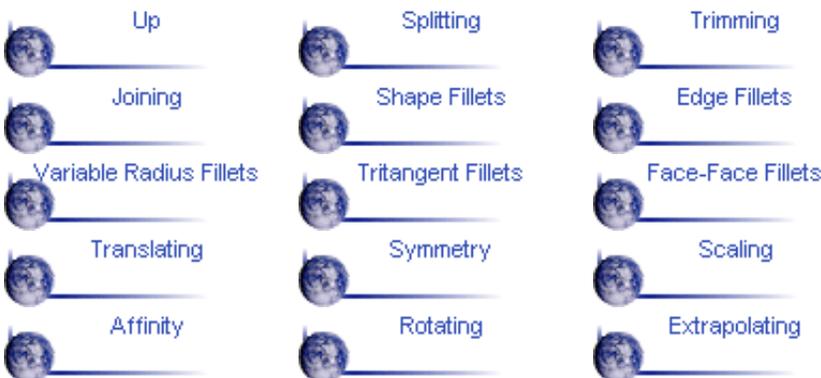
This example shows a parallel curve comprising three sub-elements.

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



4. Click OK to create the element.

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



 Extrapolating Curves

 Inverting

 Nearest Sub-element

Editing Surfaces and Wireframe Geometry

CATIA - Generative Shape Design provides powerful tools for editing surfaces and wireframe geometry.

Editing Definitions



Copying and Pasting



Deleting Geometry



Editing Surface and Wireframe Definitions



This task shows how to edit the definition of an already created geometric element.



1. Activate the Definition dialog box of the element that you want to edit in one of the following ways:

- Select the element then choose the xxx.object -> Definition command from the contextual menu
- Select the element then choose the Edit -> xxx.object -> Definition command
- Double-click 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.



Editing Definitions

Copying and Pasting

Deleting Geometry



Copying and Pasting



This task shows how to copy and paste open body entities in your part design.



1. Select the elements that you want to copy either directly in the part geometry or in the specification tree.
2. Select the Edit -> Copy command.
3. Click the Open Body entity in the tree where you want to paste the selected elements.
4. Select the Edit -> Paste command.

The elements are copied into the target Open Body.



The identifiers of copied elements are incremented with respect to the original elements.

The original elements and copied elements can be edited independently.



Deleting Surfaces and Wireframe Geometry



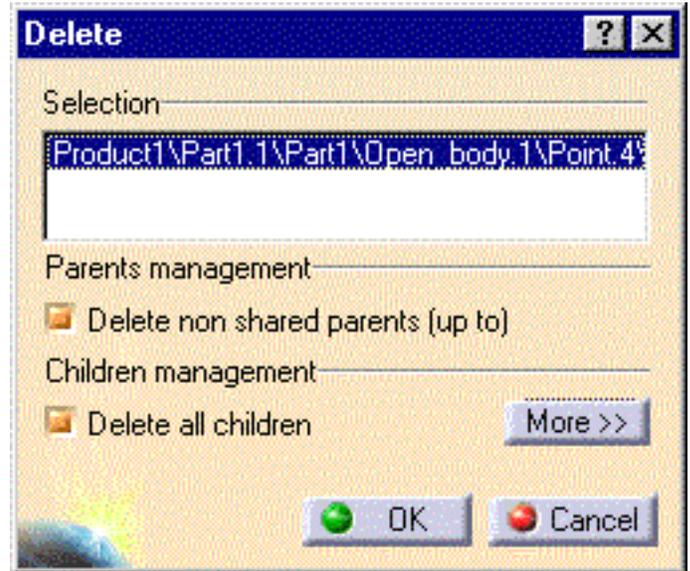
This task shows how to delete geometry from your design.



1. Select the entity you want to delete.

2. Select the Delete command either from the 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

4. Click OK to validate the deletion.



Using Tools for Shape Design

CATIA - Generative Shape Design provides powerful tools to help you manage and analyse your surfaces and wireframe geometry.



[Updating Your Design](#)



[Using the Historical Graph](#)



[Working with a Support](#)



[Creating Datums](#)



[Creating Constraints](#)



[Managing Groups](#)



[Performing a Draft Analysis](#)



[Performing a Mapping Analysis](#)



[Checking Connections between Elements](#)



Up

Performing Operations



Creating Wireframe Geometr

Editing Geometry



Creating Surfaces

Using Tools

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.



1. To update the part, click the Update icon .

However, 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.



Controlling your update is possible: just select the Tools -> Options command and uncheck the automatic update option to make sure you will update your part only when you wish to do so.

2. To update the feature of your choice, just select that feature and use the Update contextual command.



 Up	 Updating Your Design	 Using Historical Graph
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Using the Historical Graph



This task shows how to use the Historical Graph.

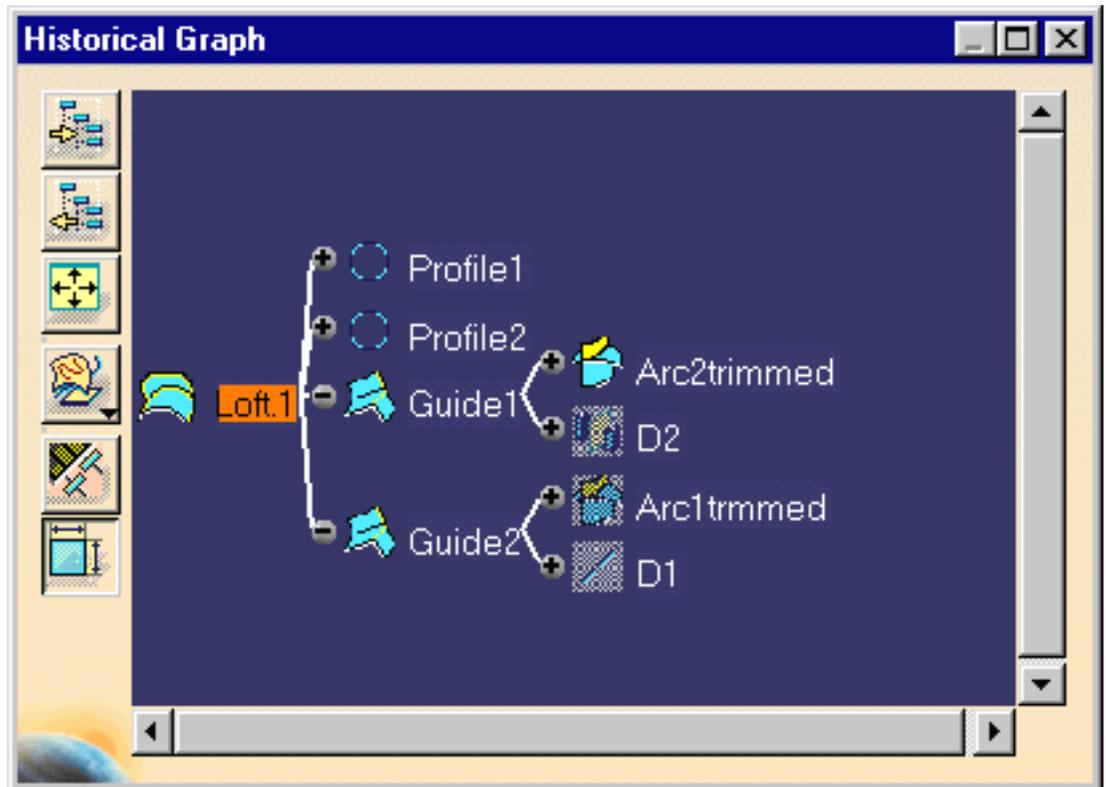


1. Select the element for which you want to display the historical graph.

2. Click the Show 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.





Up



Working with a Support



Managing Groups



Updating Your Design



Creating Datums



Checking Connections betw



Performing a Mapping Analy



Using Historical Graph



Creating Constraints



Performing a Draft Analysis

Working with a Support



This task shows how to work on a support, which may be either a plane or a surface.

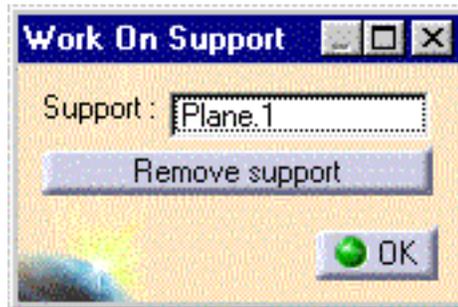
This will allow you to easily reference a surface or plane whenever you need one. For example, you will not have to explicitly select the support element again when creating geometry.



1. Click the Work on Support

icon .

The Work on Support dialog box appears.



2. Select the plane or surface to be used as support element.

If a plane is selected, a grid is displayed to facilitate visualization.



When you no longer need the support, just click on the icon again then click the Remove Support button.



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Creating Datums



This task shows how to create geometry with the History mode deactivated. In this case, when you create an element, there are no links to the other entities that were used to create that element.



1. Click the Create Datum icon  to deactivate the History mode.

It will remain deactivated until you click on the icon again.



If you double-click this icon, the Datum mode is permanent. You only have to click again the icon to deactivate the mode.

A click on the icon activates the Datum mode for the current or the next command.



The History mode (active or inactive) will remain fixed from one session to another: it is in fact a setting.



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Creating Constraints

This task shows how to set geometric constraints on geometric elements.



Such a constraint forces a limitation. For example, a geometric constraint might require that two lines be parallel.



To set a constraint between elements:

1. Multi-select two or three elements to be constrained.

2. Click the Constraint with dialog box icon

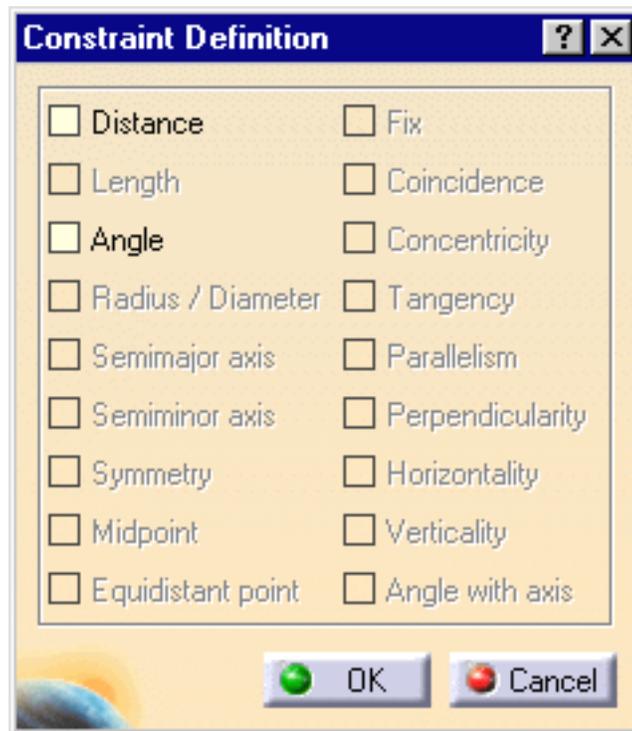


The Constraint Definition dialog box appears indicating the types of constraint you can set between the selected elements.

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.

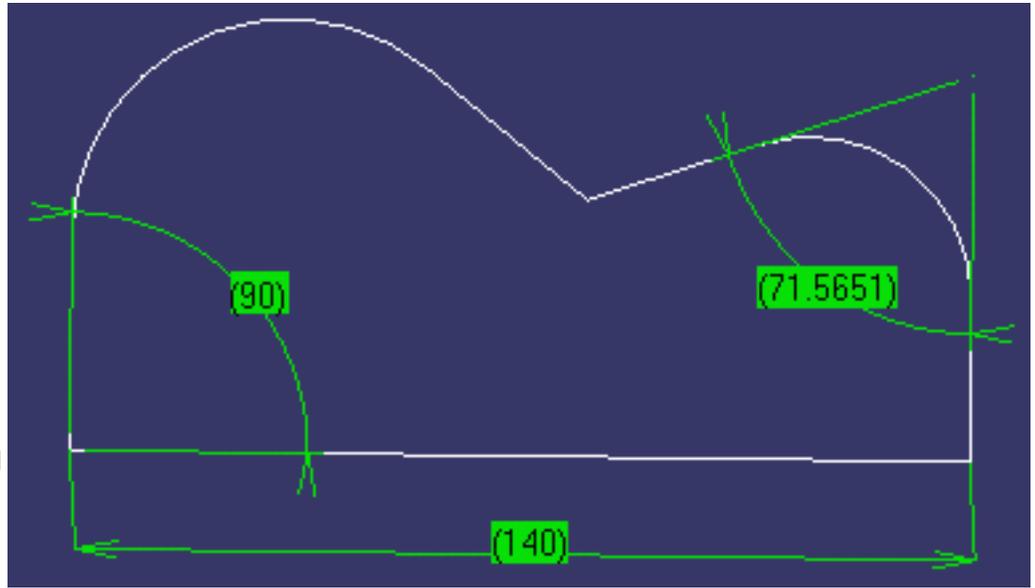


To set a constraint on a single element:

1. Select the element to be constrained.
2. Click the Constraint icon



The corresponding constraint symbol appears on the geometry.



- Up
- Working with a Support
- Managing Groups

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Managing Groups



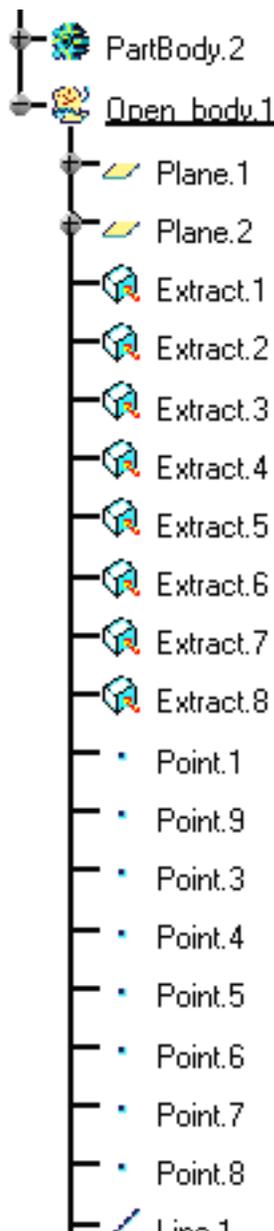
This task shows how to manage groups of elements in an Open Body entity as follows:

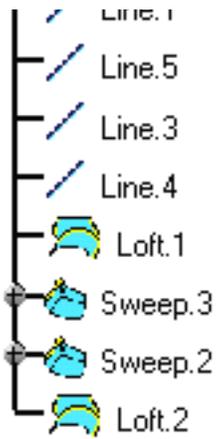
- [creating a group](#)
- [editing a group](#)
- [collapsing and expanding a group](#)
- [moving a group to a new body.](#)



Creating a group

1. Right-click the desired Open Body entity in the specification tree.
2. Choose the Create Group command from the contextual menu.





The Group dialog box appears.

The Support area indicates the name of the Open Body 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.
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 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 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 Expand Group command from the contextual menu.

All the entities belonging to the group are then visible in the specification tree.

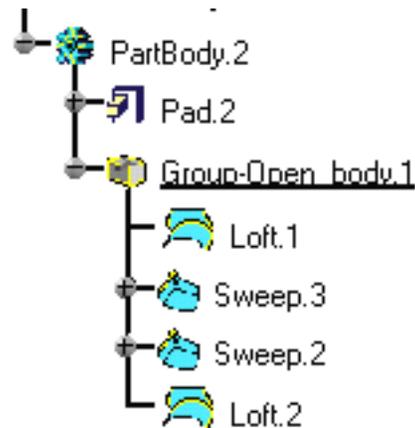
Moving a group to a new body

1. Right-click the desired group in the specification tree and select the Change Body command from the contextual menu.

A dialog box entitled Change Body appears.

2. Select the new body where the group is to be located.

3. Click OK to move the group to the new body.



new Checking Connections between Elements



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:

1. Distance: the values are expressed in millimeters
2. Tangency: the values are expressed in degrees
3. Curvature: the values are expressed in percentage.

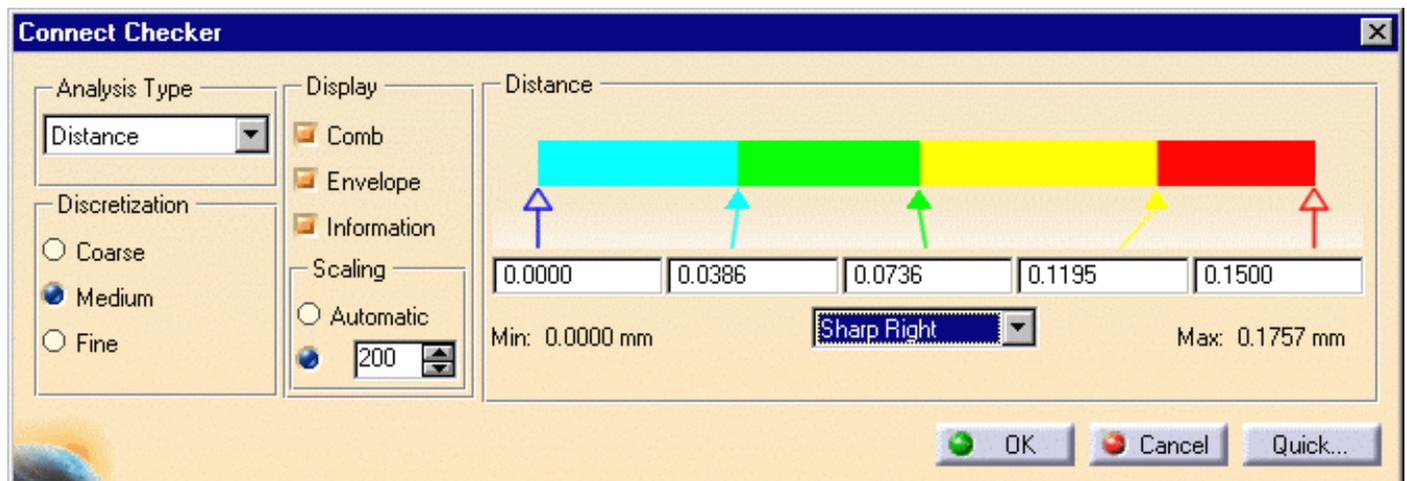
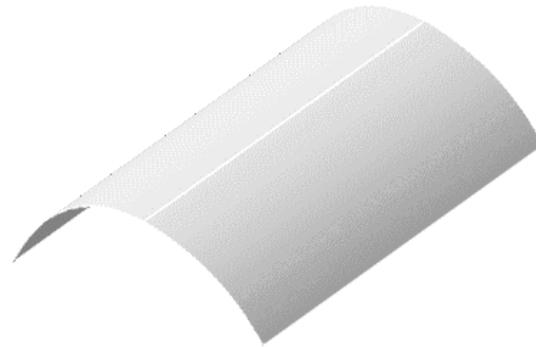


Open the Connectchecker.CATPart document from the online/Samples/ShapeDesign directory.

1. Select both surfaces to be analyzed.
2. Click the Connect Checker icon .

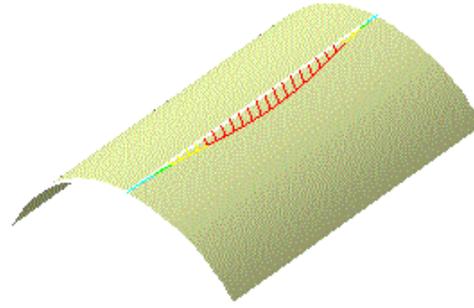
The Connect Checker dialog box is displayed, identifying by color ranges the maximum and minimum values for the analysis. The color range can be linear, sharp left, sharp right or center.

The analysis representation in the color range varies according to the chosen option.



3. Choose the type of analysis to be performed: Distance, Tangency or Curvature.
4. Check the analysis results on the geometry.

Here you are analyzing the distance between the surfaces. Each color section indicates on the geometry the distance between the surfaces.



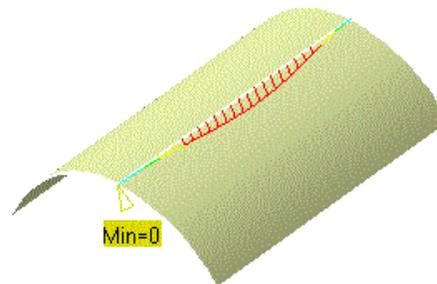
From the Connect Checker dialog box, you can choose a number of visualization and computation options:

- the comb: that is the spikes corresponding to the distance in each point.
- the envelope: that is the curve connecting all spikes together
- some information: the minimum and maximum values

Finally, the scaling option lets you define the visualization of the comb. In automatic mode, the comb size is zoom-independent and always visible on the screen, otherwise you can define a coefficient multiplying the comb exact value.

5. Check the Information button:

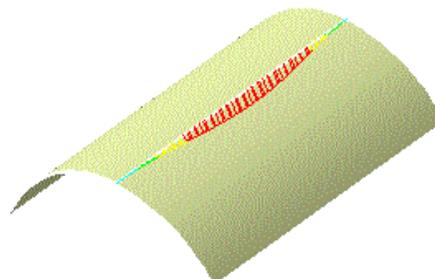
Two texts are displayed on the geometry localizing the minimum and maximum values of the analysis as given in the Connect Checker dialog box.



You can also choose the discretization, that is the numbers of spikes in the comb:

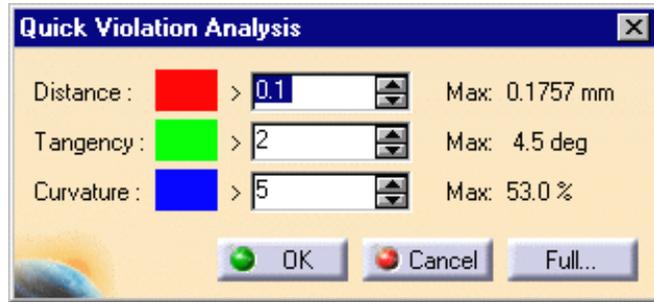
- Coarse: 15 spikes are displayed
- Medium: 30 spikes are displayed
- Fine: 45 spikes are displayed.

6. Switch to Linear mode and check the Fine discretization then compare with the previous results.



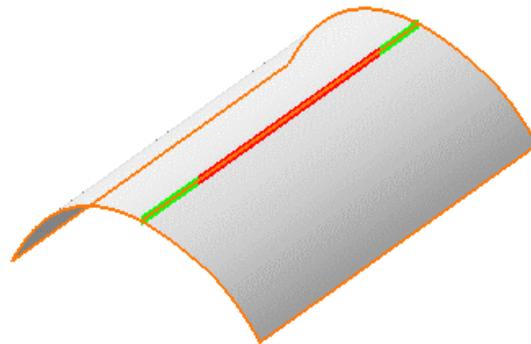
- Click the Quick... button in the Connect Checker dialog box to obtain a simplified analysis taking into account tolerances (distance, tangency and curvature).

The comb is no longer displayed. The Connect Checker dialog box changes to the Quick Violation Analysis dialog box :



- Use the spinners to define the tolerances.

For example, the red area indicates all points that are distant of more than 0.1 mm. The green area indicates points that are closer than 0.1 mm but at which the tangency difference is greater than 2 degrees. Points that would match the first two tolerance constraints and at which the curvature difference would be greater than 5% would appear in blue. There are none in this case.



The maximum deviation values on the current geometry are displayed to the right of the dialog box.

- Click OK to create the analysis as an element in the specification tree.

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 on Connection Analysis in the specification tree then choose Delete.



new Performing a Draft Analysis



This task shows how to analyze the draft angle on a surface.



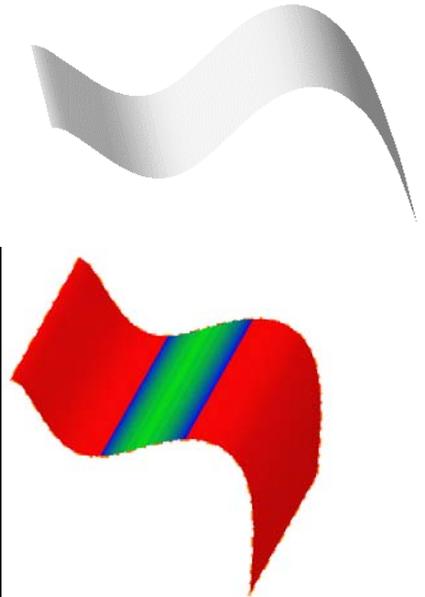
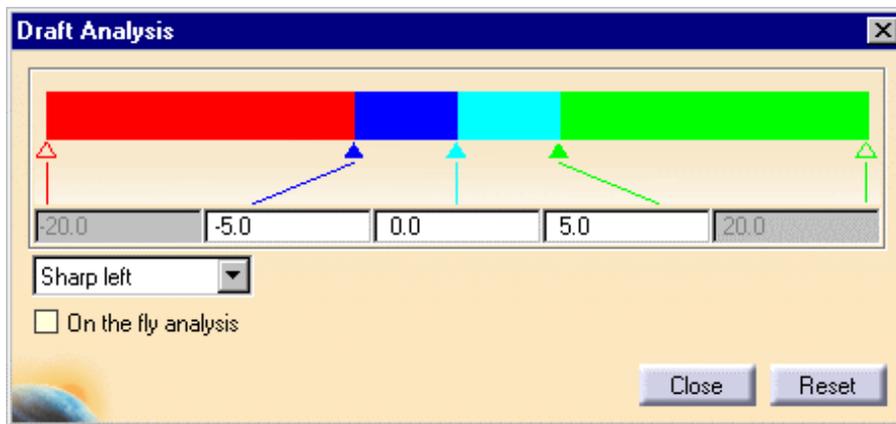
Open the Draftanalysis.CATPart document from the online/Samples/ShapeDesign directory.



1. Select a surface.

2. Click the Draft Analysis icon .

The Draft Analysis dialog box is displayed and the analysis is visible on the selected elements.



You can modify the values in the color range to highlight specific areas of the selected surface. To do this, click and drag the arrows delimiting the colors, or directly key in the values. You can also choose a different display for the color range: Linear, Sharp left, Sharp center or Sharp right.

The values are expressed in degrees, ranging from -90 to 90 degrees.

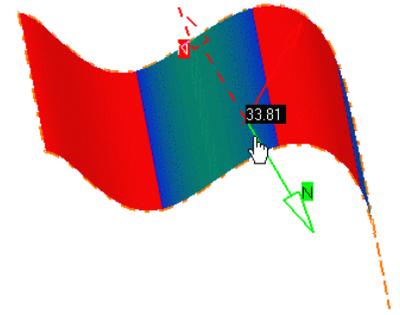
You can modify them by clicking on their corresponding arrow or by entering a value directly in the field.

The precision for entering values is 0.1 degree.

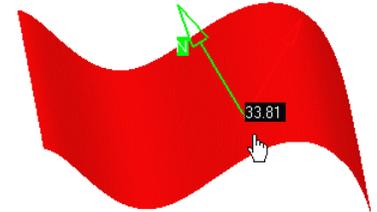
The cursor manipulation for colors is limited between -20.0 et 20.0 but the analysis is still performed between -90 and 90 degrees.

If you see no visualization on the selected element, move the pointers in the dialog box to redefine the draft values.

3. Activate the On the fly analysis checkbox and move the pointer over the surface.
Arrows are displayed under the pointer, identifying the normal to the surface at the pointer location (green arrow). As you move the pointer over the surface, the normal display is dynamically updated.



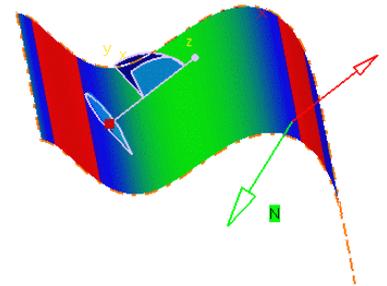
If you click the green arrow (Normal), you can invert it. In this case, the draft analysis is not possible any longer, the whole surface becomes red (highest value).



If you click the red arrow, it freezes the location for the arrow allowing general manipulations according to the compass.

The displayed value indicates the angle between the draft direction and the normal to the surface at the current point.

4. Bring the compass onto the selected surface to locally manipulate it and have a different view of the analysis without modifying the analysis values.



These values can be edited by double-clicking the compass which displays the Compass Manipulation dialog box.

5. Once you have finished analyzing the surface, click Close in the Draft Analysis dialog box.

- If you have moved to another capability, you need to redisplay the Draft Analysis dialog box by clicking the Draft Analysis icon  then by clicking Reset to remove the draft analysis from the element.
- A draft analysis can be performed just as well on a set of surfaces.
- Double-click the arrows from the color range to display the color chooser allowing you to re-define the color range.



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new Performing a Mapping Analysis



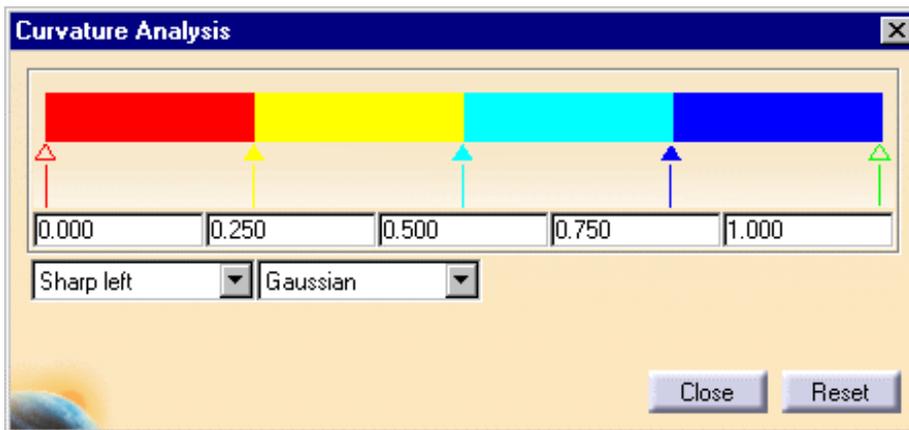
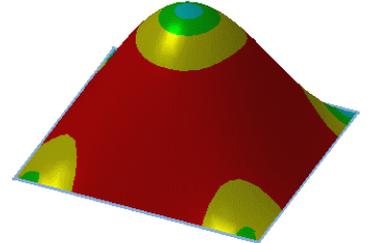
This task shows how to analyze the mapping curvature of a surface.



Open the Mappinganalysis.CATPart document from the online/Samples/ShapeDesign directory.

1. Select a surface.
2. Click the Mapping Analysis icon .

The curvature analysis dialog box is displayed and the analysis is visible on the selected element.

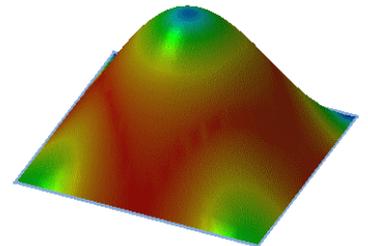


3. Choose the linear option from the dialog box.

Available options to display the color range are: linear, sharp left, sharp center or sharp right.

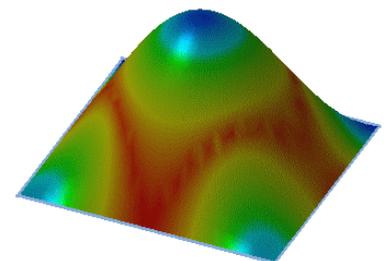
The values are ranging from 0 to 1, corresponding to the minimum and maximum Mapping curvature respectively.

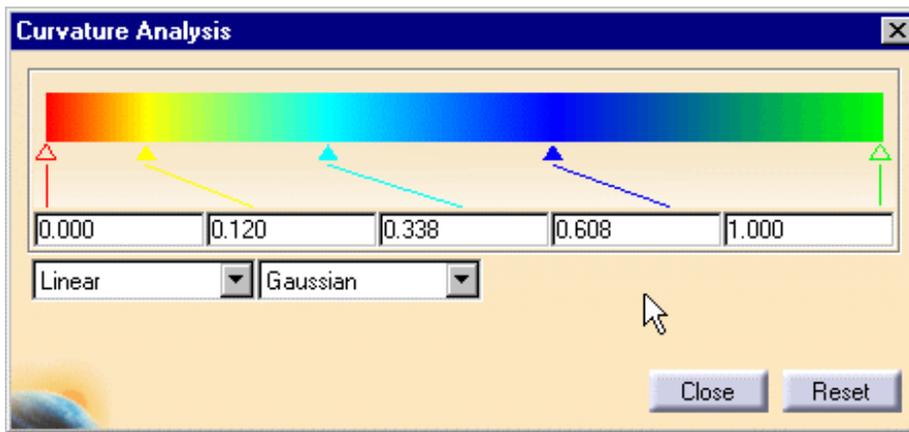
The surface now looks like this:



4. Modify the values in the color range to highlight specific areas of the selected surface.

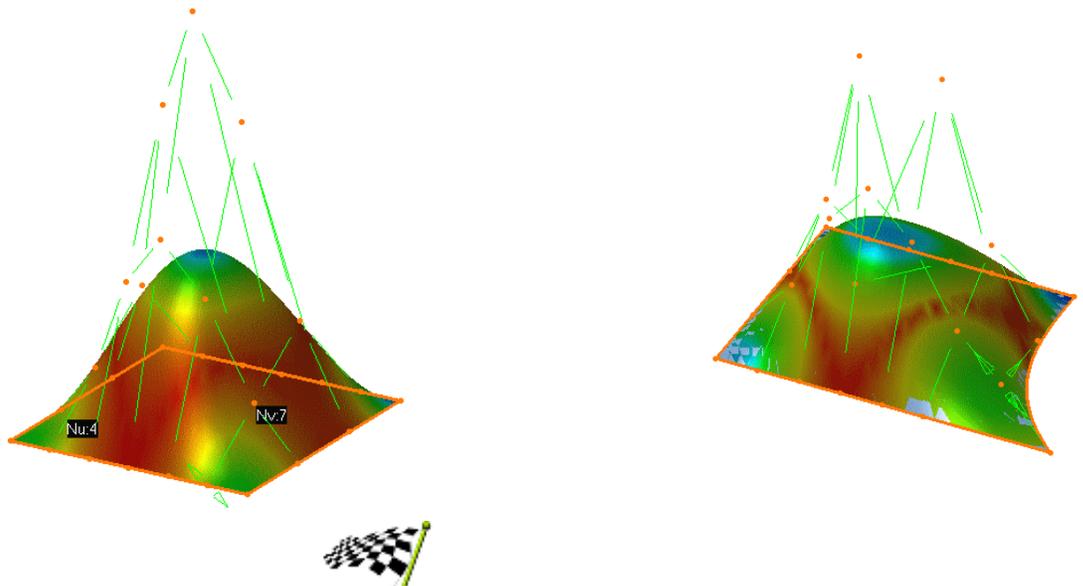
To do this, click and drag the arrows delimiting the colors, or directly key in the values.





5. Click Close to exit the analysis capability, or click Reset to come back to default values for the color range.

- Mapping analyses can be performed on a set of surfaces.
- Double-click the arrows from the color range to display the color chooser allowing you to re-define the color range.
- You can display the control points by clicking the Control Points icon , still viewing the Mapping Analysis. This allow you to check the impact of any modification on the surface according to the Mapping Analysis.



- | | | |
|--|--|---|
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| |  Performing a Mapping Analy | |

Advanced Tasks

The advanced tasks you will perform in the CATIA - Generative Shape Design workbench include managing the specification tree and interoperating with other workbenches.



Managing Open Bodies in the Specification Tree



This task shows how to manage the specification tree. This involves;

- inserting open body entities
- removing open body entities
- changing body.

You will find other useful information in the [Managing Groups](#) section.

You can insert and manipulate open bodies in the specification tree in much the same way as you manage files in folders.

These management functions have no impact on the part geometry.



You should refer to the [Copying and Pasting](#) section for information about how open bodies can be used in a part edition context.

Inserting an Open Body



1. In the specification tree, select the branch where you want the new open body to be inserted.

This branch is known as a father location which can be a part, a body or another open body entity.

2. Select the Insert > Open Body menu command.

The Insert Open Body dialog box appears.

3. Select the entities that are to be included in the new open body.

4. Click OK to create the open body at the desired location.



Removing an Open Body

This is only possible when the father location of the open body is another open body.

1. Right-click the desired open body then select the Remove Open Body contextual command.

The open body is removed and its constituent entities are included in the father open body.

Moving an open body to a new body

1. Right-click the desired open body in the specification tree and select the Change Body command from the contextual menu.

The Change Body dialog box appears.



2. Select the new body where the open body is to be located.

3. Click OK to move the open body to the new body.



Using Hybrid Parts



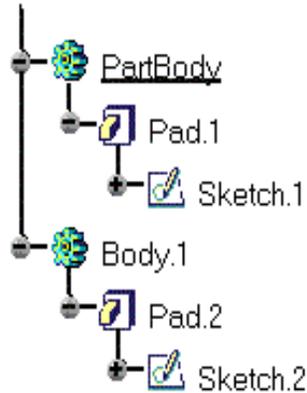
This task shows how to create a hybrid part comprising wireframe, surface and solid geometry.



You must have access to the Part Design product.

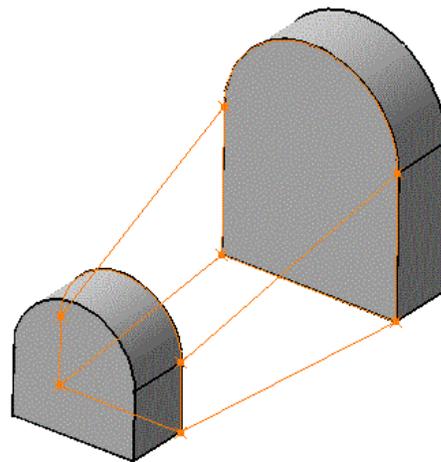


1. In the Generative Shape Design workbench, open a document comprising solid entities.



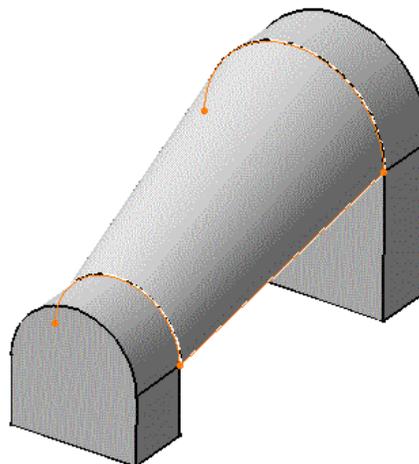
2. Click the Line icon  then create construction lines between the opposite vertices of the two pads.

These lines are created in an Open Body entity.



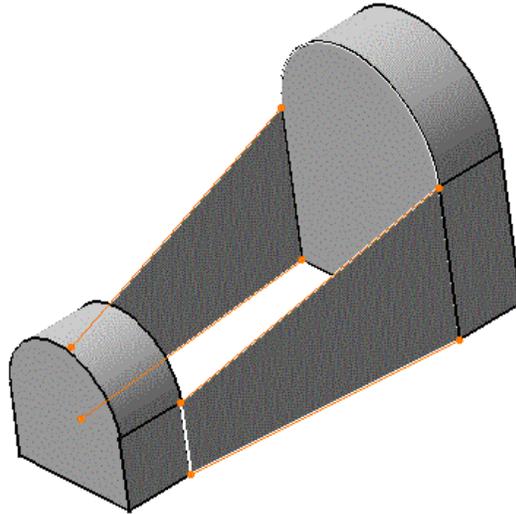
3. Click the Loft icon  and create a lofted surface between the curved edges of the two pads.

Create another lofted surface between the bottom edges of the two pads.

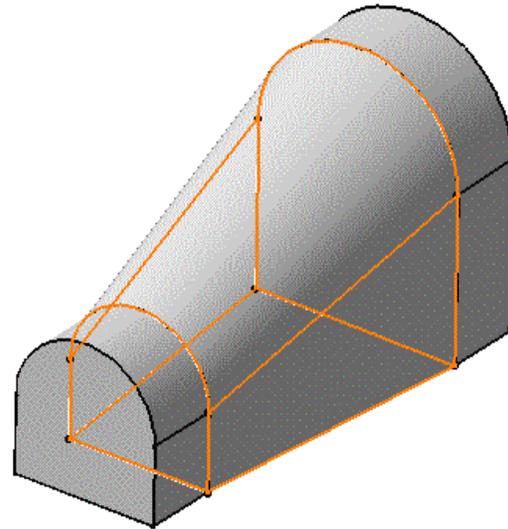


4. Click the Sweep icon  and create a swept surface between two opposite vertical edges of the two pads.

Create another swept surface on the other side of the side of the two pads.



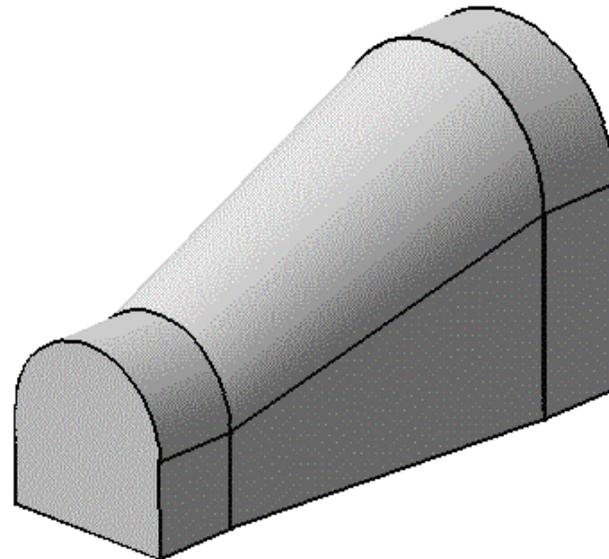
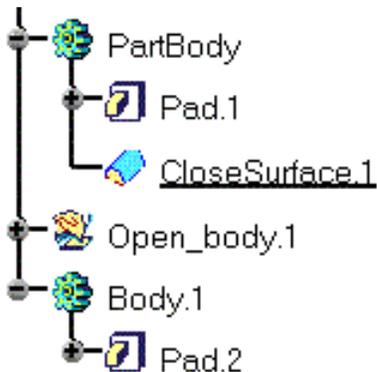
5. Click the Join icon  then select the four surfaces to create a single joined surface.



6. Open the Part Design workshop and select the Closed Surface icon .

7. Select the joined surface in order to close it.

The model and specification is updated with the Close Surface feature.





Up



Managing Open Bodies



Using Multi-Documents



Using Hybrid Parts



Using Multi-Documents



This task shows how to use multi-documents in order to design in the context of a product.

You will copy an open body from one document to another, edit the original part and the copied part independently and finally update the copied part with modifications made to the original part.

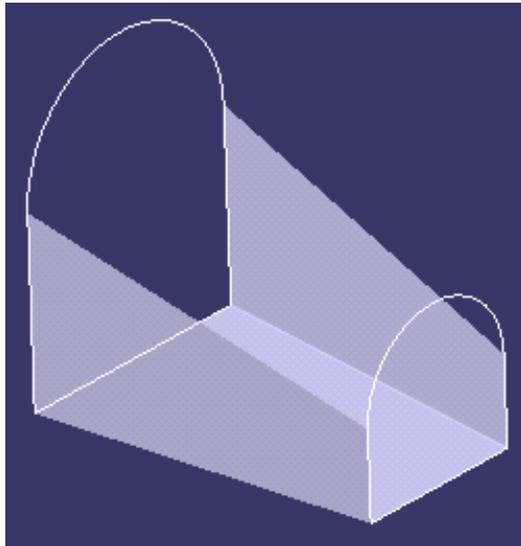


Select Tool -> Options command and in the Part General page make sure that the Keep link with selected object and Synchronize all external references for update options are set.



1. Select the Open Body of Part1 in the specification tree.

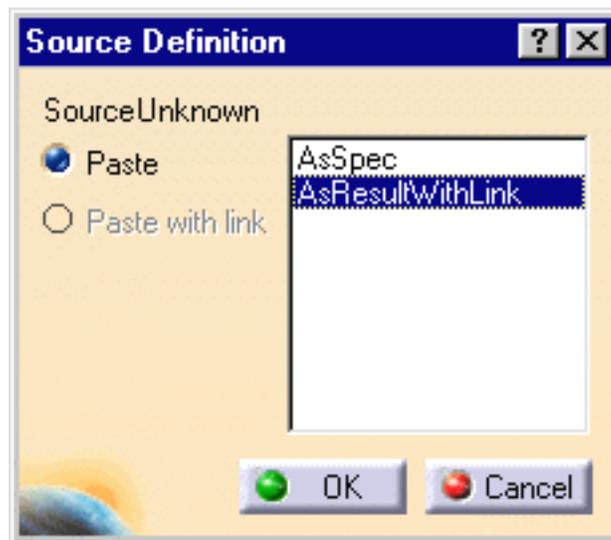
2. Select the Edit -> Copy command.



3. Open a new CATPart document called Part2 and position the cursor anywhere in the specification tree.

4. Select the Edit
-> Paste Special
command.

The Source
Definition dialog
box appears.

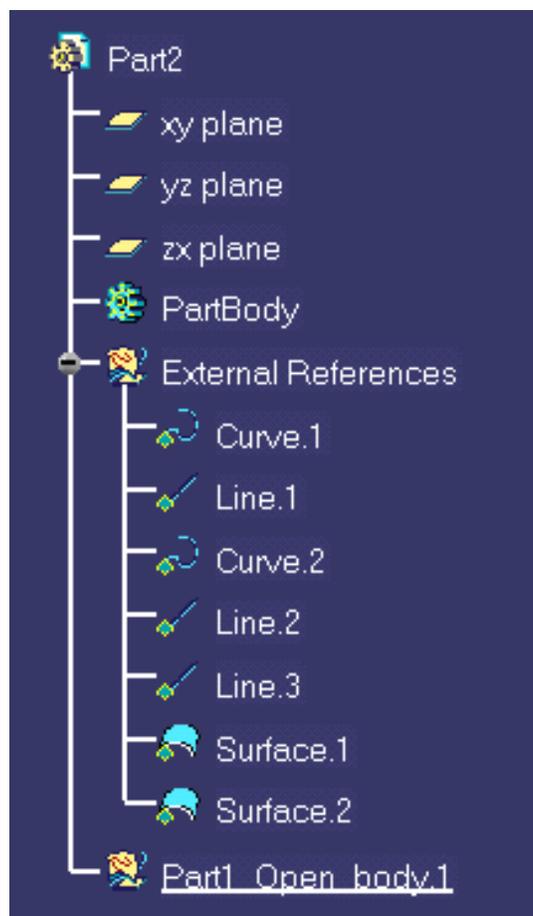


5. Select the
AsResult
WithLink option
and click OK.

In this case the
Open Body is
copied without its
design
specifications.

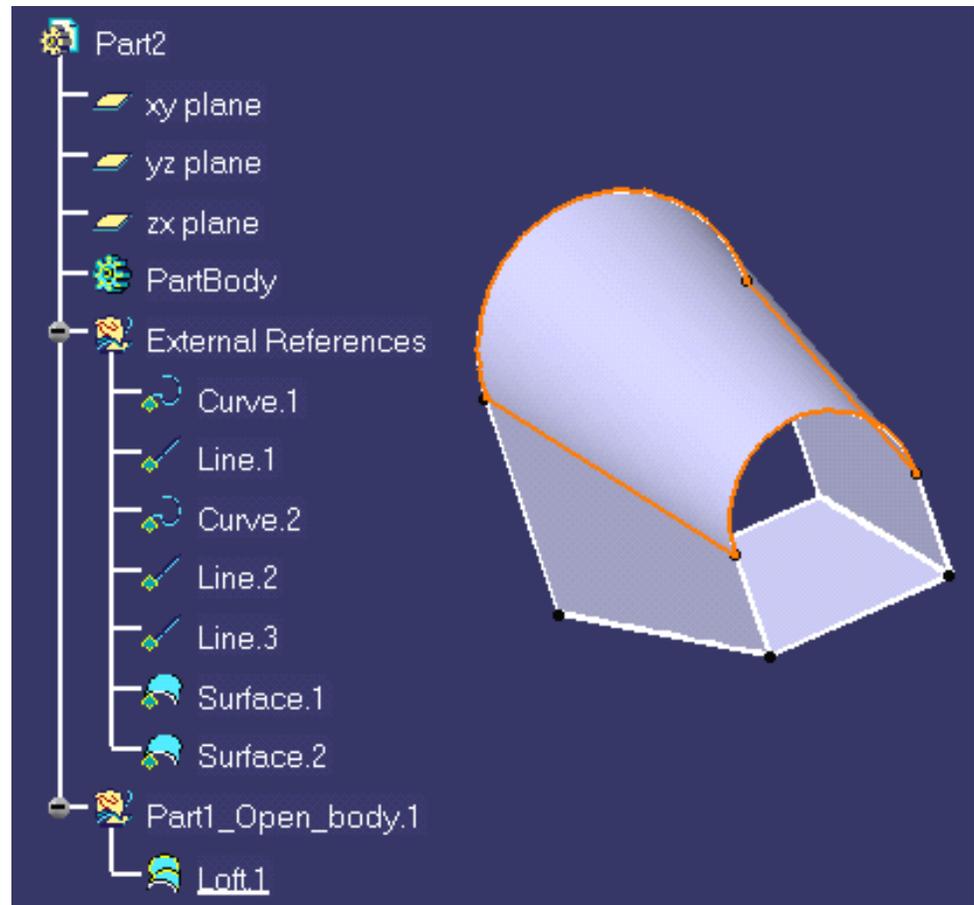
Note that the
AsSpec option
copies the
selected object
along with its
design
specifications.

The Open Body
is represented in
Part2 as an
External
Reference
object.

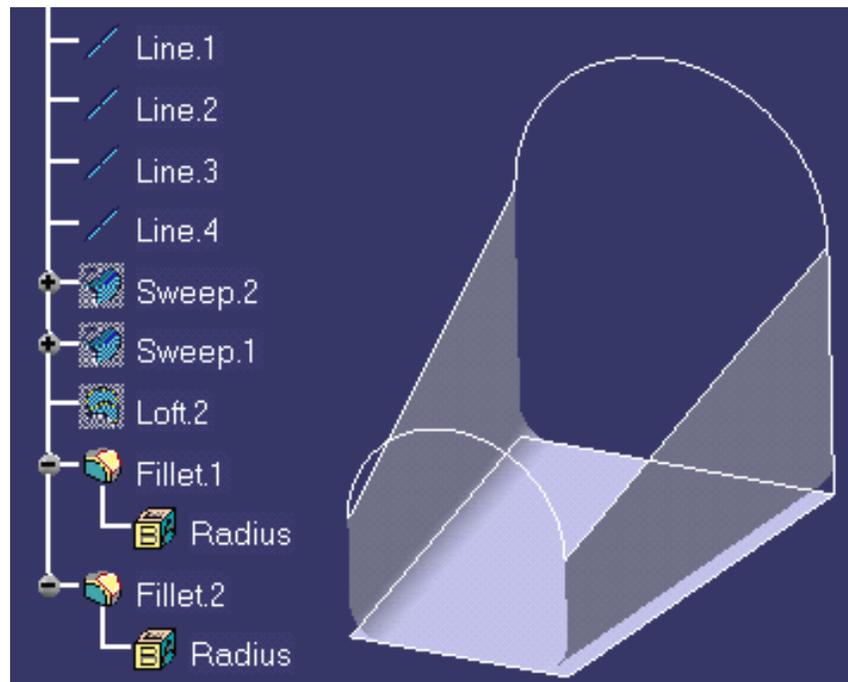


6. Edit Part2 by adding a lofted surface between the two curved end sections.

Remember that this modification has no effect on the original part.

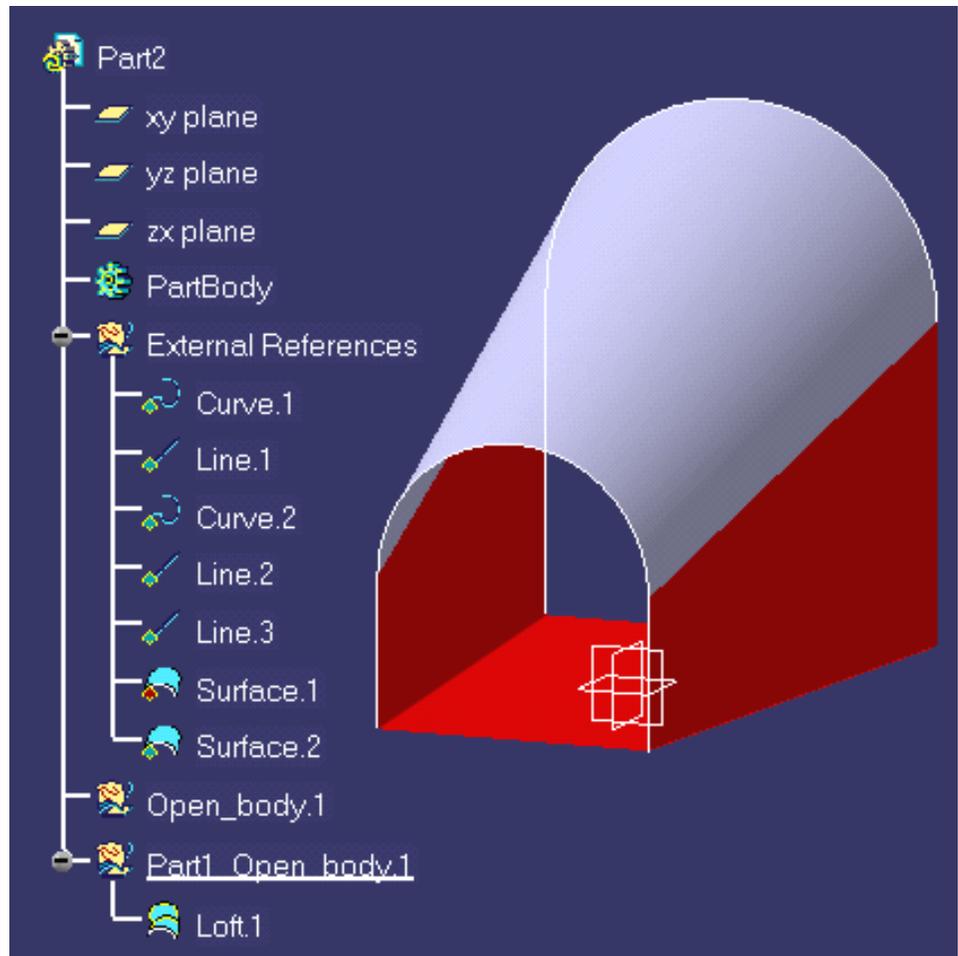


7. Return to the Part1 document and edit the two fillet surfaces by increasing the radius values, for example.



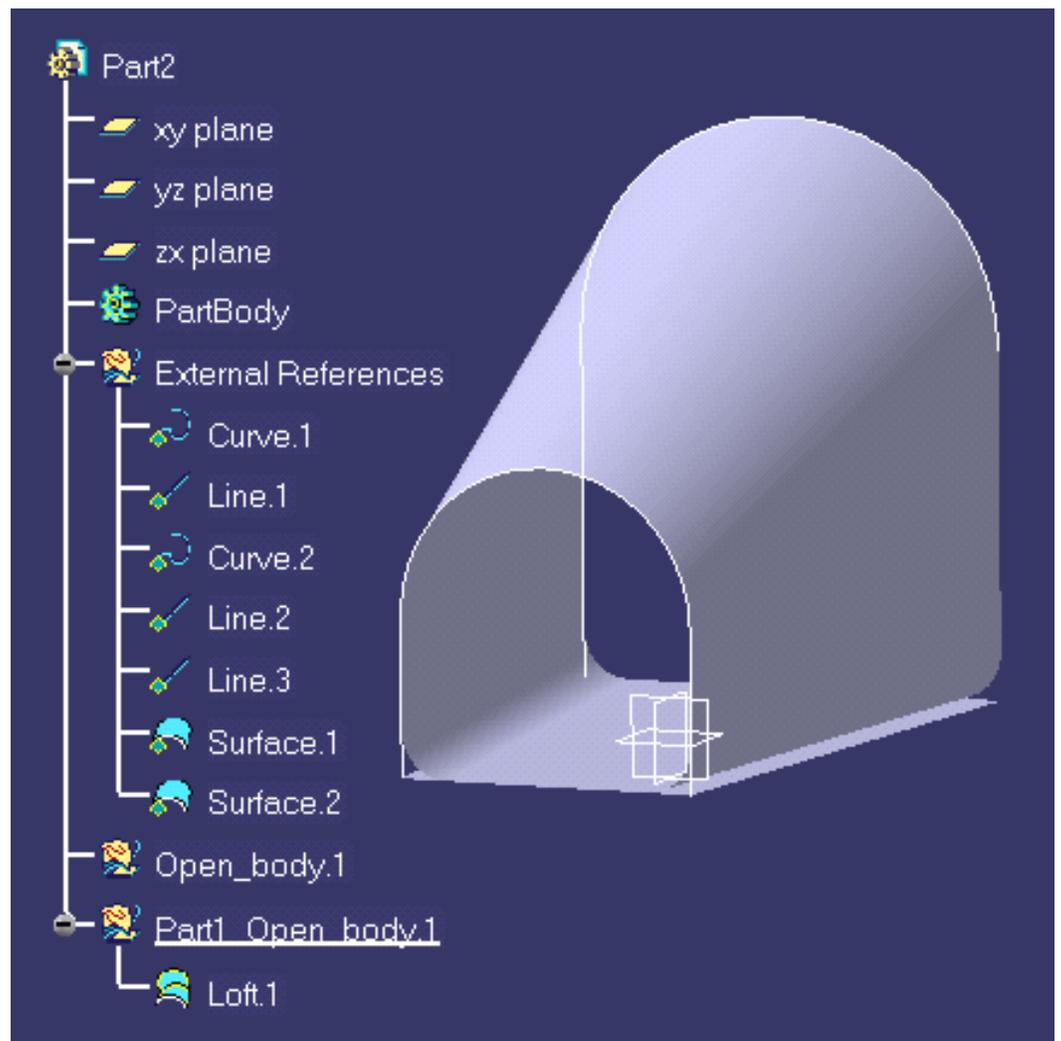
In the Part2 document, an update symbol is displayed next to Part2 in the specification tree.

The elements in the part that are impacted by the modifications to Part1 are highlighted in red.



8. Select the Update command to update Part2.

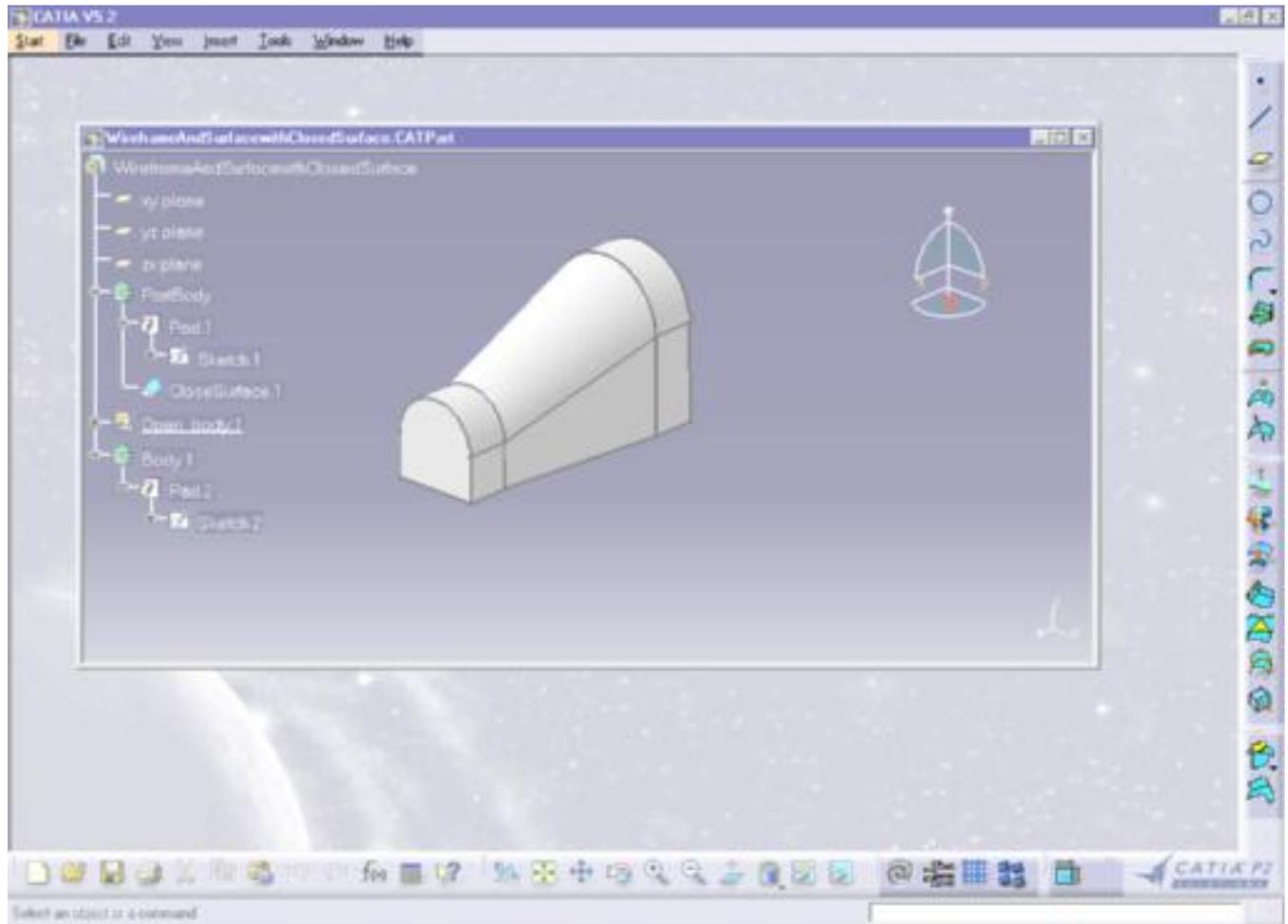
Part2 now includes the modifications made to Part1.



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](#)

[Wireframe Toolbar](#)

[Surfaces Toolbar](#)

[Operations Toolbar](#)

[Tools Toolbar](#)

[Constraints Toolbar](#)

[Historical Graph](#)

CATIA - Generative Shape Design Menu Bar

The various menus and menu commands that are specific to CATIA - Generative Shape Design are described below.



Tasks corresponding to general menu commands are described in the *CATIA Version 5 Infrastructure User's Guide*.

Edit

Please note that most of the edit commands available here are common facilities offered by the CATIA Version 5 Infrastructure.

The specific CATIA - Generative Shape Design edit commands depend on the type of object being edited: Open Body or other entity.

	Command...	Command...	Description...
	Undo	Ctrl+Z	
	Redo	Ctrl+Y	Undo Cancels the last action.
	Update	Ctrl+J	Redo Recovers the last action that was undone. Update See Updating Your Design
	Cut	Ctrl+X	Cut
	Copy	Ctrl+C	Copy See Copying and Pasting and Using Multi-documents .
	Paste	Ctrl+V	Paste
	Paste Special...		Paste Special
	Delete	Del	Delete See Deleting Geometry
	Search...		Search Allows searching and selecting objects.
	Links...		Links Manages links to other documents.
	Properties	Alt+Enter	Properties Allows displaying and editing object properties.
	Scan or Define In Work Object...		Scan or Define in Work Object Allows scanning the part and working locally on an object.
	Circle.1 object		
	Definition...		
	Change Body...		XXX object -> Definition See Editing Definitions
	Reset Properties		XXX object -> Change Body Allows Managing Open Bodies
			Reset Properties Allows resetting object properties

Insert



For...

See...

Open body
 Sketcher
 Wireframe
 Surfaces
 Operations
 Constraints and Analysis

[Managing Open Bodies](#)
 Refer to the *CATIA Version 5 Part Design User's Guide*.
[Insert -> Wireframe](#)
[Insert -> Surfaces](#)
[Insert -> Operations](#)
[Insert -> Constraints and Analysis](#)

Insert -> Wireframe



For...

See...

Point [Creating Points](#)
 Line [Creating Lines](#)
 Plane [Creating Planes](#)
 Projection [Creating Projections](#)
 Intersection [Creating Intersections](#)
 Circle [Creating Circles](#)
 Spline [Creating Splines](#)
 Corner [Creating Corners](#)
 Connect Curve [Creating Connect Curves](#)
 Parallel Curve [Creating Parallel Curves](#)
 Boundary [Creating Boundary Curves](#)

Insert -> Surfaces



For...

See...

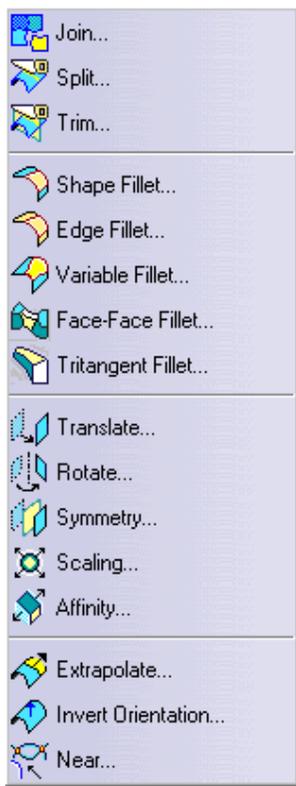
Extrude [Creating Extruded Surfaces](#)
 Revolve [Creating Revolution Surfaces](#)
 Offset [Creating Offset Surfaces](#)
 Sweep [Creating Swept Surfaces](#)
 Fill [Creating Fill Surfaces](#)
 Loft [Creating Lofted Surfaces](#)
 Extract [Extracting Geometry from Solids](#)

Insert -> Operations



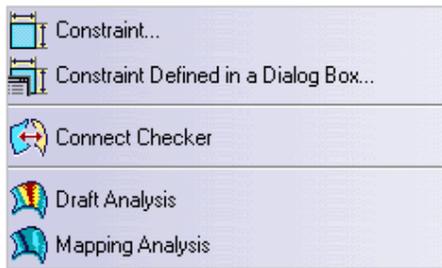
For...

See...



Join	Joining Curves and Surfaces
Split	Splitting Geometry
Trim	Trimming Geometry
Shape Fillet	Filleting Surfaces
Edge Fillet	Edge Fillets
Variable Fillet	Variable Radius Fillets
Face-Face Fillet	Face-Face Fillets
Tritangent Fillet	Tritangents Fillets
Translate	Translating Geometry
Rotate	Rotating Geometry
Symmetry	Performing Symmetry on Geometry
Scaling	Transforming Geometry by Scaling
Affinity	Transforming Geometry by Affinity
Extrapolate	Extrapolating Geometry
Invert orientation	Inverting the Orientation of Geometry
Near	Creating Nearest Entity of a Multiple Element

Insert -> Constraints and Analysis



For...	See...
Constraint	Creating Constraints
Constraint Defined in Dialog Box	Checking Connections between elements
Connect Checker	Checking Connections between elements
Draft Analysis	Performing a Draft Analysis
Mapping Analysis	Performing a Mapping Analysis

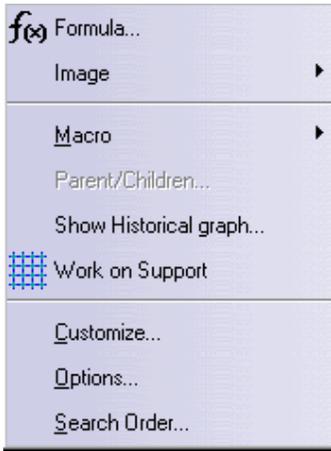
Tools

Please note that most of the Tools commands available here are common facilities offered by the CATIA Version 5 Infrastructure.

Specific CATIA - Generative Shape Design commands are described in the present document.



Command...	Description...
Formula	Allows editing parameters and formula.
Image	Allows capturing images.
Macro	Allows recording, running and editing macros.
Parent/Children	Allows viewing the parents and children of a selected object.
Show Historical graph	See Using the Historical Graph
Work on Support	See Working with a Support



Customize
Options
Search Order

Allows customizing the workbench.
Allows customizing settings.
Allows specifying a search order list.



Wireframe Toolbar

This toolbar contains the following tools for creating wireframe elements.



-  See [Creating Points](#)
-  See [Creating Lines](#)
-  See [Creating Planes](#)
-  See [Creating Projections](#)
-  See [Creating Intersections](#)
-  See [Creating Circles](#)
-  See [Creating Splines](#)
-  See [Creating Corners](#)
-  See [Creating Connect Curves](#)
-  See [Creating Parallel Curves](#)
-  See [Creating Boundary Curves](#)



Surfaces Toolbar

This toolbar contains the following tools for creating surfaces.



-  See [Creating Extruded Surfaces](#)
-  See [Creating Revolution Surfaces](#)
-  See [Creating Offset Surfaces](#)
-  See [Creating Swept Surfaces](#)
-  See [Creating Fill Surfaces](#)
-  See [Creating Lofted Surfaces](#)
-  See [Extracting Geometry from Solids](#)



Operations Toolbar

This toolbar contains the following tools for performing operations on surface and wireframe elements.



-  See [Joining Curves and Surfaces](#)
-  See [Splitting Geometry](#)
-  See [Trimming Geometry](#)
-  See [Tritangent Fillets](#)
-  See [Shape Fillets](#)
-  See [Edge Fillets](#)
-  See [Variable Radius Fillets](#)
-  See [Face-Face Fillets](#)
-  See [Translating Geometry](#)
-  See [Performing a Symmetry on Geometry](#)
-  See [Transforming Geometry by Scaling](#)
-  See [Transforming Geometry by Affinity](#)
-  See [Rotating Geometry](#)
-  See [Extrapolating Geometry](#)



Tools Toolbar

This toolbar contains the following tools to help you model your shape designs.



See [Updating Constraints](#)



See [Using the Historical Graph](#)



See [Working with a Support](#)



See [Creating Datums](#)



Constraints and Analysis Toolbar

This toolbar contains the following tools to help you manage constraints between geometric elements and perform analysis.



See [Creating Constraints](#).



See [Checking Connections between Elements](#)



See [Performing a Draft Analysis](#)

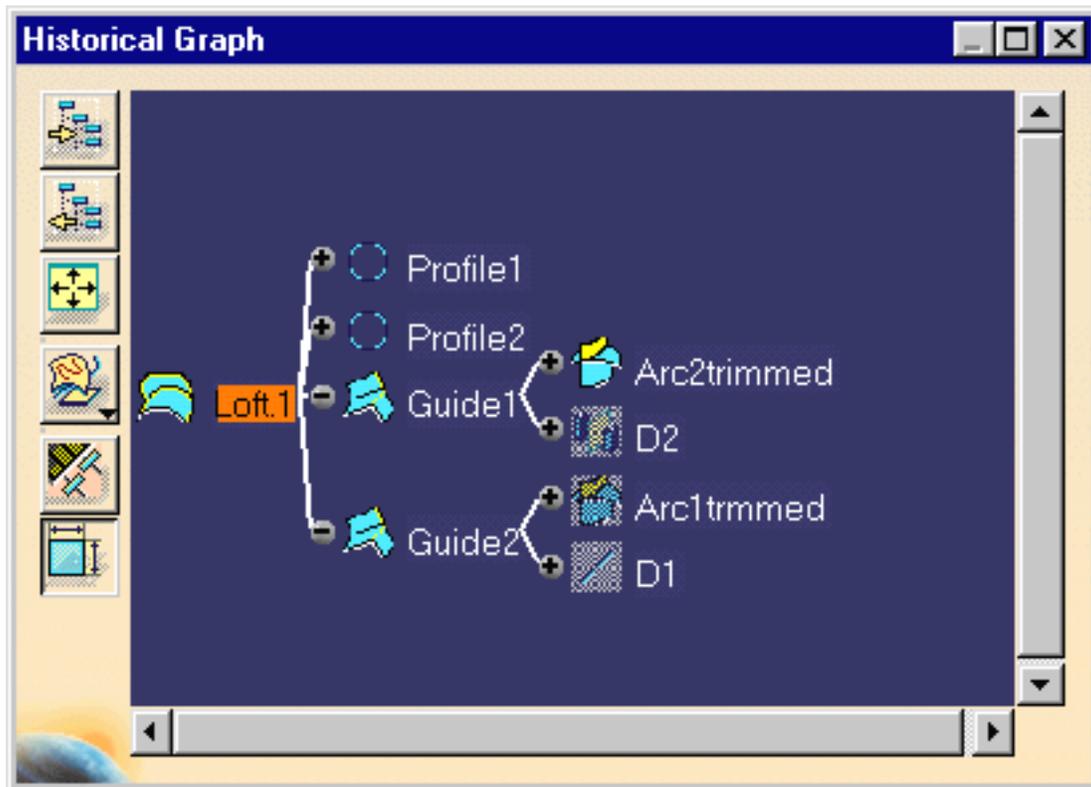


See [Performing a Mapping Analysis](#)



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 to the elements in the graph.

Constraints

Displays any constraints associated to 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.



Glossary

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.

J

join An operation in which adjacent curves or adjacent curves can be joined.

L

lofted surface A surface that is obtained by sweeping one or more planar section curves along a spine, which may be automatically computed or user-defined. The surface can be made to follow one or more guide curves.

O

offset surface A surface that is obtained by offsetting an existing surface a specified distance.

P

parent A status defining the hierarchical relation between a feature or element and another feature or element.

part A 3D entity obtained by combining different features. It is the content of a CATPart document.

part body A component of a part made of one or several features.

profile An open or closed shape including arcs and lines.

R

revolution surface A surface that is obtained by revolving a profile around an axis.

rotate An operation in which an element is rotated by a specified angle about an given axis.

S

scaling An operation that resizes an element to a percentage of its initial size.

sketch A set of geometric elements created in the Sketcher workbench. For instance, a sketch may include a profile, construction lines and points.

split An operation in which one element is cut by another element.

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

symmetry An operation in which an element is transformed by means of a mirror symmetry with respect to a reference plane, line or point.

T

translate An operation in which an element is displaced a specified distance along a given direction.

trim An operation in which two element cut each other mutually.

W

wireframe element Elements such as points, lines or curves that can be used to represent the outline of a 3D object.

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