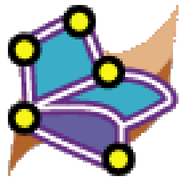
Quick Surface Reconstruction



Preface

Quick Surface Reconstruction easily and quickly recovers surfaces from digitized data that has been cleaned up and tessellated using the Digitized Shape Editor product.

Quick Surface Reconstruction offers several approaches to recover surfaces depending of the type of shape :

- Organic shapes, i.e. free form surfaces, without features such as cylinders, fillets, planes, etc.
- Mechanical shapes (plane, cylinder, sphere, cone).

Thanks to Quick Surface Reconstruction tools that analyze curvature or iso-slope properties, users can easily create polygon segmentations in pertinent surfaces area. Quick Surface Reconstruction includes its own quality checking tools.

The Quick Surface Reconstruction user's guide has been designed to show you how to reconstruct surfaces using these powerful tools. We recommend that you go through the "Getting Started" chapters first, to learn more about those working methods.

Using this Guide
More Information
Conventions

Using this Guide

This User's Guide is intended for the user who needs to become quickly efficient with Quick Surface Reconstruction. Before reading it, you should be familiar with the basic concepts, such as the document windows, standard toolbars and menus.

To make the most out of this book, we suggest that a beginning user reads the Getting Started chapter first of all and the Workbench Description to find his way around the Quick Surface Reconstruction workbench.

Where to Find More Information

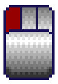
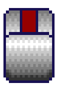
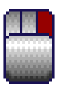
Prior to reading this book, we recommend that you read the Infrastructure User's Guide that describes generic capabilities common to all products. It also describes the general layout and interoperability between workbenches.

Conventions

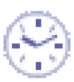
Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications. The following text conventions may be used:

- The titles of CATIA documents *appear in this manner* throughout the text.
- **File -> New** identifies the commands to be used.

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

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

Graphic conventions are denoted as follows:

	indicates the estimated time to accomplish a task.
	indicates a target of a task.
	indicates the prerequisites.
	indicates the scenario of a task.
	indicates tips
	indicates a warning.
	indicates information.
	indicates basic concepts.

indicates methodological information.

indicates reference information.

indicates information regarding settings, customization, etc.

indicates the end of a task.

indicates functionalities that are new or enhanced with this Release.
Enhancements can also be identified by a blue-colored background in the left-hand margin or on the text itself.

indicates functionalities that are P1-specific.

indicates functionalities that are P2-specific.

indicates functionalities that are P3-specific.

allows you to switch back the full-window viewing mode.

These icons in the table of contents correspond to the entries or mode.

"Site Map".

"Split View" mode.

"What's New".

"Preface".

"Getting Started".

"Basic Tasks".

"User Tasks" or the "Advanced Tasks".

"Workbench Description".

"Customizing".

"Reference".

"Methodology".

"Glossary".

"Index".

What's New?

Enhanced Functionalities

Curve from Scans

In the Smoothing mode, curves are now created with a maximum order value (instead of a minimum order value in previous releases). The number of segments remains the maximum possible one. Split points can now be moved dynamically along the scan, and the Remove all points option has been added to the contextual menu of Split points.

Curves Network

Overlapping input curves are detected. A contextual menu is available to remove selected curves in the dialog box. The node tolerance value is visualized as a sphere. A tab page has been added to freeze some input curves.

Distance Analysis

A texture mapping option has been added.

Cloud Display

The Vertex display option has been added for a lighter display of meshes.

Getting Started

The following tutorial aims at giving you a feel of what you can do with Quick Surface Reconstruction. It provides a step-by-step scenario showing you how to use key capabilities.

Quick Surface Reconstruction offers several approaches:

- N-side domain approach with boundary constraints and deviation tolerance through inner points:
    - reconstructs surfaces that do not require the conservation of fillets or mechanical features,
    - is a good compromise between quality and productivity,
    - the result can be adjusted by deformation of the domain.
- Untrimmed approach:
    - reconstructs surfaces including the fillets,
    - does not require the creation of curves,
    - enables the recovery of virtual sharp edges,
    - applies fillets on virtual edges (adjustable radius).
- Mechanical approach:
    - reconstructs surfaces with fillets and mechanical features
    - models virtual sharp edges,
    - identifies mechanical features,
    - reconstructs fillets on virtual edges
    - enables to modify fillets and mechanical features through their parameters.

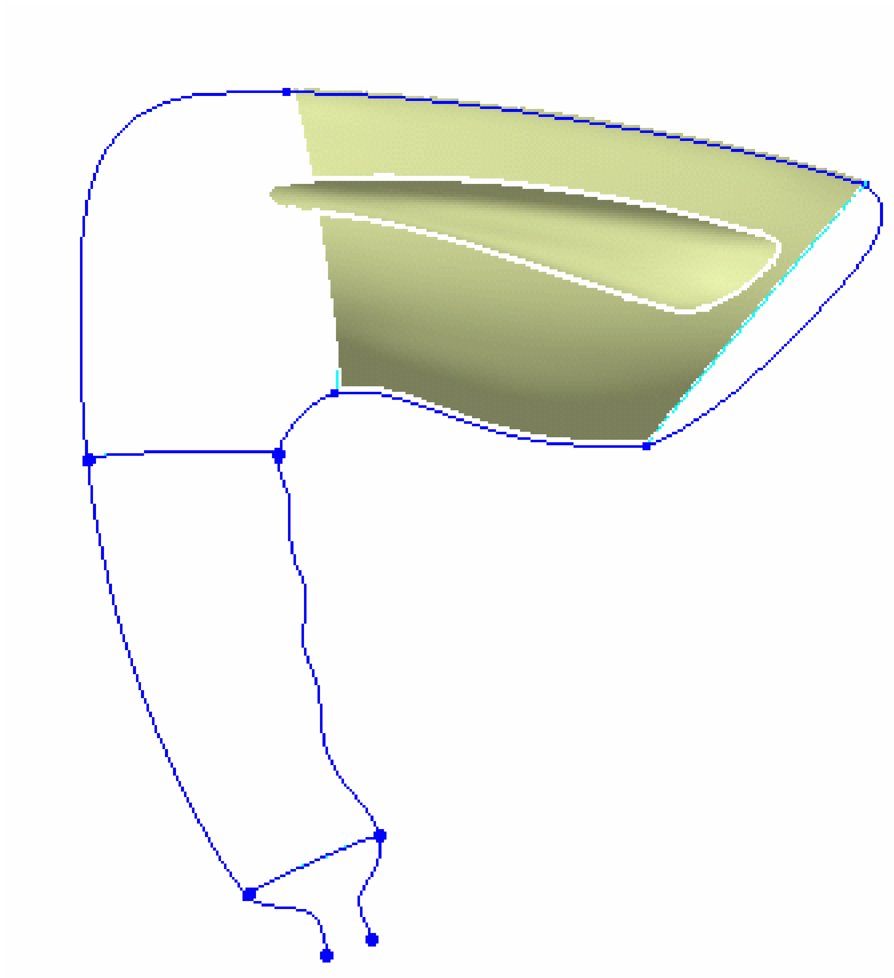
Whatever the method you choose, we recommend to work with polygons rather than with clouds of points.

The "Getting Started" part of this guide will illustrate the first approach, with the following tasks:

Note that we have changed the color of curves to blue and eventually renamed elements in the specification tree.

All together this scenario should take 15 minutes to complete.

The final cloud element will look like this:



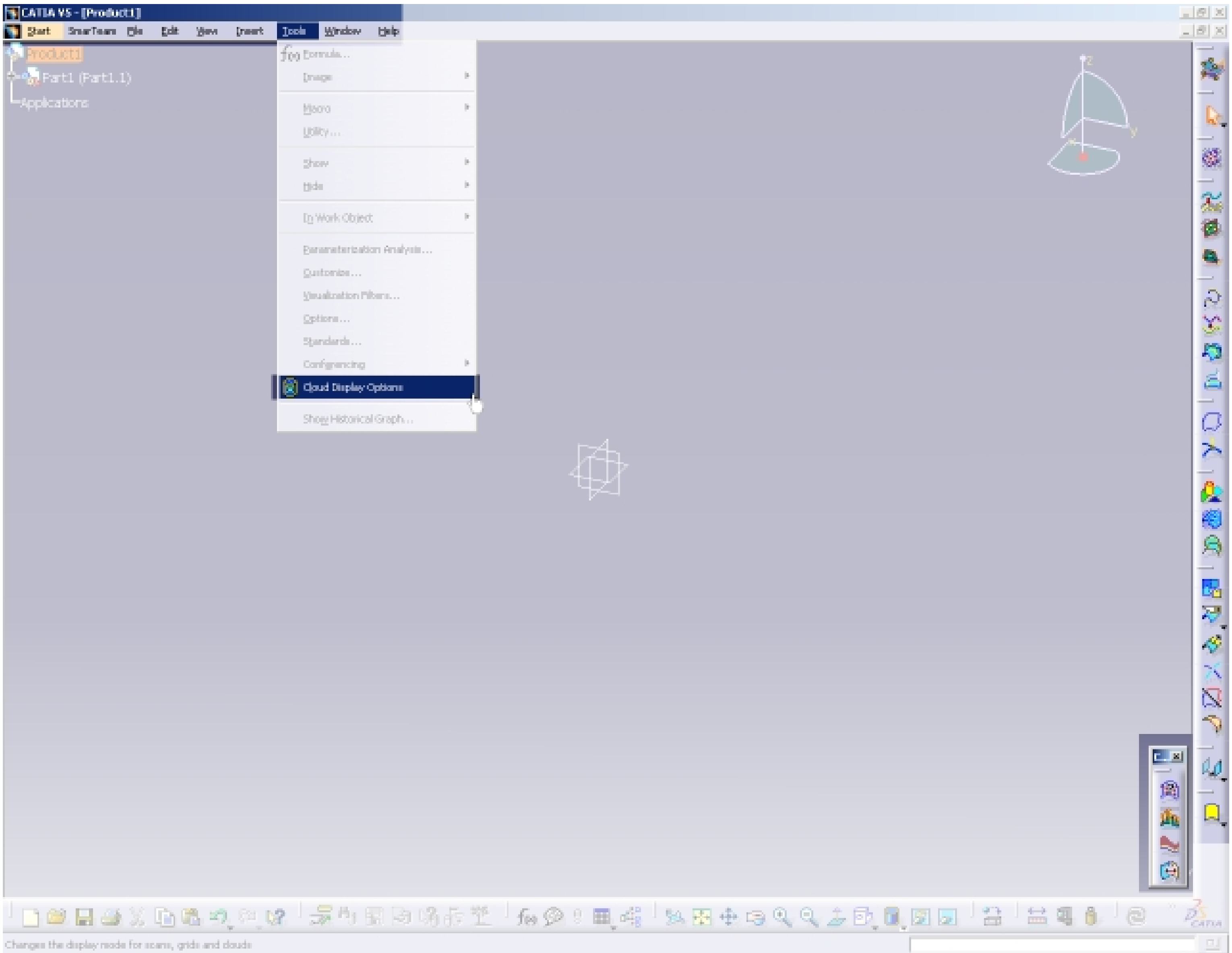
Starting the Quick Surface Reconstruction Workbench

The first task will show you how to enter the Quick Surface Reconstruction workbench and open data.

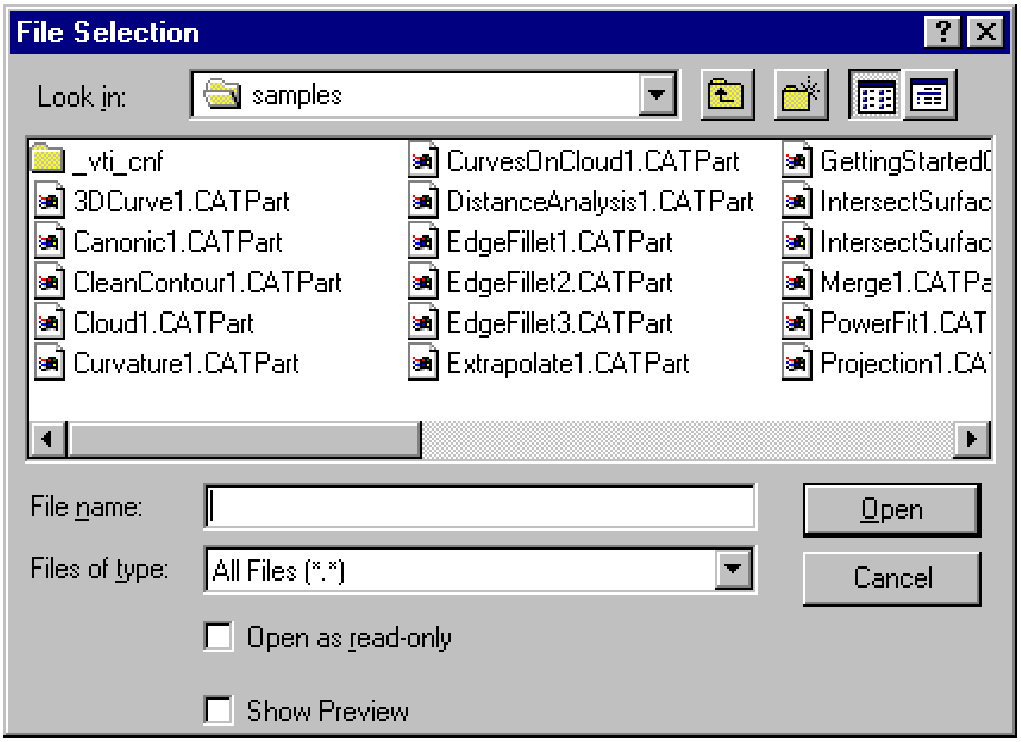
The only pre-requisites for this task is to have a current session running.

1. Choose **Quick Surface Reconstruction** from the **Start** menu.

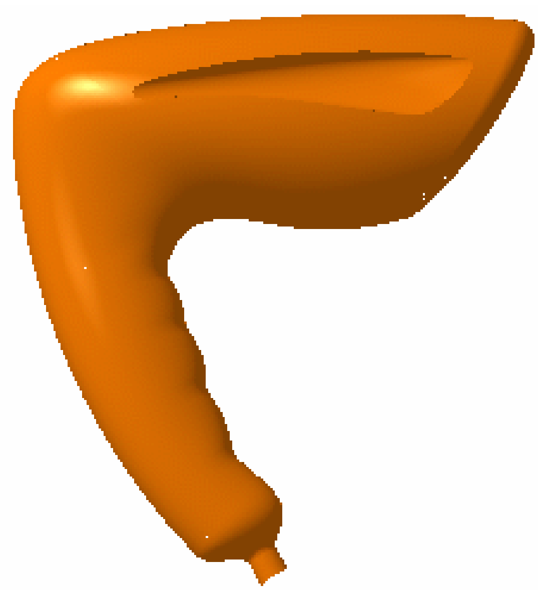
The Quick Surface Reconstruction workbench is displayed and ready to use.



2. Click the **Open** icon or select the **File->Open...** command. The following dialog box appears:



3. In the File Selection box, select the file location. Open the GettingStarted01.CATPart from the samples directory.



For more information, see Creating, Opening and Saving Documents in the *Infrastructure User's Guide*.

If you wish to use the whole screen space for the geometry, uncheck **Specification** in the **View** menu.

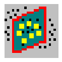
Preparing the part

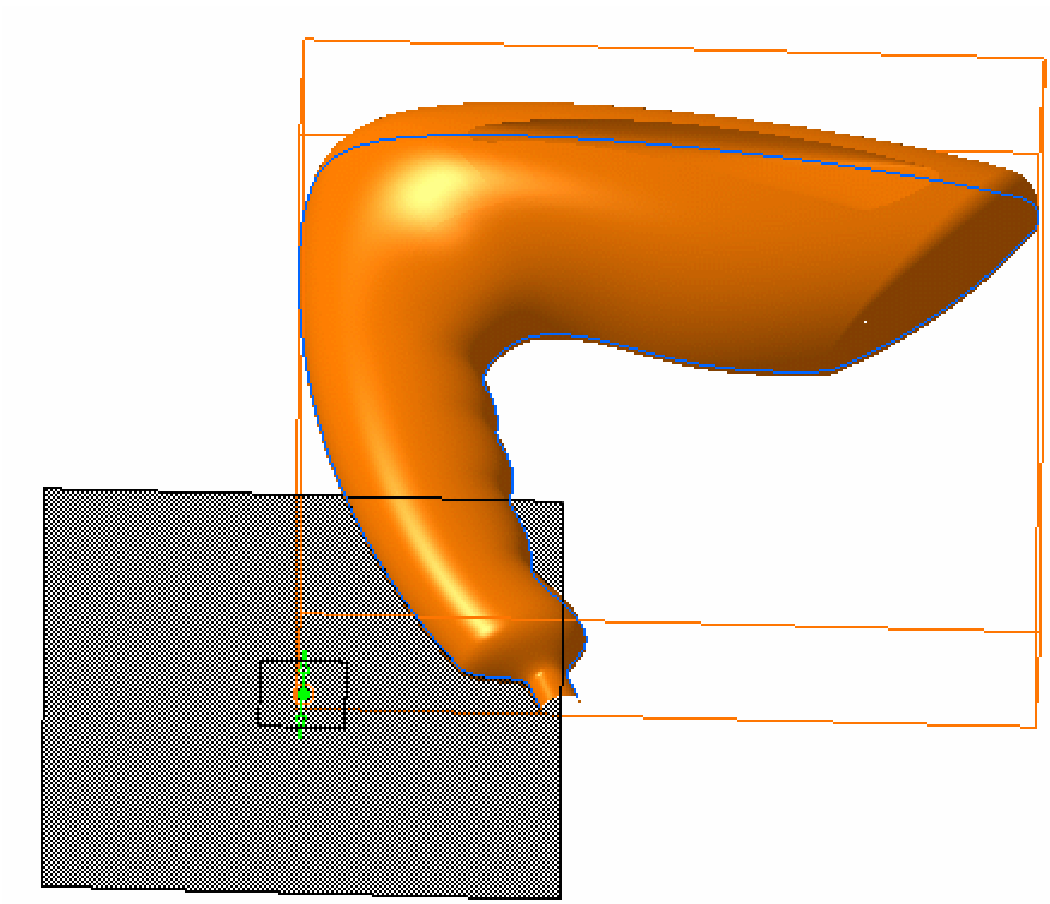
This task prepares the part for the reconstruction:

- creation of a curve defining a symmetry plane,
- creation of a surface tangent to the polygon,
- definition of reconstruction zones.

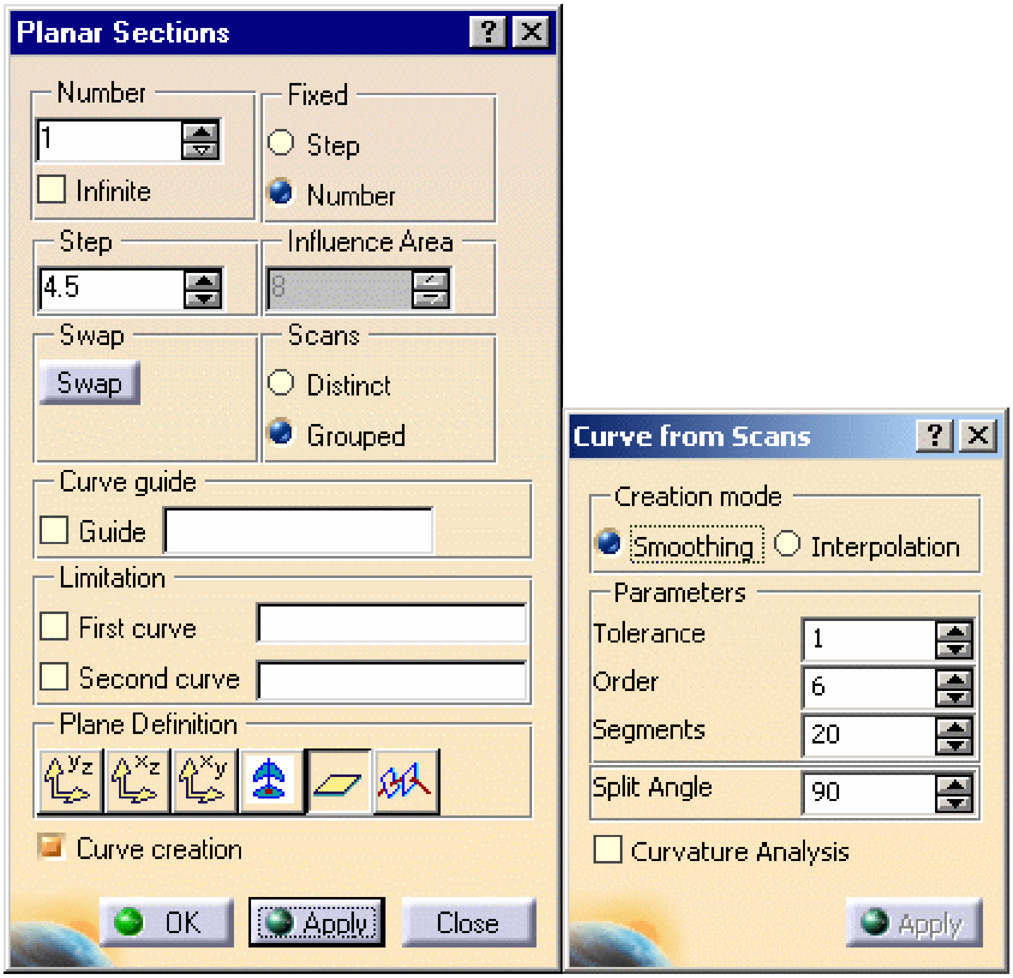
Open the GettingStarted01.CATPart from the samples directory.

Create the curve defining the symmetry plane:

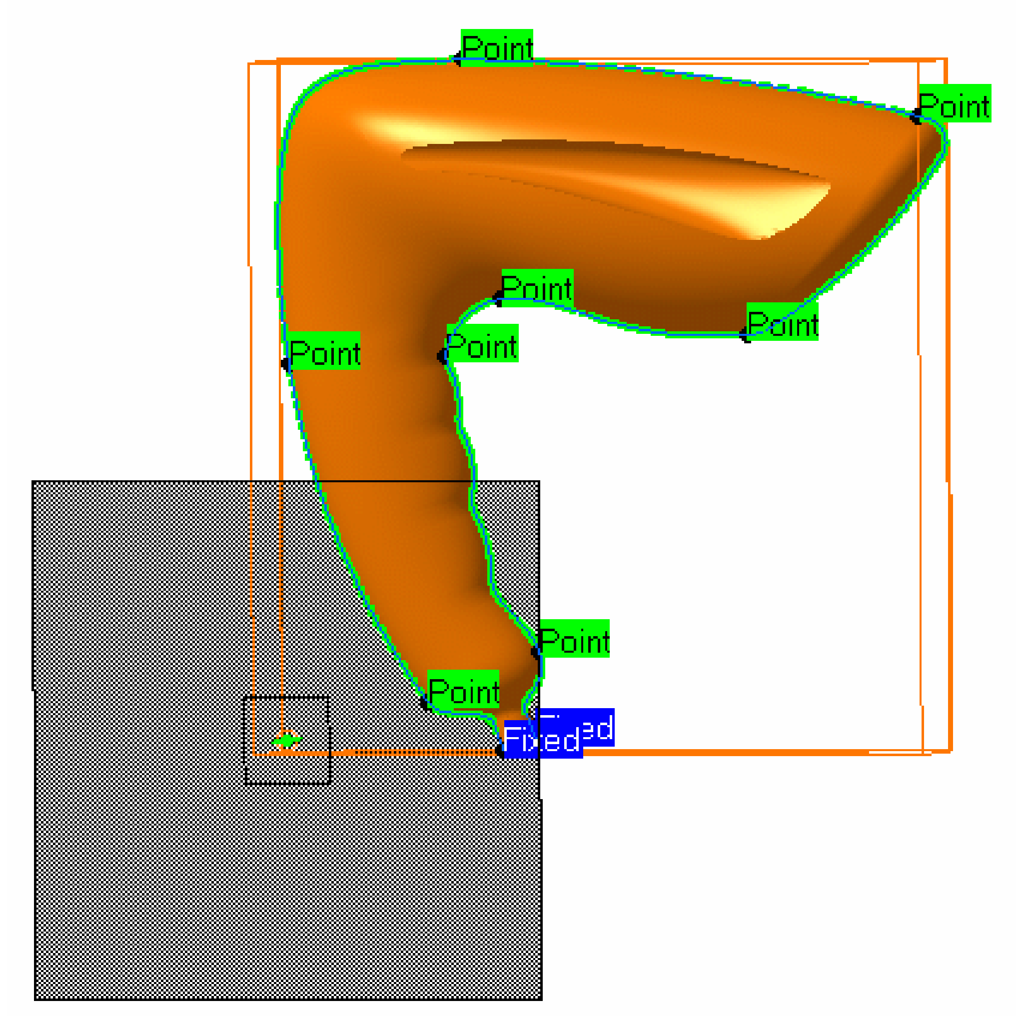
1. Click the **Planar Sections** icon  and the select the polygon.
2. Select the xy plane, enter 1 for the number of sections. Move the plane slightly upwards until a complete scan is visible. Clik **Apply**, the scan is computed, **PlanarSections.1** is displayed in the specification tree.



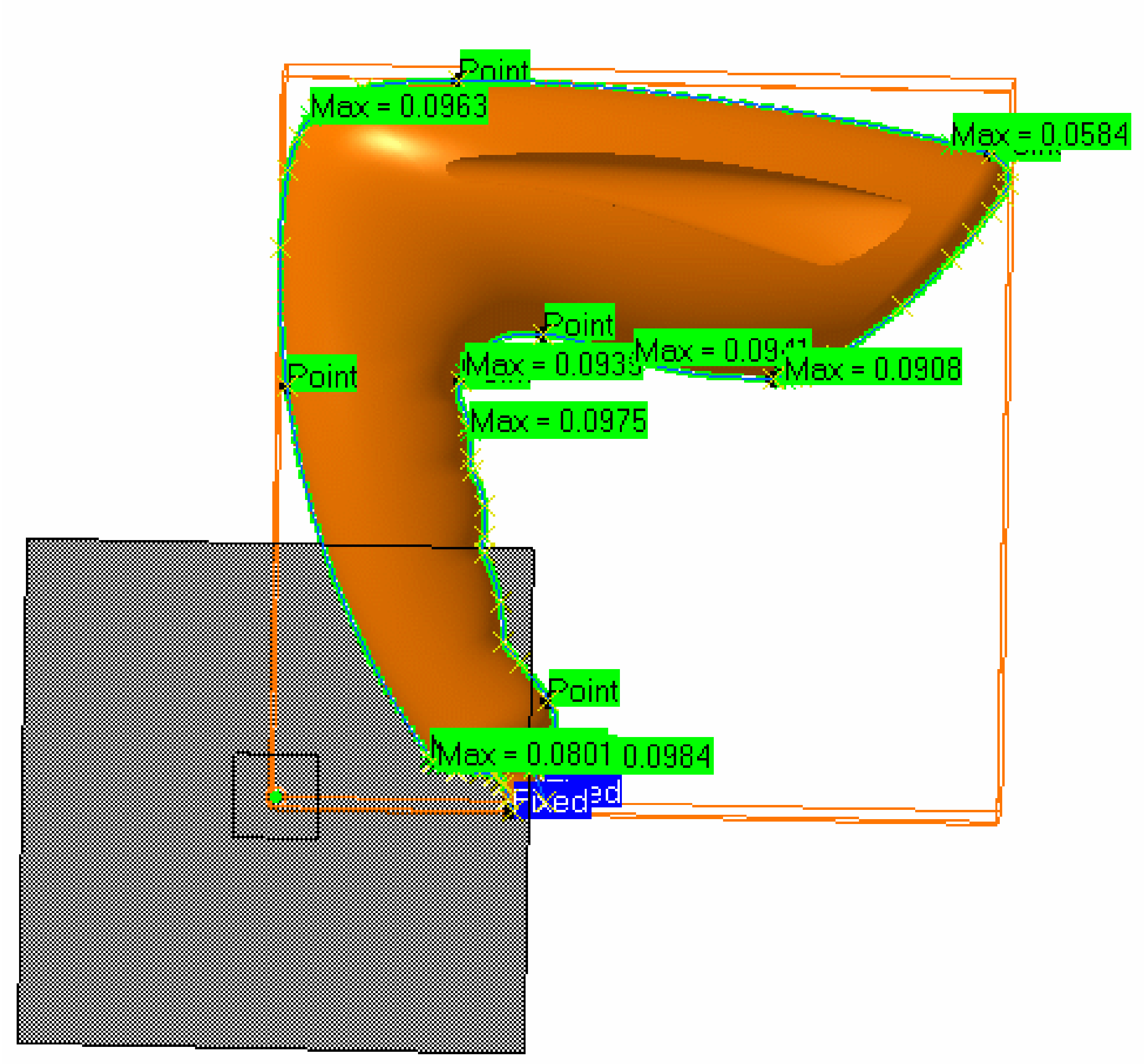
3. Check the curve creation option. The **Curve from Scans** dialog box is displayed. A curve is computed. Its segmentation is displayed, and **Curve.1** is displayed in the specification tree.



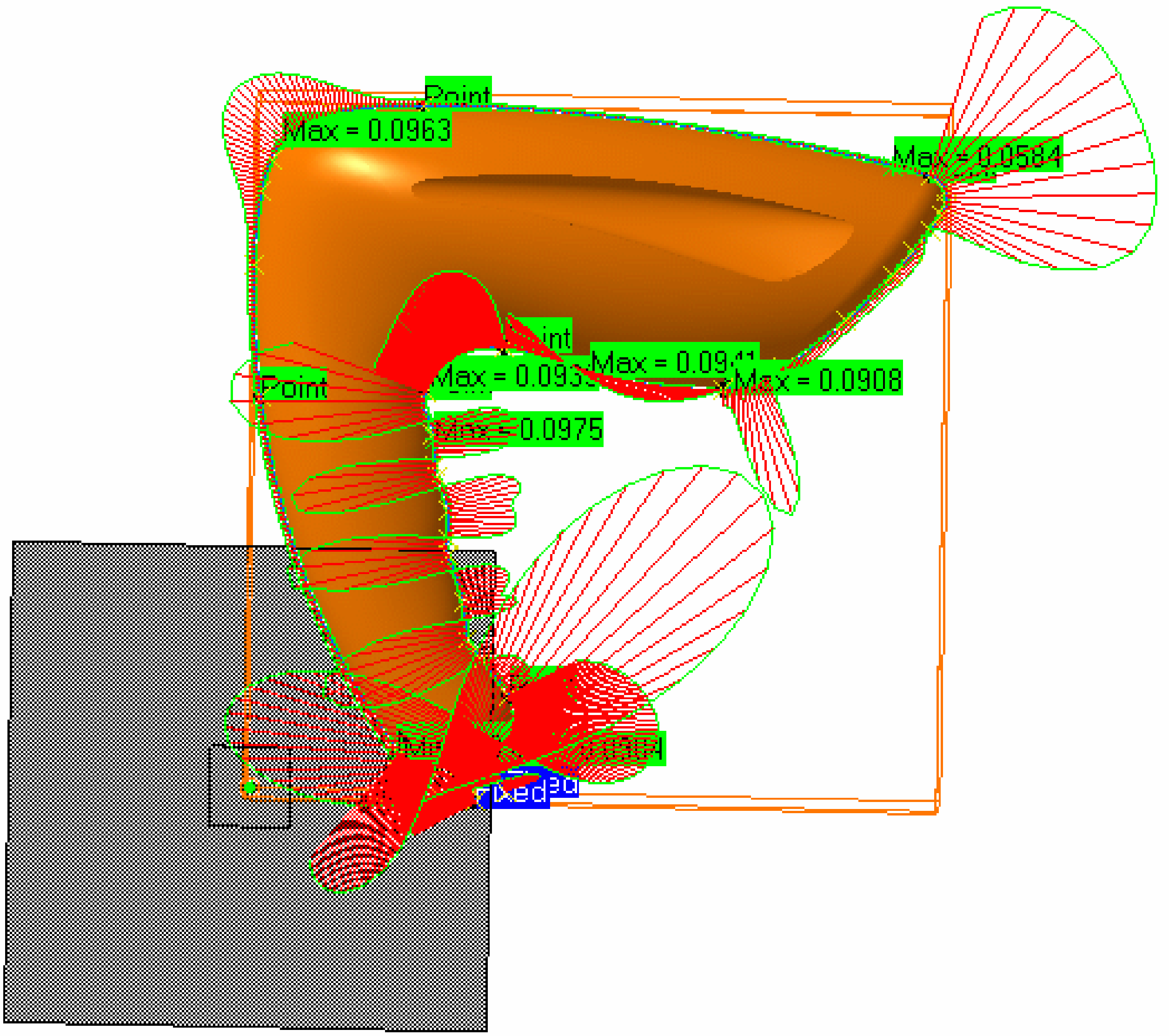
4. Set the parameters in the **Curve from Scans** dialog box to your needs and pick the requested points on the computed curve to split it into several smaller curves. The first segmentation proposed is erased and the splitting points are displayed.



5. Click **Apply** in the **Curve from Scans** dialog box: the curves are computed. Their segmentation is displayed as well as the gaps between the curves and the polygon. If necessary, pick on a green square to remove this split point, and pick another point of the scan to create a new split point. Click **Apply** to take those modifications into account.

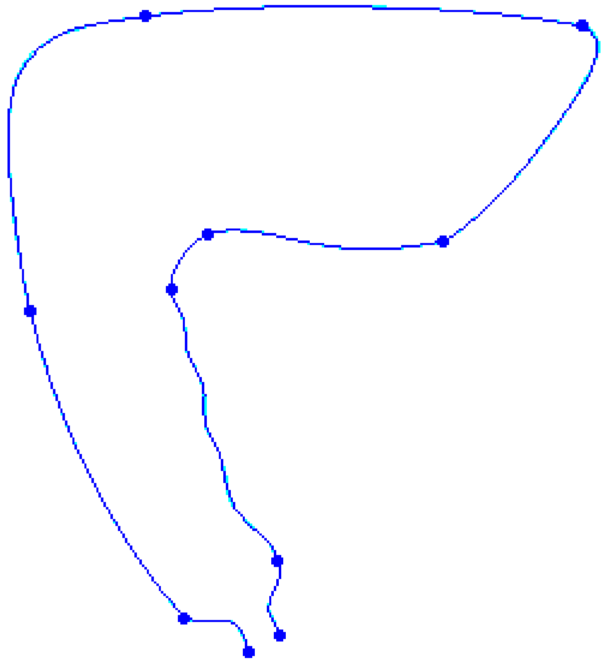


6. If necessary, check the **Curvature analysis** option to check the quality of the curves created.



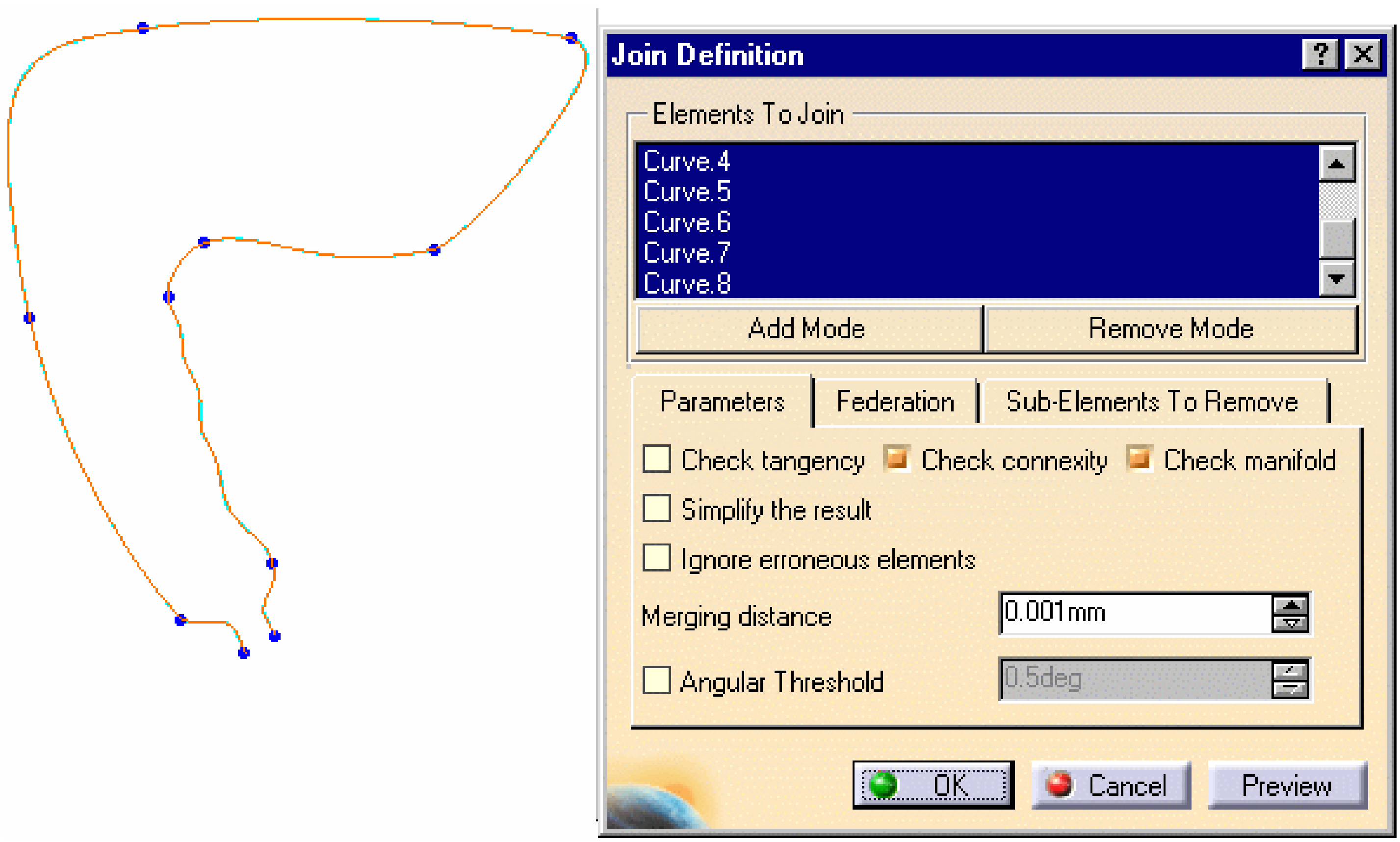
7. Once you are satisfied, click **OK** to validate the curves. The **Curve.1** to **Curve.10** elements are created in the specification tree.

8. If you wish, you can change the color of those curves using the **Edit/Properties** menu.

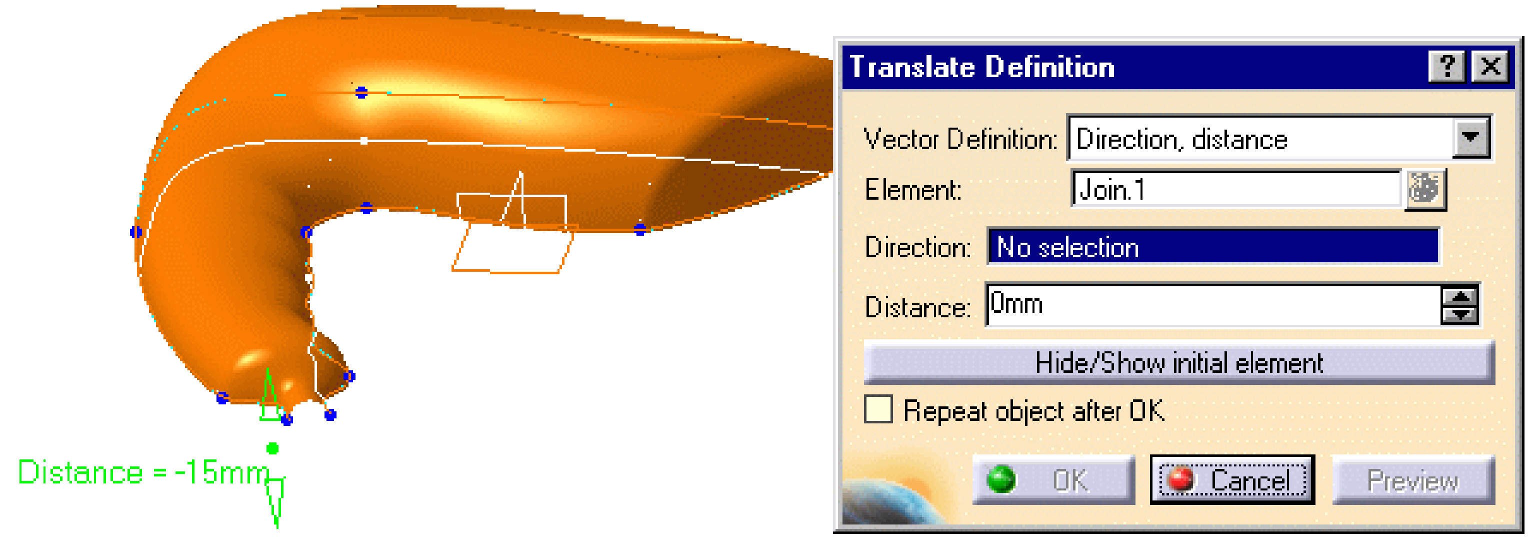


Create a tangent surface around the polygon:

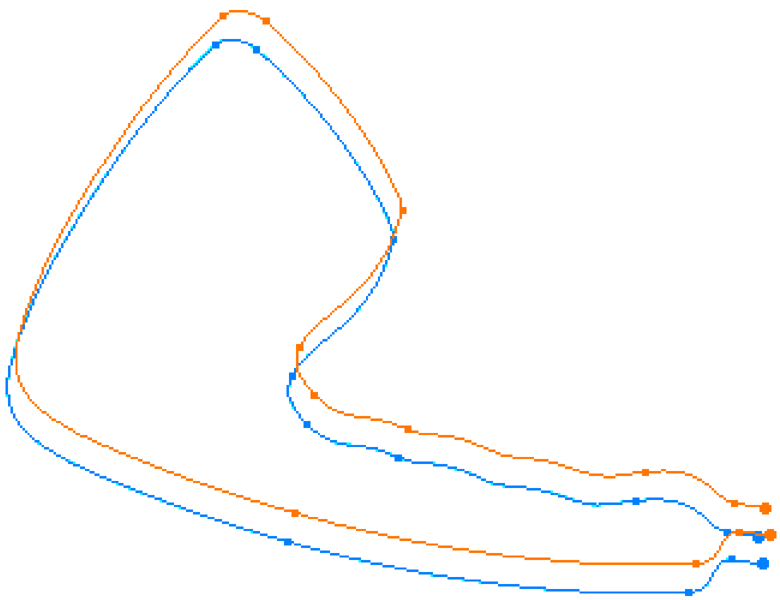
1. Click the **Join** icon and select the curves on the screen. Change the merging distance to 0.01mm. Click **OK**. The Join.1 element is created in the specification tree.

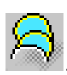


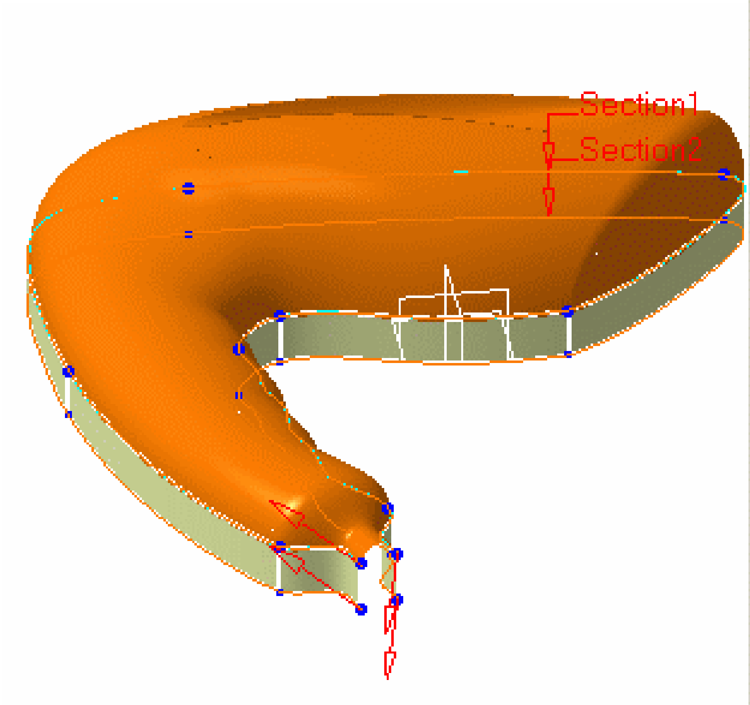
2. Click the **Translate** icon and select **Join**.1 Enter the xy plane for **Direction** and -15 mm for **Distance**.

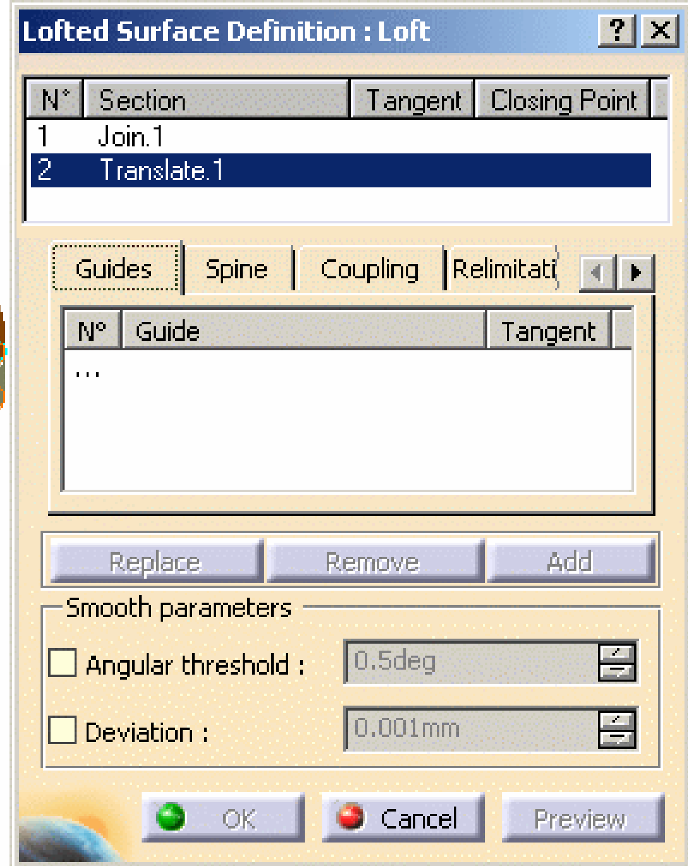


3. Click **Preview** and **OK**. The **Translate.1** element is created in the specification tree.

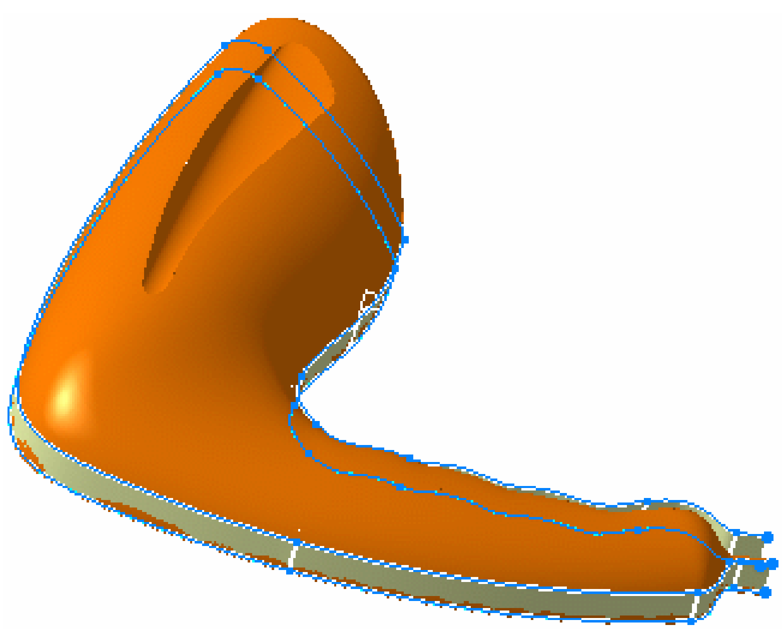


4. Click the **Loft Surface** icon . Select **Join.1** and **Translate.1**. Click **Preview** and **OK**.



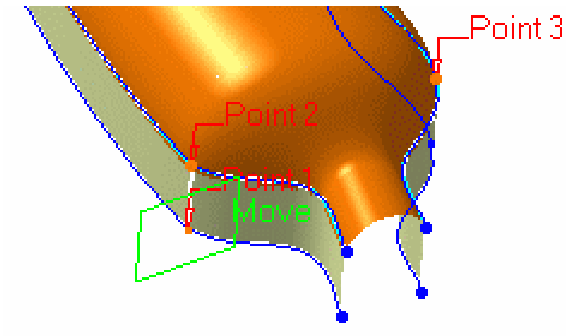


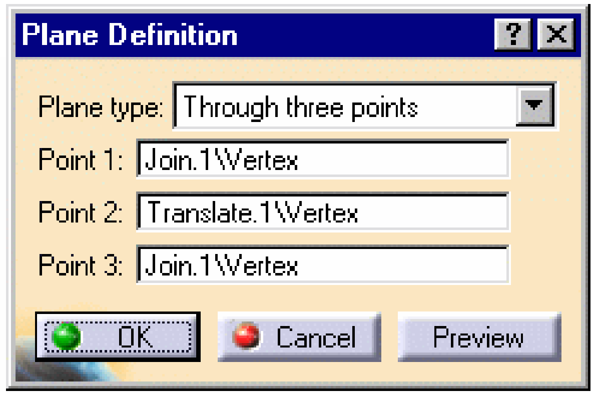
Loft.1 is created in the specification tree.

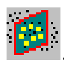
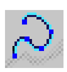


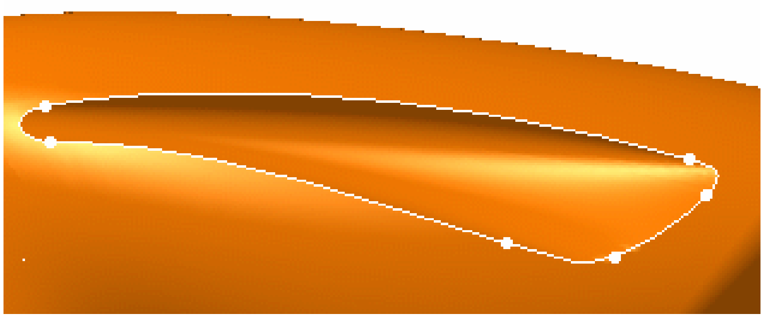
Define reconstruction zones on the polygon:

1. Use the Plane action from the WireFrame toolbar to create planes through 3 points using split points of step 1 and their counterpart on **Translate.1**:

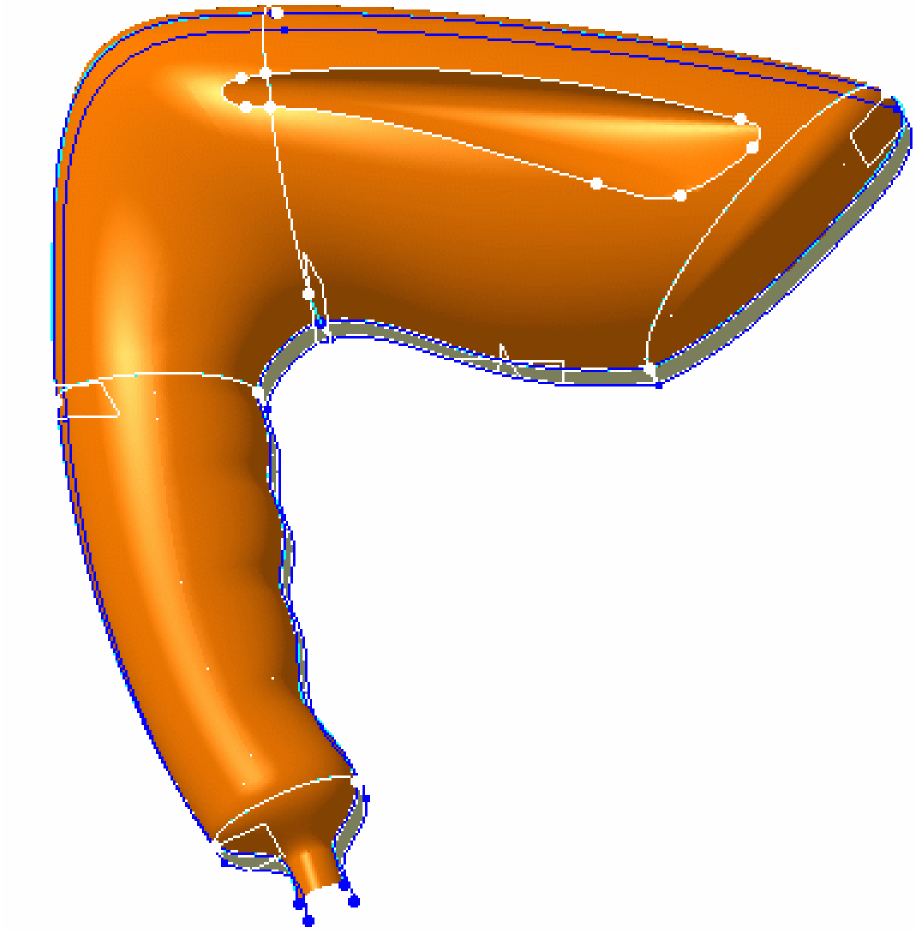




2. Click the **Planar Sections** icon . Select the polygon.
3. Click the **Plane** icon in the dialog box and select one of the plane you have created. Enter 1 in the **Number** field. Create a curve as above. Repeat this step for each plane. **Curve.11** to **Curve.16** are created in the specification tree.
4. Click the **3D Curve** icon  and create curves as follows:



3D Curve.16 to **3D Curve.21** are created in the specification tree.



Reconstruction of Surfaces

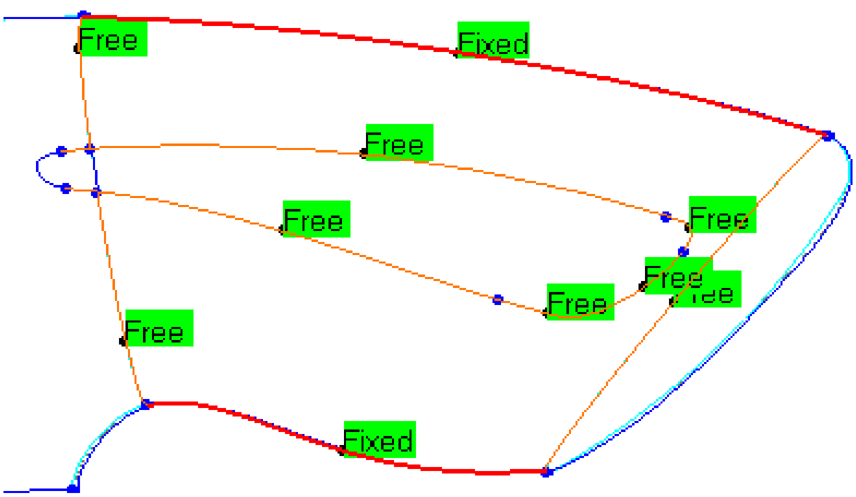
The first task will show you how to create a clean contour

To make selection easier, hide the following elements:

- the polygon
- the curves created to build the tangent surface.

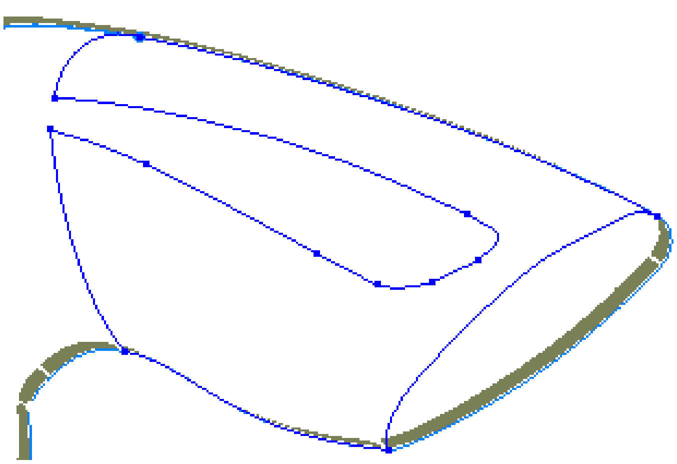
Create a clean contour:

1. Click the **Clean Contour** icon and select the following curves or edges:

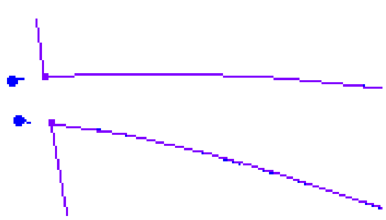


You see that the clean contour action takes the existing constraints into account: the edges of the surface are fixed.

Click **Apply** and **OK**. A **Join.2** element is created in the specification tree. The input curves have been sent to the NoShow.

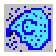


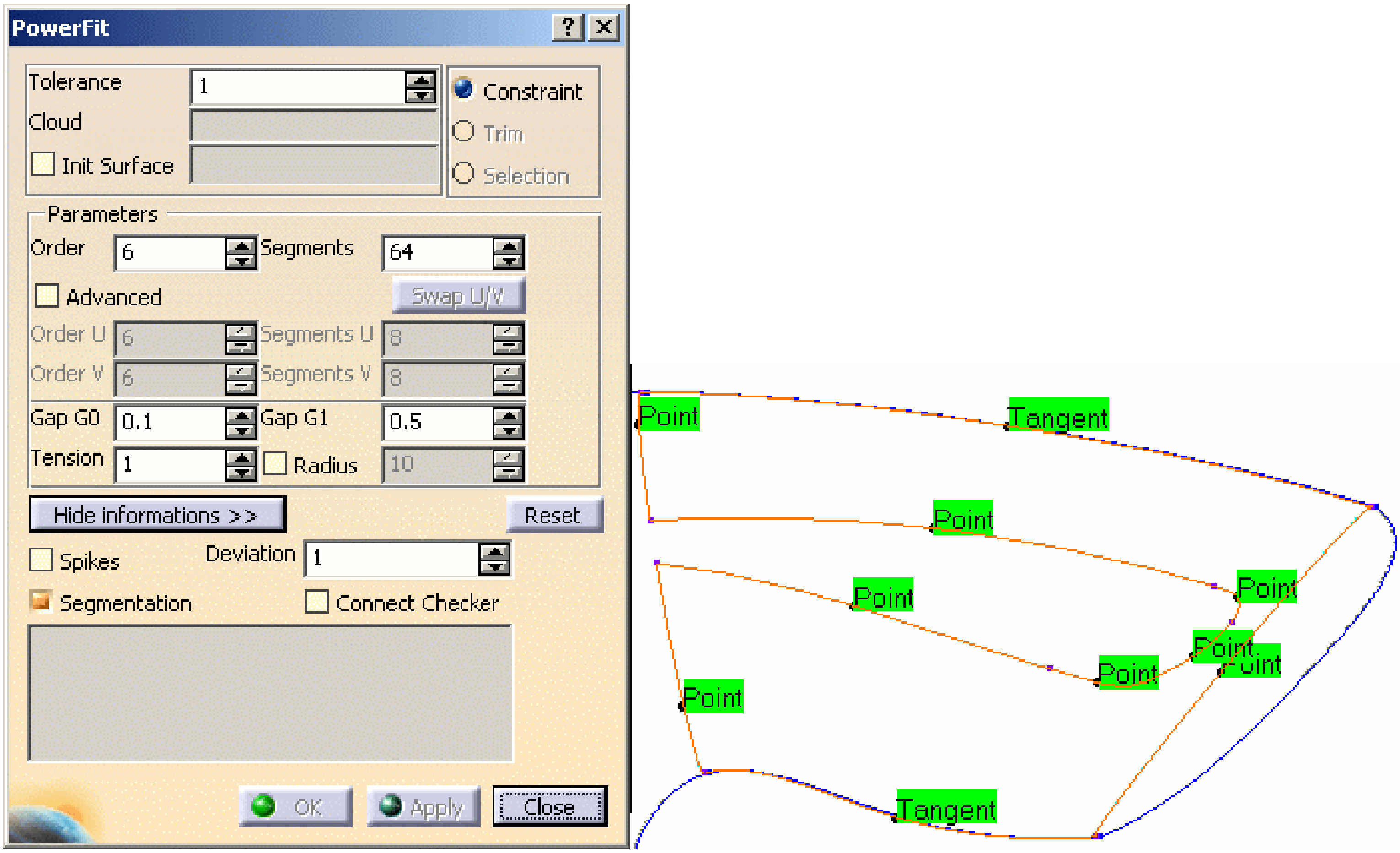
You see that curves have been trimmed.



2. Recall the polygon from the NoShow.

Reconstruct a surface:

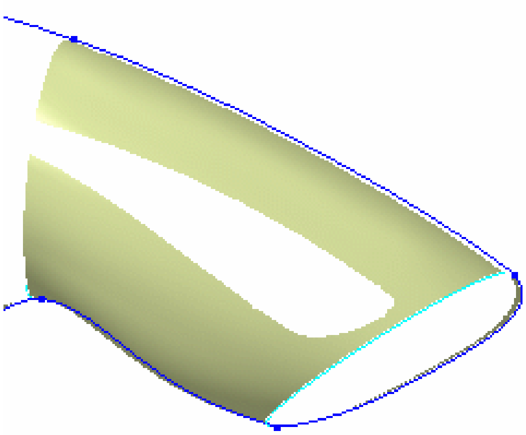
1. Click the **PowerFit** icon . Select the polygon. It is sent to the NoShow. Select **Join.2**.



You can see that existing constraints are again taken into account.

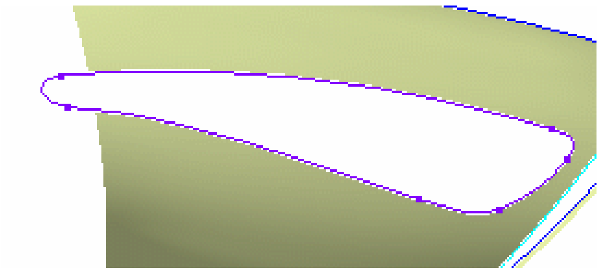
2. In the **Parameters** frame, check the **Radius** option and enter 10 as a radius value.

3. Click **Apply** and **OK**. A **Surface.1** element is created in the specification tree.



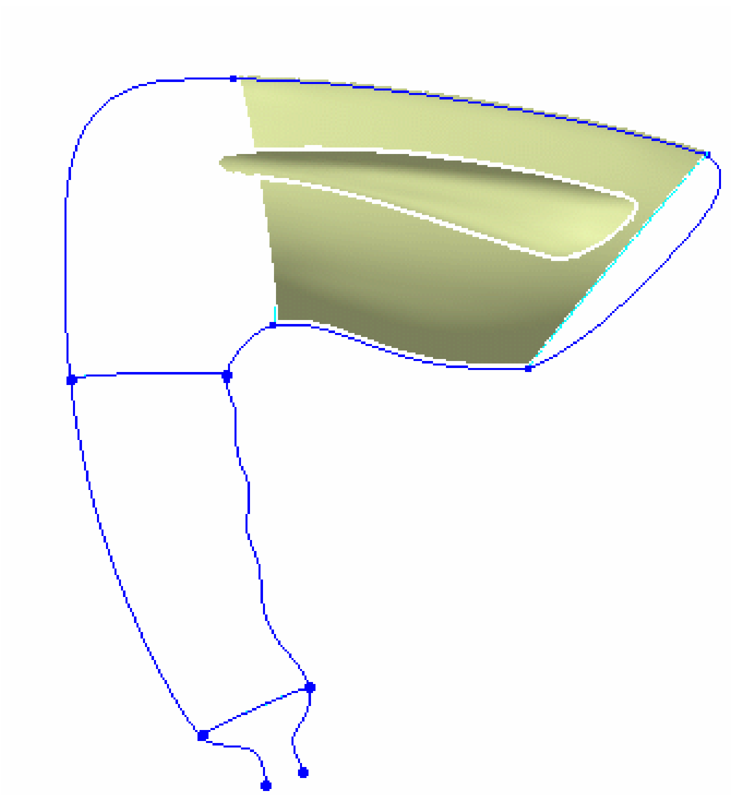
Reconstruct a second surface:

1. If necessary, recall hidden curves from the NoShow and create a clean contour as above. A **Join.3** element is created in the specification tree.



2. Recall the polygon from the NoShow.

3. Create a second PowerFit surface as above using the recently created clean contour.



You can reconstruct the other surfaces the same way.

User Tasks

Cloud Edition

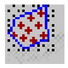
This chapter deals with the Activation of portions of clouds of points or polygons.

Activating a Portion of a Cloud of Points

This task shows how to select a portion of a cloud of points or of mesh in order to create a work area, either:

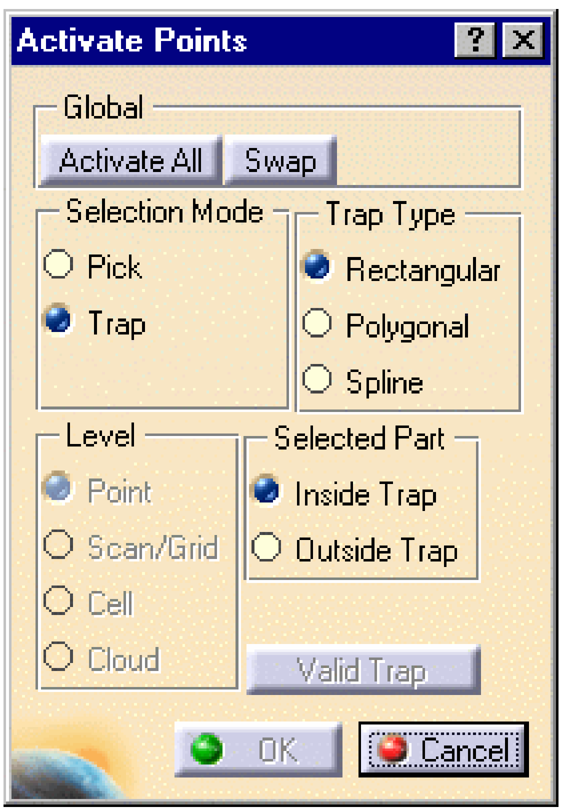
- by picking directly elements of the cloud (points, scans, grids, cells, clouds) or
- by defining a portion of the cloud or mesh with a 2D or 3D trap.

Open the Cloud.CATPart model the from the samples directory.

1. Click the **Activate** icon  and the cloud of points or mesh. The **Activate Points** dialog box is displayed.

The **Apply** button has been replaced by the **Valid Trap** button and the **OK** button:

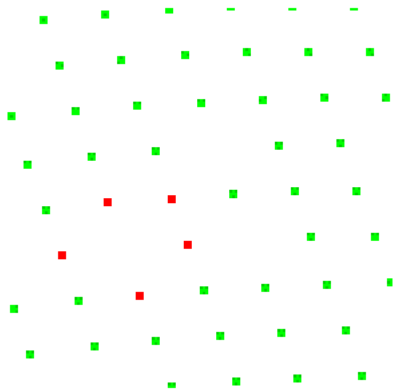
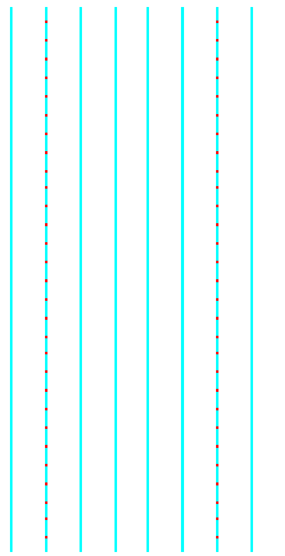
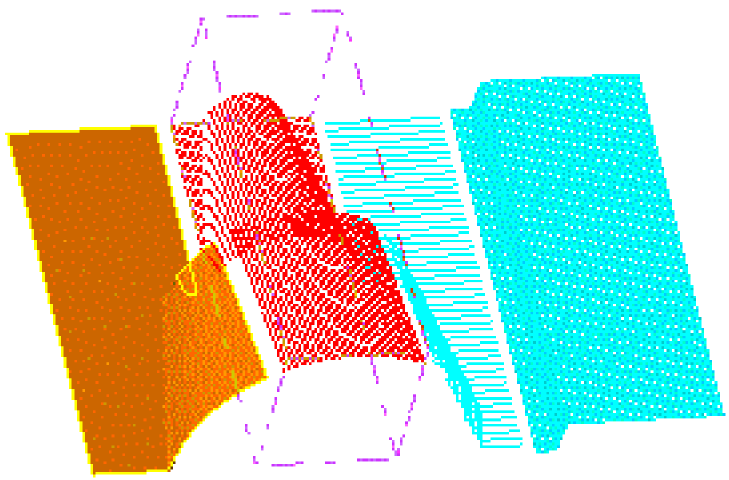
- To activate an area by picking elements, select them and click **OK** to confirm the activation and close the dialog box.
- To create a single activation area by trap, draw the trap, modify it when necessary and click **OK** to confirm the activation and close the dialog box.
- To create several activation areas by traps, draw the first trap, modify it when necessary, click **Valid Trap** to validate this trap. Repeat these steps for each trap then click **OK** to confirm the activation and close the dialog box.



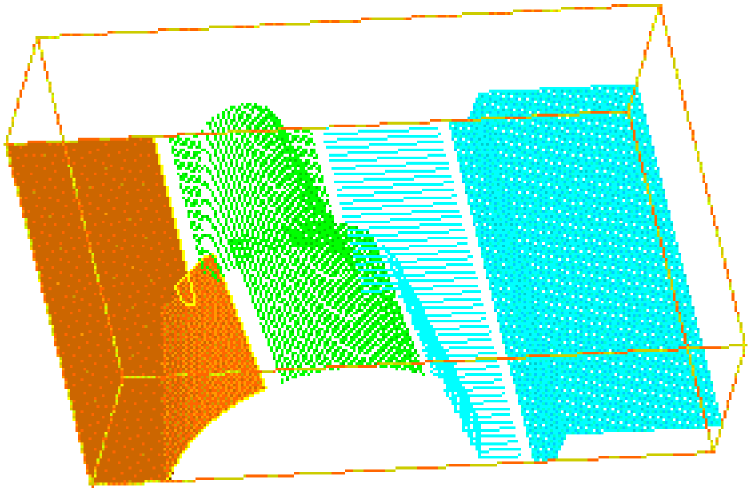
You can define :

- the **Selection Type**:

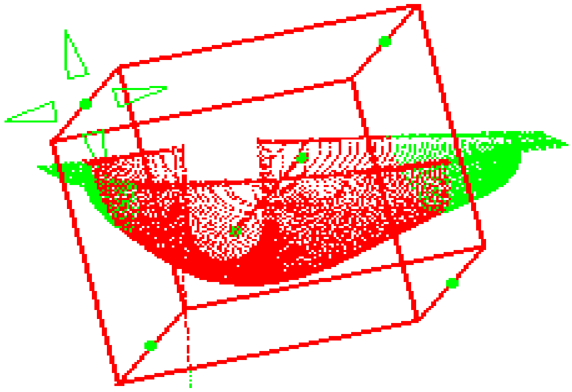
Pick: activation occurs by direct picking, according to the level of selection:

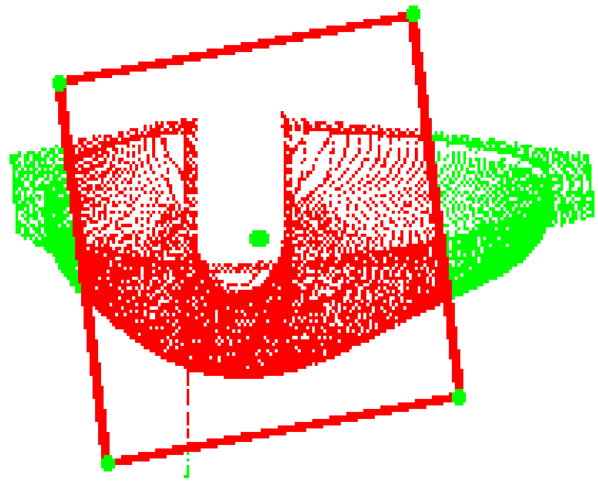
Point	
Scan or Grid	
Cell (sub-element of a cloud)	

Cloud (global cloud)

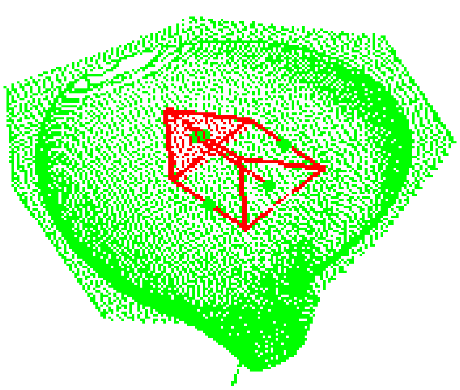


- **Trap**: the points are selected using a graphic trap:
  - either **Rectangular** or

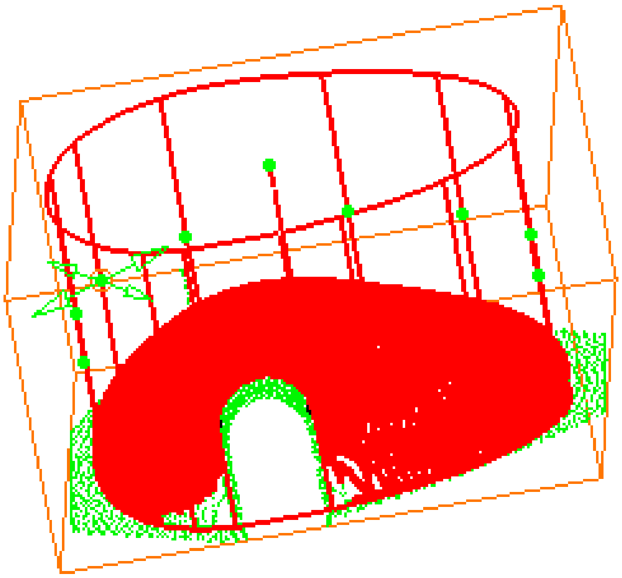
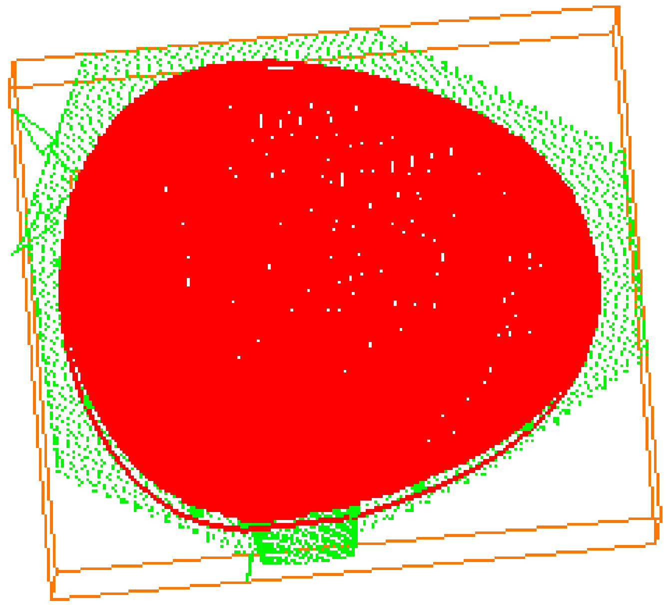




  - **Polygonal** or

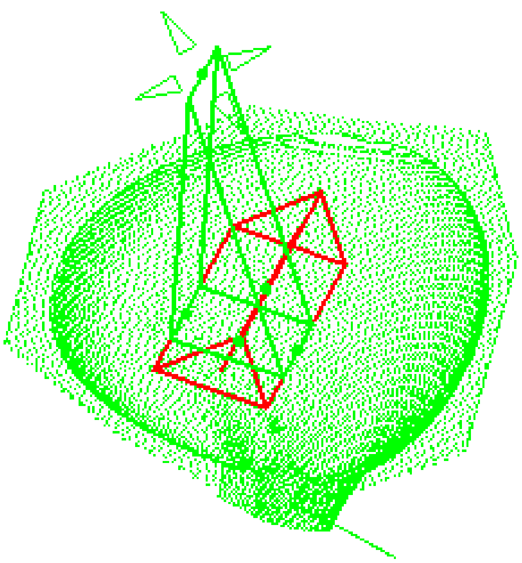


- **Spline.**

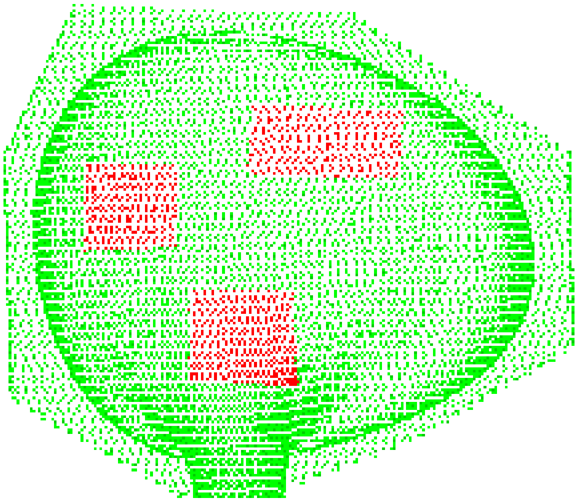
 

As long as you have not double-clicked to end the polygonal trap, you can undo/redo each pick of the polygonal trap.

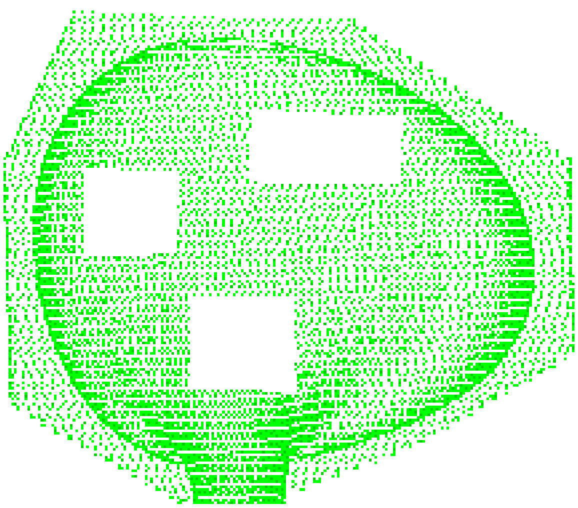
- The activated points are those that appear in red during the selection.
- By default, as shown above, the trap is displayed in the view plane. It is thus a 2D trap.
- You can rotate the model to display it as a 3D trap.
- You can modify the shape of the trap using the green arrows that appear on the edges of the trap.



- You can create as many areas as you wish by clicking **Valid Trap** and drawing another trap, until you click **OK** to validate and exit the action.

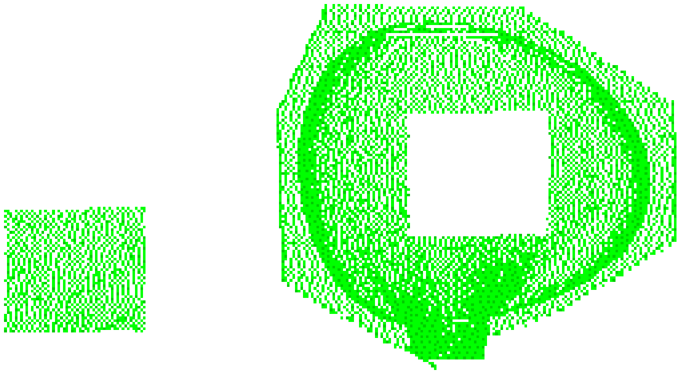


- If you draw a trap and click **Valid Trap**, and then choose **Swap**, you validate the complement of the trap.



2. You can choose to keep the points :

   - Inside the trap or
   - Outside the trap.



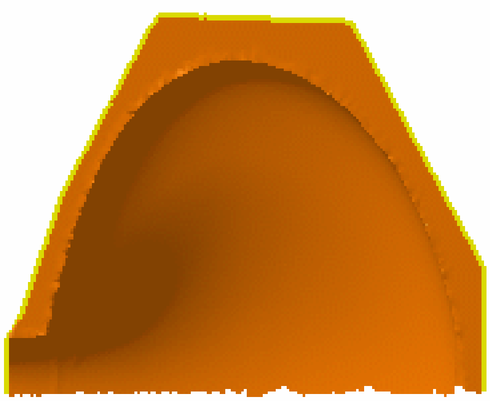
You can select an activated portion of a cloud of points or a mesh and recall the activation action to:

- **Activate all** (the whole cloud of points or mesh is re-activated),
- **Swap** the selection (the complement of the current selection becomes active whereas the current selection is hidden)

- The **Activation** action displays only triangles that are fully selected (i.e. the whole triangle is inside the selection trap, or all its vertices have been picked). If you select only one or two vertices of a triangle, or if the selection trap intersects the triangle, it is not displayed.
- As a consequence, when you push the button **Swap**, the triangles displayed are not the exact complement of the previous selection.

The free edges displayed are those of the complete mesh:

- if you activate only a portion of a mesh, the free edges of that portion are not displayed.



Scan and Curve Creation

You can use the following tools to create scans or curves:

Project Curves
Planar Sections
Create Free Edges
3D Curve or 3D Curve on scan
Curve from scans
Intersection
Projection

Refer to Selecting Using Multi-Output to find out how to display and manage the list of selected elements

Projecting Curves

This task shows how to project curves on clouds of points or meshes.

The action proposes options to:

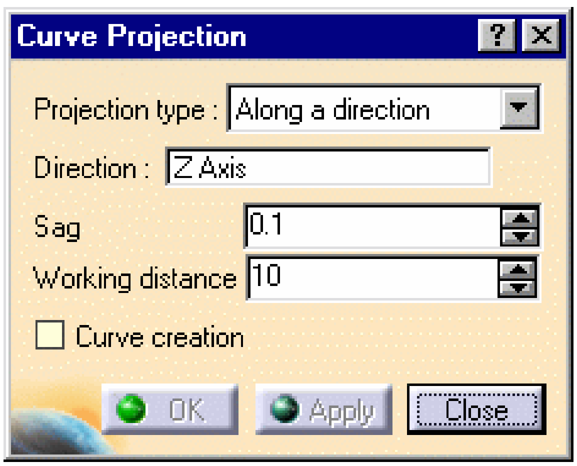
project the curves perpendicularly onto a mesh (not available for clouds of points),

modify the projection direction,

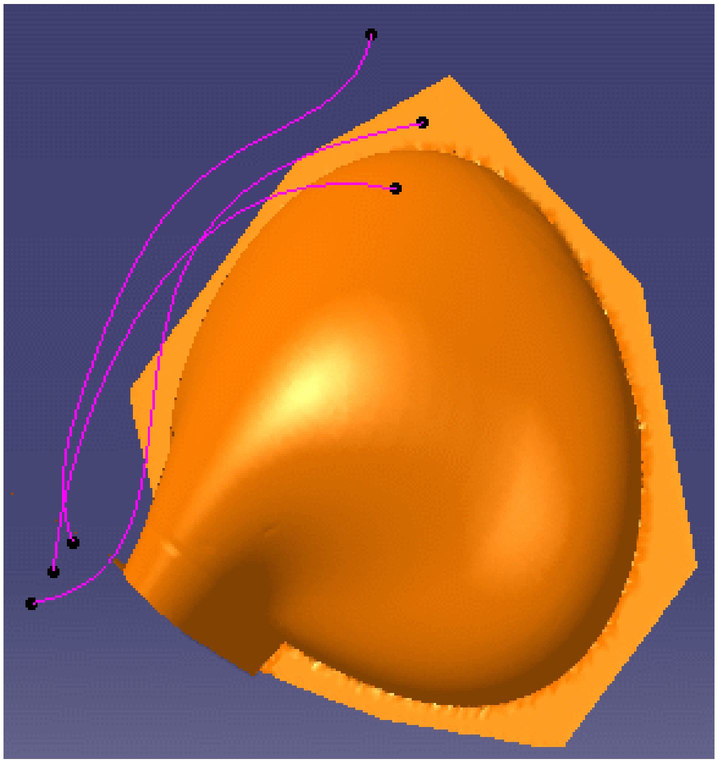
smooth computed scans directly into curves.

Open the CurvesOnCloud1.CATPart model the from the samples directory. You can use either the cloud or the mesh.

1. Click the **Project Curves** icon . The curve projection dialog box is displayed.



2. Select the curves to project and the target cloud or mesh.

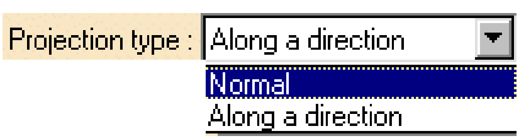


Selection of curves has been improved. Curves can be selected:

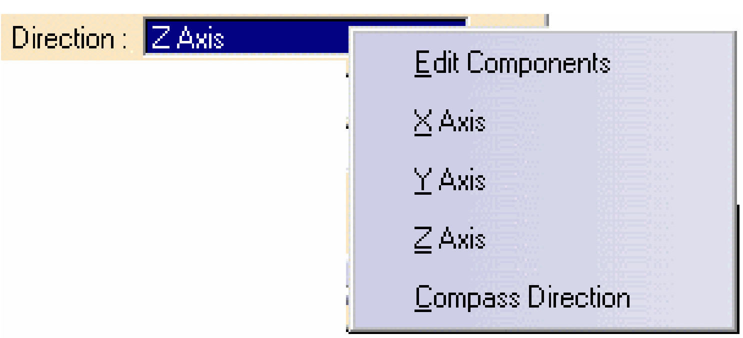
- from the specification tree,
- with a selection trap,
- with the preselection navigator.

3. If the target is a cloud of points, the projection is automatically computed along a direction.

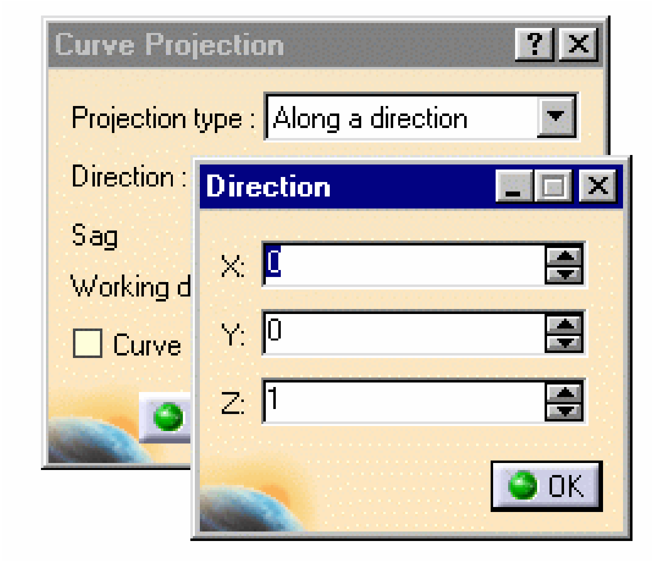
   If the target is a mesh, you can select the **Projection type** from the list:



4. If the projection is computed along a direction, the direction proposed by default is the Z axis. You can choose another direction, using the contextual menu of the **Direction** field:

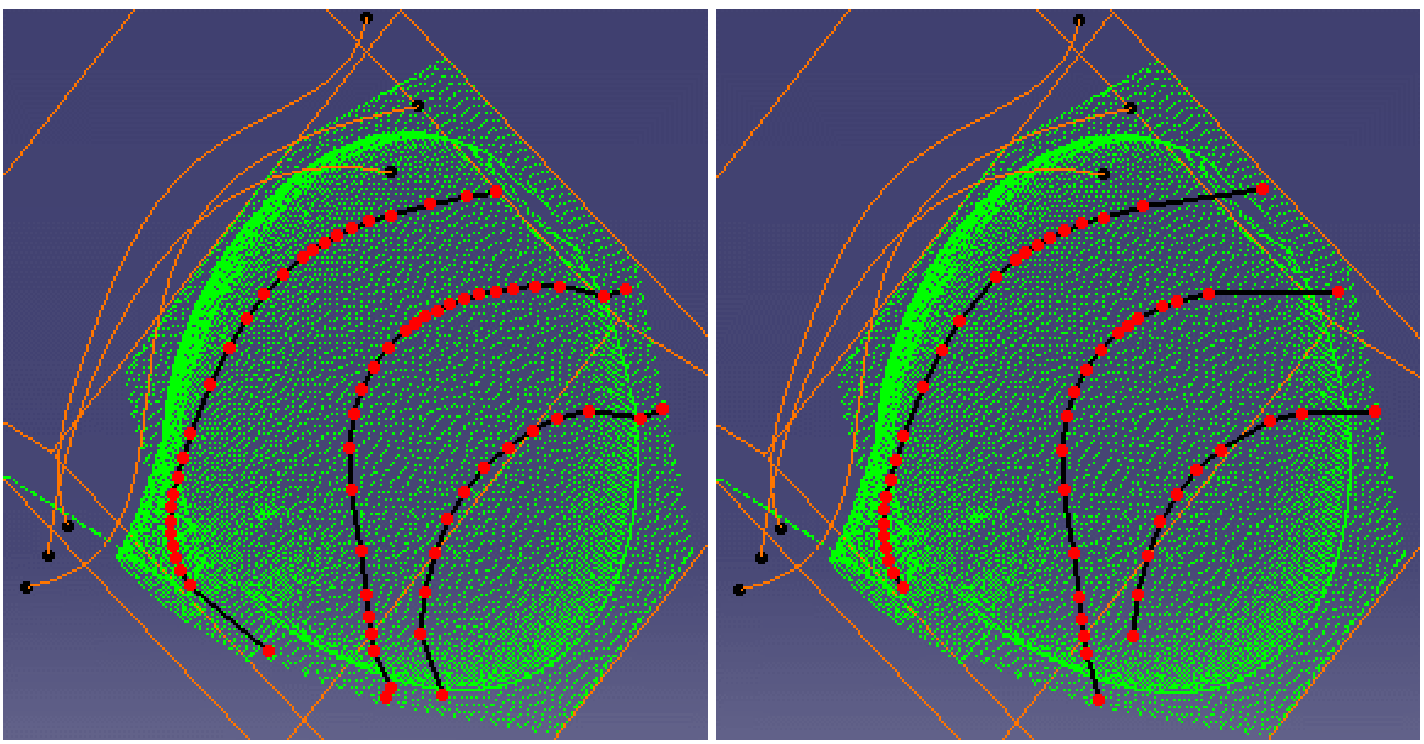


- the **Edit Components** option let you enter the coordinates of the direction:

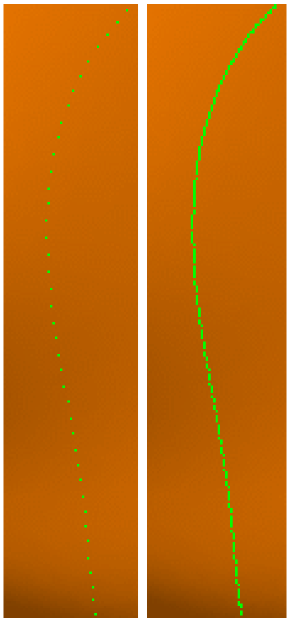


- The **Compass Direction** option takes the compass current orientation as the projection direction. If you want to change this direction, modify the compass orientation and re-select **Compass Direction** to take the new direction into account.
- otherwise, select one of the main axes as a direction.

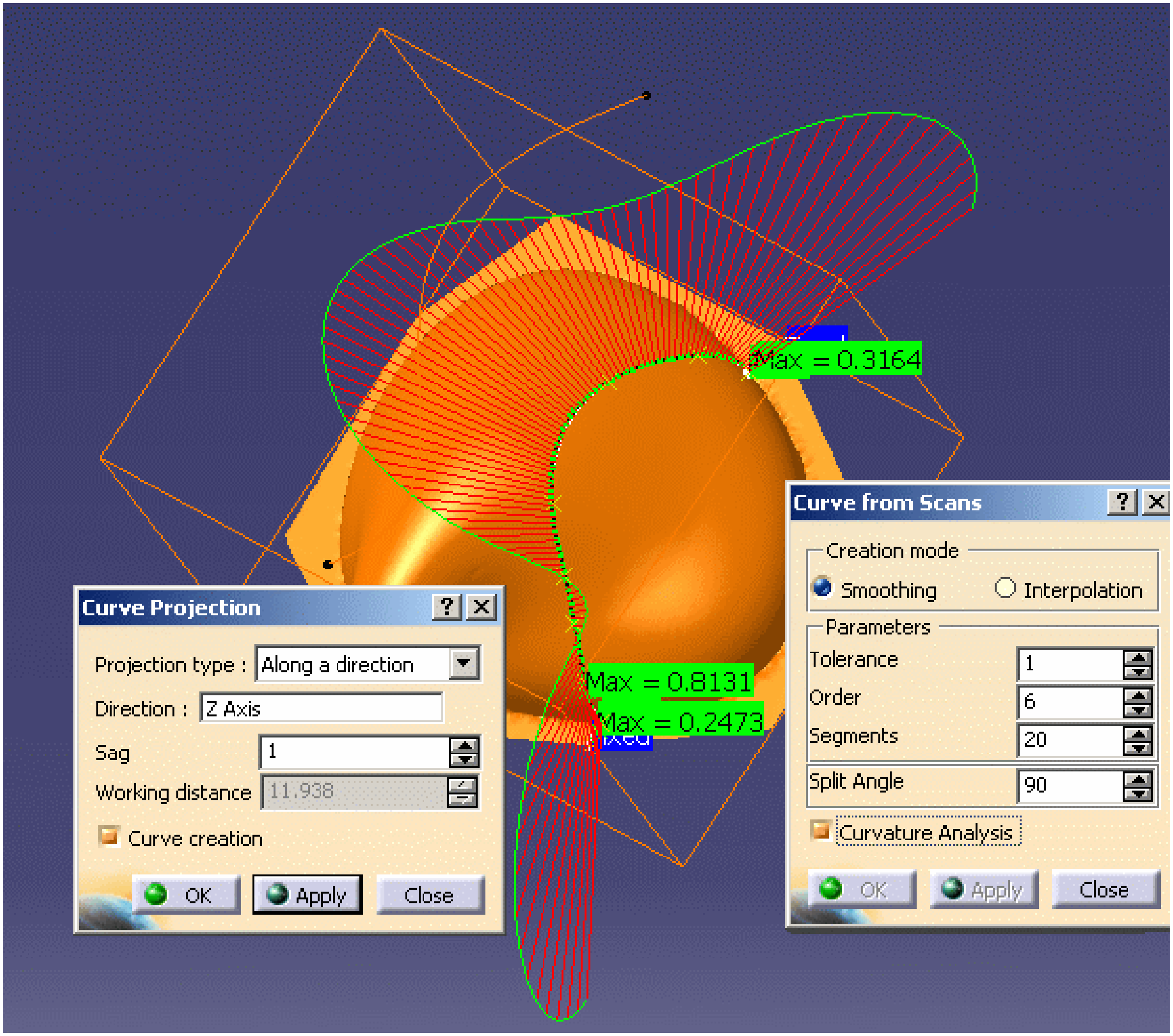
5. If the target is a cloud of points, you may set the **working distance**: the input curve is discretized, and each discretization point is projected on the cloud. The working distance is the distance taken into account around each projection point to compute the output scan. Enter 10 then 3 :



6. You can set a **sag** value: the curve to project is discretized according to this sag value, and each discretization point is projected on the mesh. In the example below, the sag was set to 1, then to 0.01:



7. If you want to create curves, check the **Curve creation** option. The operating mode is the same as for the **Curve from Scans** action except that only the curvature comb is displayed, instead of the complete curvature analysis.



- If the **Curve creation** option is checked, curves and only curves will be created.
- If the **Curve creation** option is not checked, scans and only scans will be created.
- If you need a complete curvature analysis of the curves you create, you have to create the scans first, and then create the curves with the **Curve from Scans** action.
- When you modify a parameter, click **Apply** in the corresponding dialog box to take it into account.

8. Click **Apply** to check or update the result. Then click **OK** to confirm the result and exit the action.

- Scans are created in the specification tree under the name **Curve Projection.x**.
- Curves are created in the specification tree under the name **Curve.x**.

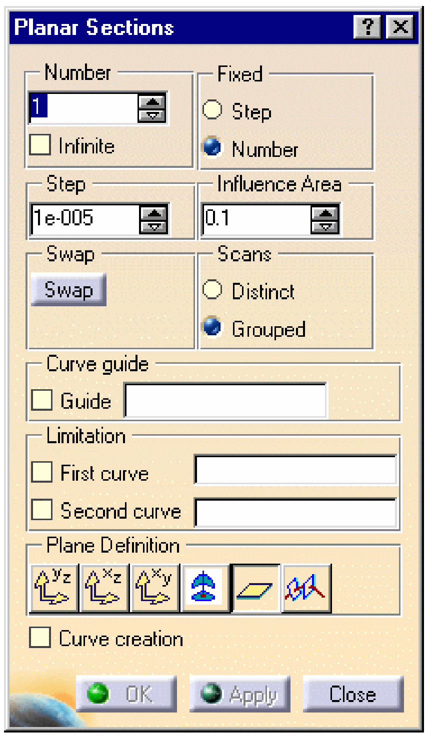
Cutting a Cloud of Points or a Mesh by Planar Sections

This task shows how to cut a cloud of points or a mesh by planes to compute scans and to smooth those scans directly to curves.

Open the Cloud.CATPart model the from the samples directory.

1. Click the **Planar Sections** icon and the cloud of points or the mesh.
   The **Planar Sections** dialog box is displayed with its default settings:
   - **Number** (of sections) is 1
   - **Step**: it is computed on L1 the largest dimension of the containment box, and is equal to L1/50 rounded to the best suitable value.
   - The plane of the section is parallel to the main plain of the compass. The section is distant from one step from the origin of the containment box.
   - The **Scans** option is set to **Grouped**.

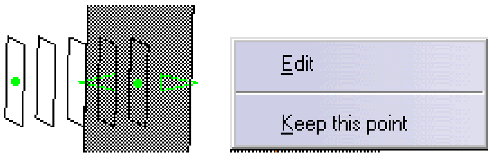
   The **Number** option is modal.



2. Use the **Plane Definition** icons to select the reference plane according to your need:

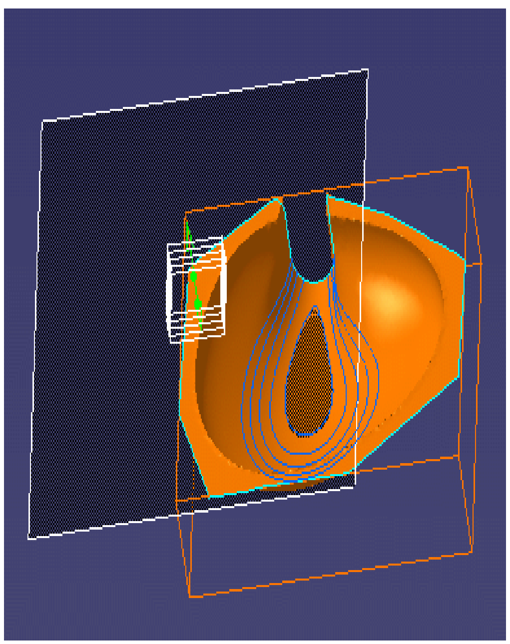
- push the YZ or XZ or XY icon to select a predefined plane, or
- push the compass icon to orient the reference plane with the compass, or
- push the plane icon to select an existing plane or

- One manipulator is available on the reference plane. It can be used to position the reference plane either by dragging the manipulator or by using the contextual menu **Edit**.



- Another manipulator is available on the last cutting planes proposed. This manipulator is used to modify either the step between cutting planes, or the number of planes, depending on the option selected in the **Fixed** field or in the contextual menu attached to this manipulator.

- Check the **Keep this point** option on any of those manipulators to create the corresponding point.



Push the section guide icon and select a curve: the sections will be perpendicular to this curve. The degree of the section guide must be greater than 2.

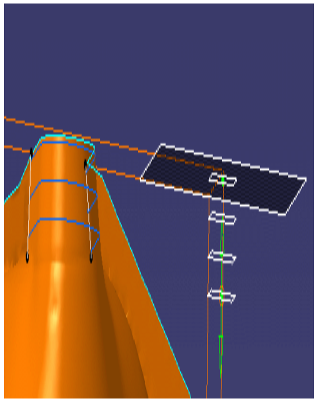
3. After setting the orientation of the reference plane, you can move it along its normal or along the section guide by dragging the center of the green manipulator in the required direction.

4. If required, you can select one or two limiting curves for any of the plane option. Pick the first limiting curve, its name is displayed in the **First curve** field. You can then select a second limiting curve. Its name is displayed in the **Second curve** field.

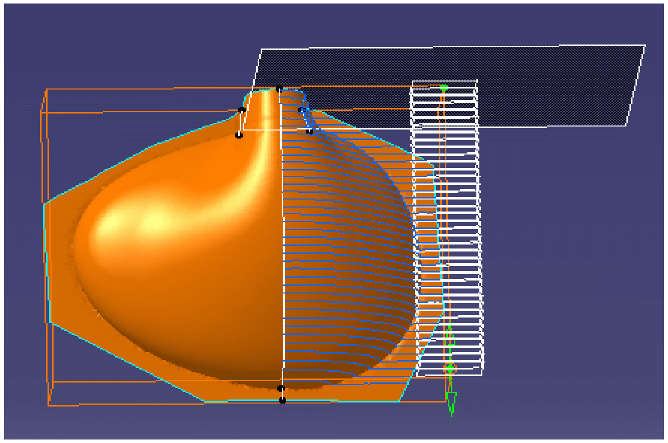
   The limiting curves should lay on the cloud of points or the mesh.

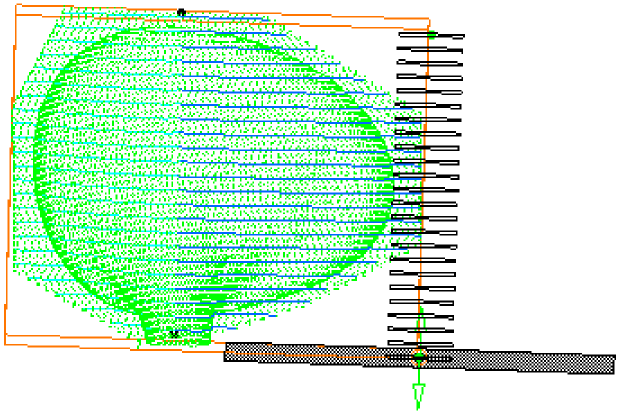
   The section guide curve can be selected as second limiting curve (not as the first).

   To replace a limiting curve by another, uncheck the corresponding field: the name is erased. Check the field again and select the new curve. Its name is displayed.



When using a limiting curve, the scans may be created on the "wrong" side of the curve. In fact, this side is determined by the origin of the reference plane. So move the reference plane to create the scans on the "right" side, either with the contextual **Edit** menu of the plane, or using the compass.





The computation of sections with limiting curves has been improved :

- You no longer need to check the **Lock Privileged Plane Orientation Parallel to Screen** option of the compass.
- But be careful to choose a view parallel to the screen.

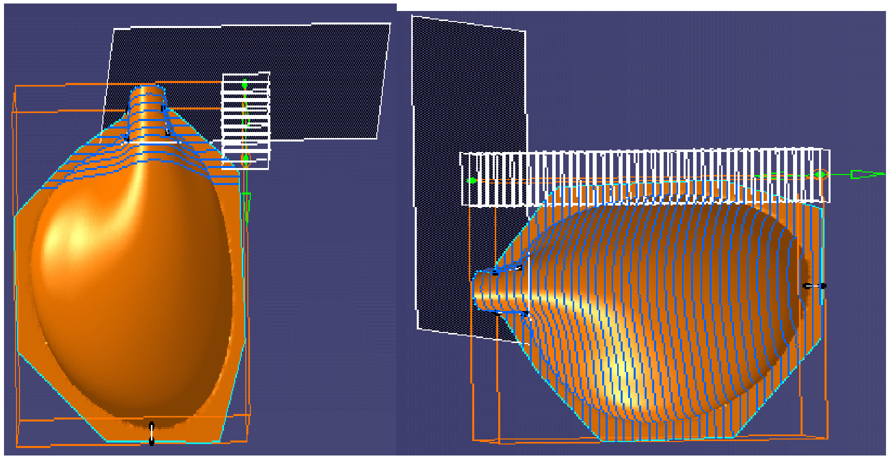
This enhancement enables you to define planar sections with the compass while using limiting curves.

For an easier selection of curves, you can use the pre-selection navigator (see the Infrastructure User Guide for more information).

5. Cutting planes can be defined :

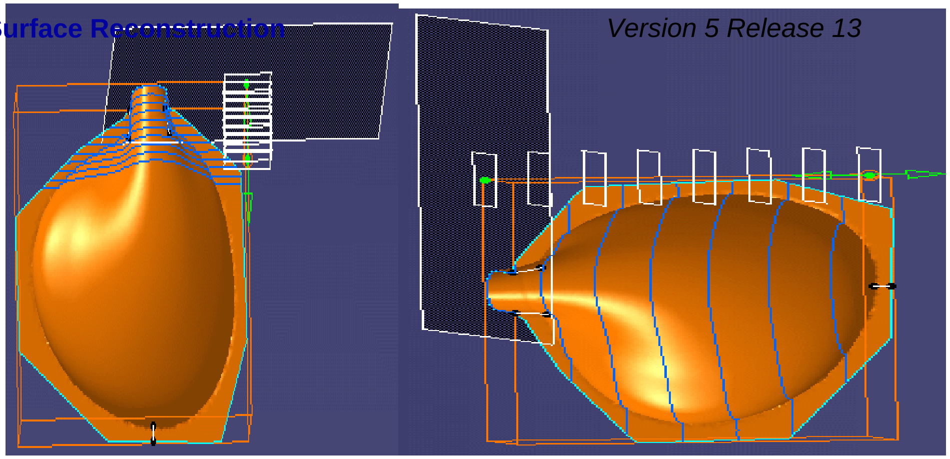
either from the step (distance) between two consecutives planes:

- check **Fixed, Step**,
- then enter the value of the **Step** in the dialog box,
- and enter the **Number** of planes in the dialog box or drag the green arrow until you reach the required number of planes (the dialog box is updated automatically).

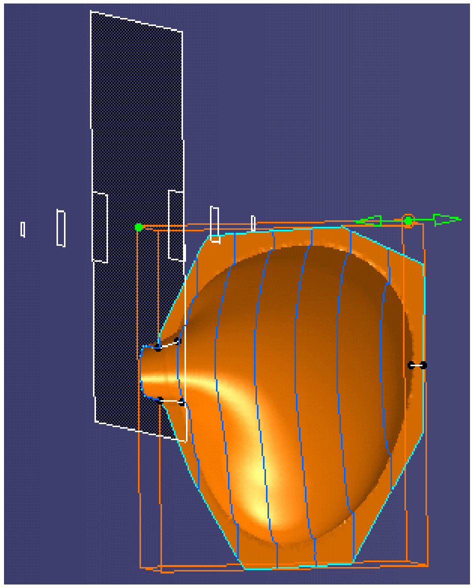


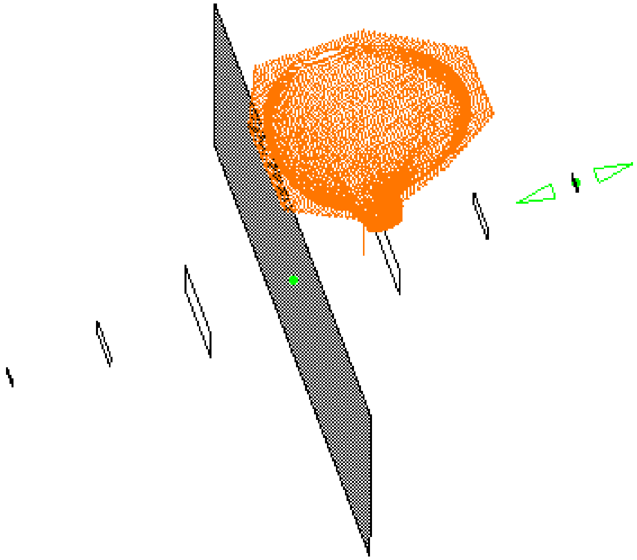
or from their number :

- check **Fixed, Number**,
- then enter the Number of planes in the dialog box,
- and enter the **Step** between two planes or drag the green arrow until you reach the required step (the dialog box is updated automatically).

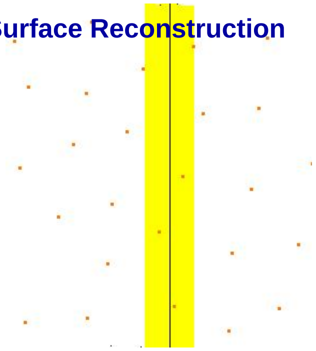


You can also check the **Infinite** option, especially when dealing with large models. In that case, the planes displayed on screen are used only to position the reference plane and define the step between two planes, if it is not fixed. The system computes all the cutting planes necessary to cut the whole model.



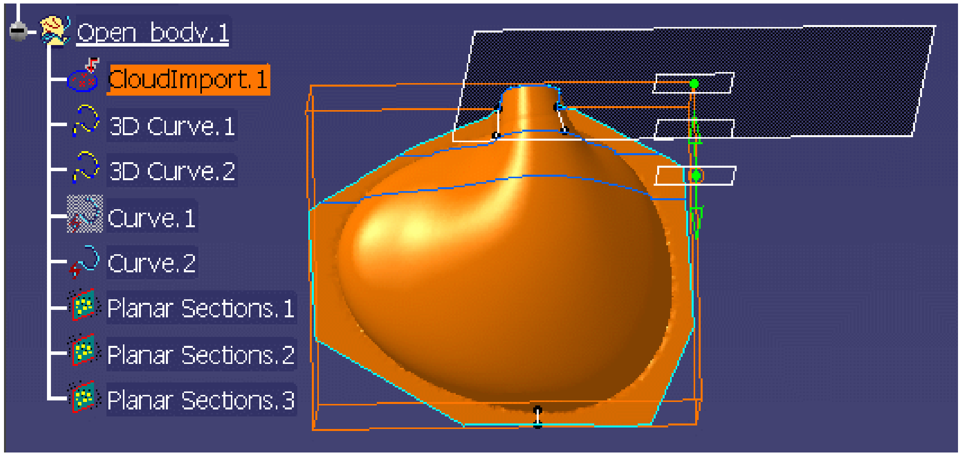


6. The **Influence Area** parameter defines a computation area around the cutting planes: when the points are not dense, a cutting plane (black line) may be unable to intersect the points. The **Influence area** is the area shown in yellow that contains the points considered to intersect the cutting plane. You can define its value according to your needs.

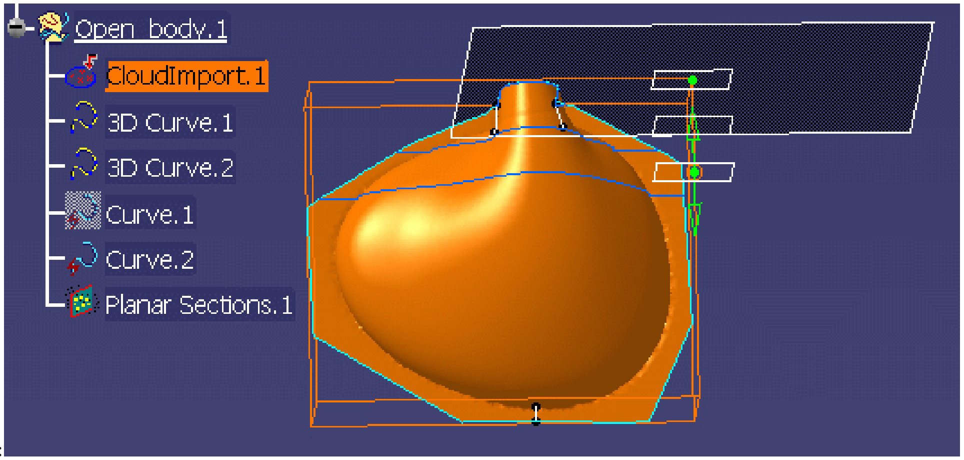


7. Select the type of the result scans: either :

Distinct: the intersections are created as as many Planar Sections elements in the specification tree.



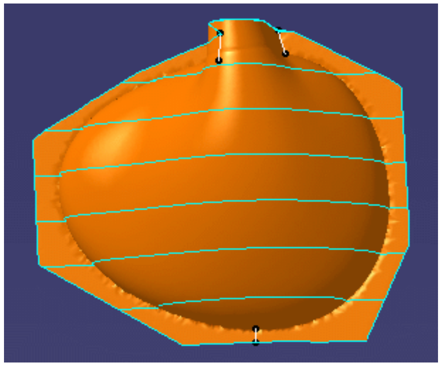
Grouped: the intersections are created as one cloud (one Planar Sections element) in the specification tree.



8. If you want to create curves, click **Apply** in the **Planar Sections** dialog box to display a scan in the specification tree and check the **Curve creation** option. The operating mode is the same as for the **Curve from Scans** action except that only the curvature comb is displayed, instead of the complete curvature analysis.

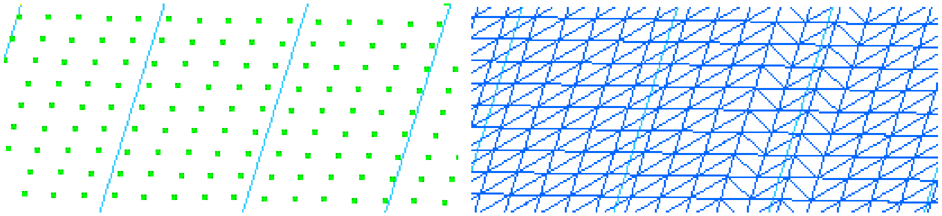
- If the **Curve creation** option is checked, curves and only curves will be created.
- If the **Curve creation** option is not checked, scans and only scans will be created.
- If you need a complete curvature analysis of the curves you create, you have to create the scans first, and then create the curves with the **Curve from Scans** action.
- When you modify a parameter, click **Apply** in the corresponding dialog box to take it into account.

9. Click **Apply** to check or update the result. Then click **OK** to confirm the result and exit the action.



Although cutting a cloud of points is quicker (no need to mesh first), creating planar sections on a mesh rather than on a cloud of points has some advantages:

- the action is dynamic on meshes: no need to apply to visualize the modifications (position of the reference plane, step, number of planes,...),
- In the case of a cloud of points, the intersection may be interpolated, since the plane does not necessarily intersect points. That problem is reduced with meshes since the plane intersects facets, providing a better accuracy.



- if you process an hybrid CloudsUnion element made of a mesh and a cloud of points, the planar sections will be created on the mesh only.

- The scans are created in the specification tree, as **Planar Sections.x**.
- The scans created are ordered.
- Scans can then be exported to an ASCII file.

Creating Free Edges

This task shows you how to create scans or curves by creating the free edges of a mesh.

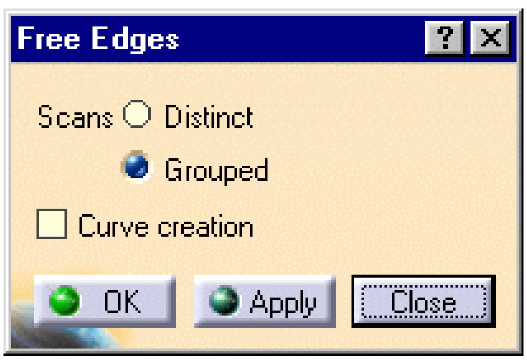
You can:

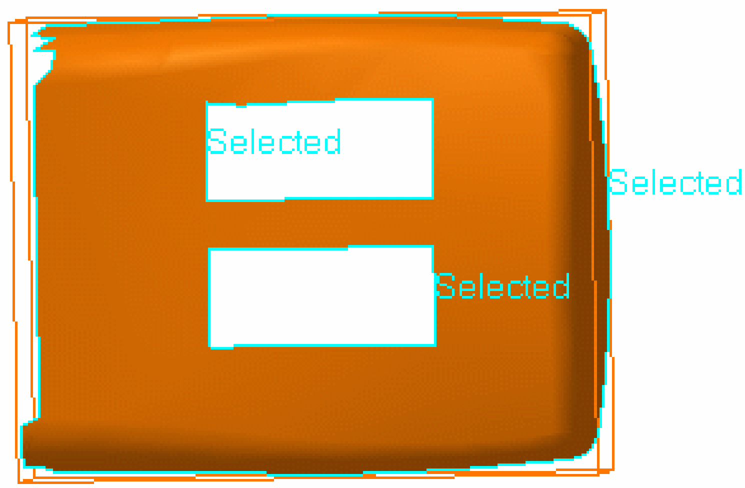
- create scans on all the existing free edges, or only selected free edges,
- create scans on the whole free edge or only a portion of it, and select which portion,
- create curves directly from these scans and check their curvature if required.

- This action is available for meshes only!
- This action is available on a complete mesh or on a portion of it .

Open the FreeEdges1.CATPart from the samples directory

1. Click the Create Free Edges icon and select a mesh. The dialog box is displayed and scans are proposed in cyan.



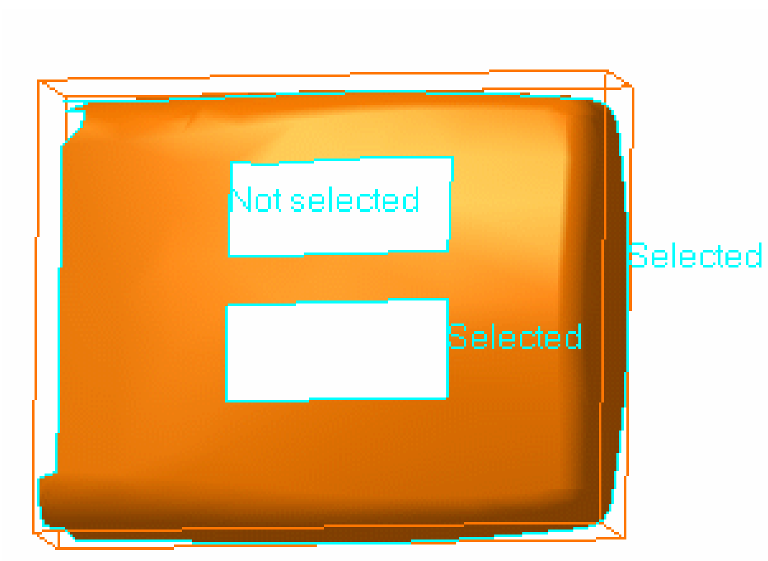


2. Select the scans to process.

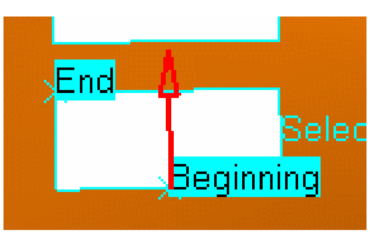
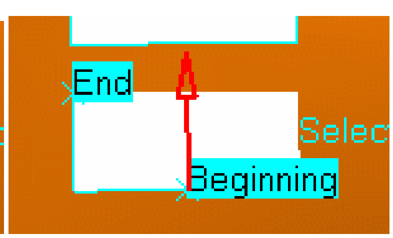
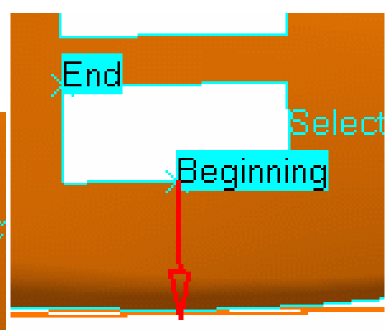
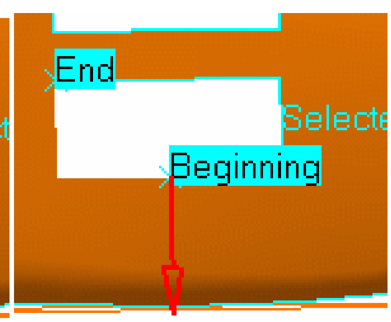
By default, all computed scans are proposed. Place the cursor on a label Selected and right-click to launch the contextual menu.

You can:

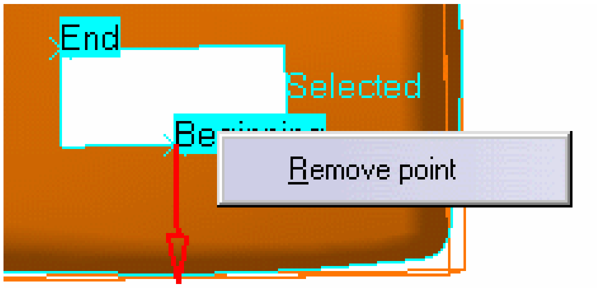
- Select or deselect singles scans,
- Select or deselect all scans.



3. If necessary, pick two points on a selected scan to define a portion of scan to create. Both extremities of the scan are displayed, together with its direction (as a red arrow). You can invert this direction by clicking on the red arrow.

Proposed direction, Before Apply	After Apply	Inverted direction, Before Apply	After Apply
			

4. To remove one extremity point, right-click on its label to start the contextual menu:



- Click Apply to apply any change you make.

- If you want to create scans, check the required option:
  - **Distinct** to create distinct scans,
  - **Grouped** to create one single scan.

- If you want to create curves, check the **Curve creation** option. The operating mode is the same as for the **Curve from Scans** action except that only the curvature comb is displayed, instead of the complete curvature analysis.

- If the **Curve creation** option is checked, curves and only curves will be created.
- If the **Curve creation** option is not checked, scans and only scans will be created.
- If you need a complete curvature analysis of the curves you create, you have to create the scans first, and then create the curves with the **Curve from Scans** action.
- When you modify a parameter, click **Apply** in the corresponding dialog box to take it into account.

Creating Associative 3D Curves

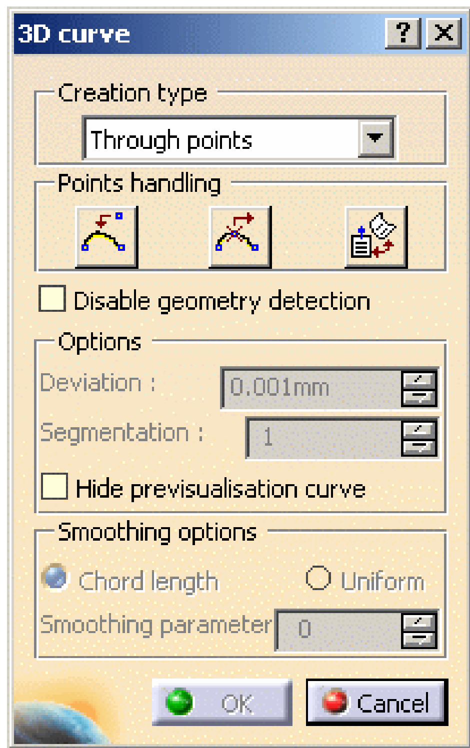
This task explains how to create a 3D curve that is associative meaning you can add or delete points (whether control points or passing points) both at creation time or when editing.
These curves can be created in space or lie on a geometrical element, or both. When the curve lie on a geometrical element and the later is modified, the curve is updated automatically, provided you choose the **Automatic** update option in **Tools -> Options -> Mechanical Design -> Assembly Design -> General** tab.

- Selecting 3D points
- Editing
- Keeping a point
- Imposing a Tangency Constraint
- Imposing a Curvature Constraint
- Setting as Arc Limit
- Removing a Point
- Constraining a Point

Open a new .CATPart document.

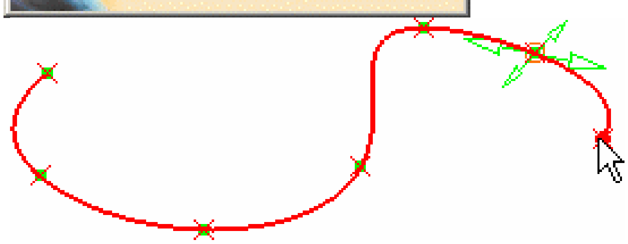
1. Click the **3D Curve** icon .

   The 3D curve dialog box is displayed.



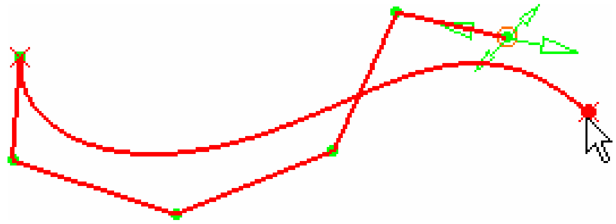
2. Choose the curve creation type.

   Whatever the chosen type, the curve is previewed in dotted line as you move the pointer.

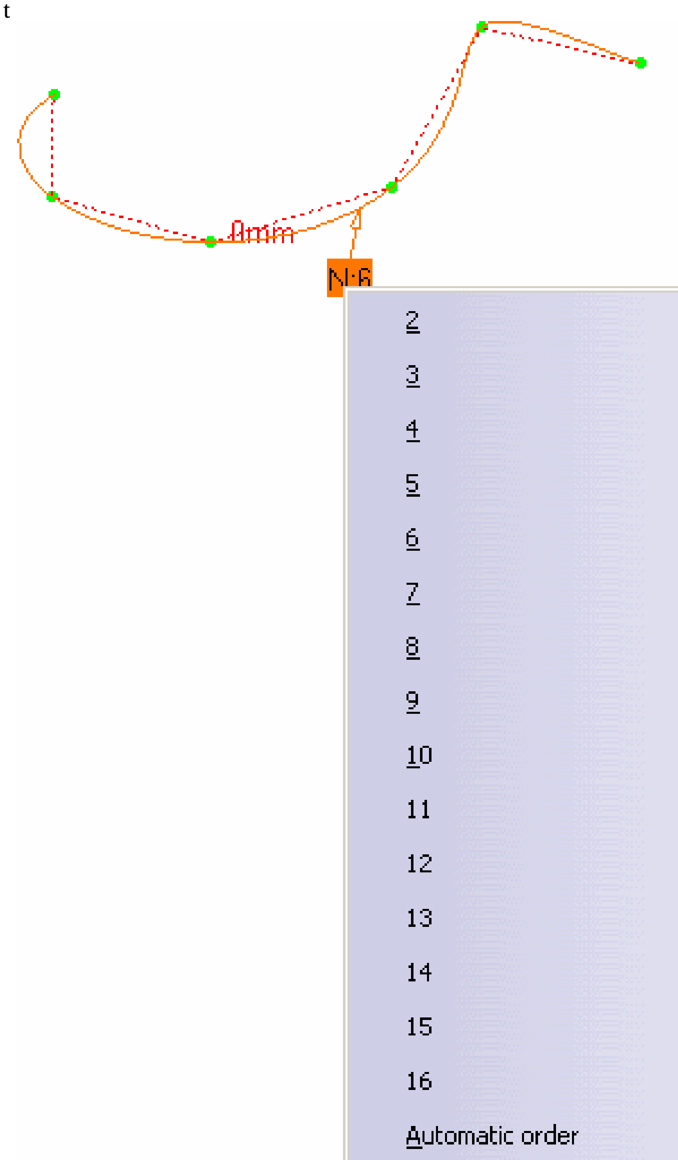


- **Through points**: the resulting curve is a multi-arc curve passing through each selected point.

- **Control points**: the points you click are the control points of the resulting curve

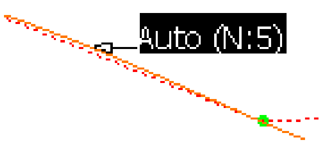


- **Near points**: the resulting curve is a single-arc, with a set degree and smoothed through the selected points.



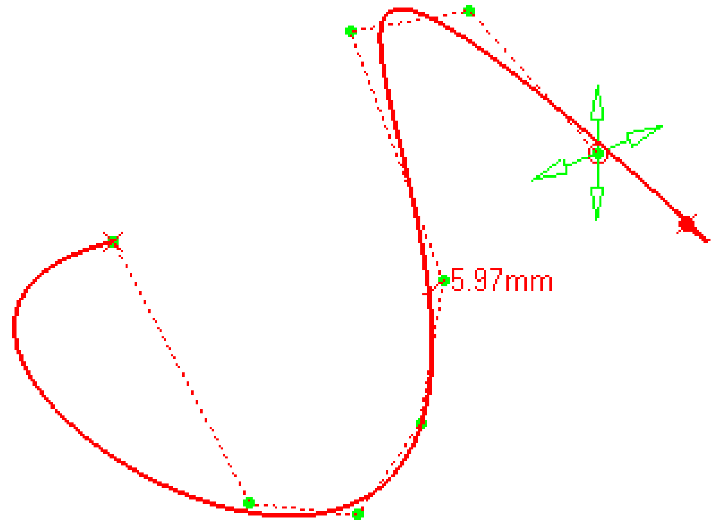
You can edit the order by right-clicking the displayed text (displayed using the U, V Orders icon  from the FreeStyle Dashboard or the **Order** option from **Tools -> Options** menu, **Shape -> FreeStyle -> General** tab), and choosing a new order value.

The **Automatic order** option enables you to automatically compute an order that will respect at best all the curve constraints.
The computed value is displayed near the Auto tag.



- The **Deviation** option enables the user to set the maximum deviation between the curve and the construction points.

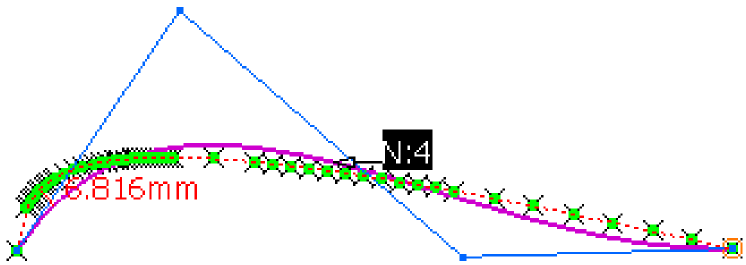
The result is a set degree through the selected points.



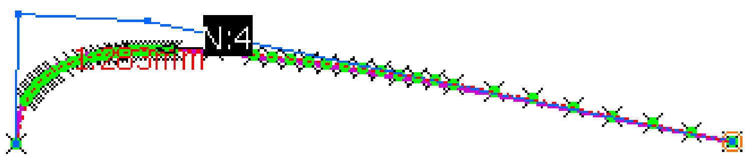
- The **Segmentation** option enables the user to set the maximum number of arc limits. These arcs are construction points and are inserted into the curve automatically.
  The minimum value is set to 1.

- Smoothing options are now available to parameterize the curve:

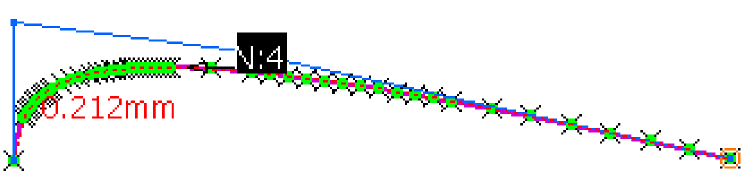
  - **Chord Length** (default parameterization)
    Smoothing parameter = 0

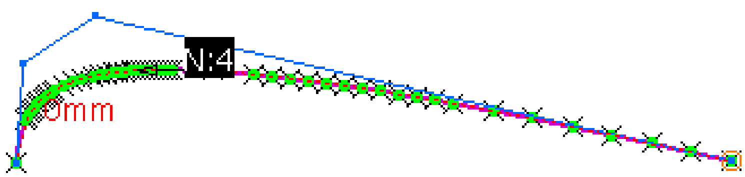


  - **Uniform**
    Smoothing parameter = 0



  - **Smoothing parameter**: enable a better control points distribution of the smoothed curve.





Smoothing parameter = 50 *Smoothing parameter = 130*

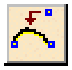
Deviation, **Segmentation**, and **Smoothing** options are only available for the Near Points creation type.

3. Move the pointer over a point.

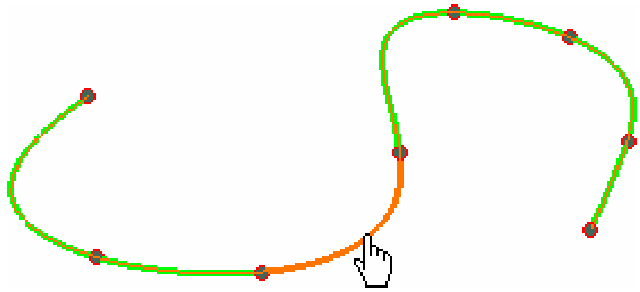
   A manipulator is displayed allowing you to modify point location as you create the curve.
   By default, this manipulator is on the last created point.

   A contextual menu proposes several options to construct the 3D curve.

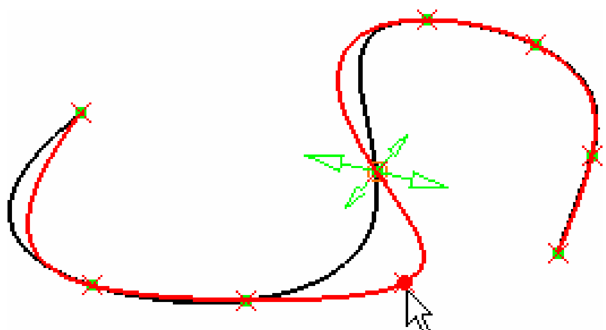
Right-click on the manipulator to display the contextual menu. From then on you can choose the **Edit** item to display the Tuner dialog box and enter space coordinates for the selected point, or choose the **Impose Tangency** item to set a tangency constraint on the curve at this point.

4. Click the **Insert a point** icon  within the dialog box.

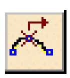
   The curve freezes.



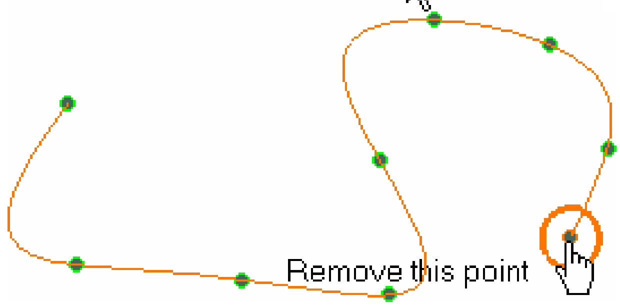
5. Click the segment, between two existing points where you wish to add a new point and click the point location.



   Once the point has been created, you are back to the edition capabilities on the curve.

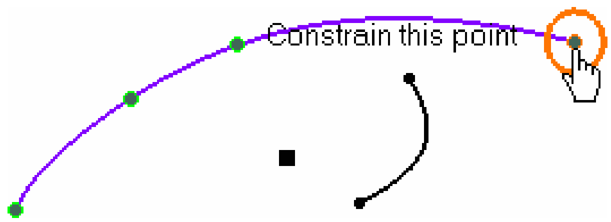
6. Click the **Remove a point** icon  within the dialog box, and select one of the existing points.

   The curve is recomputed immediately without the selected point.

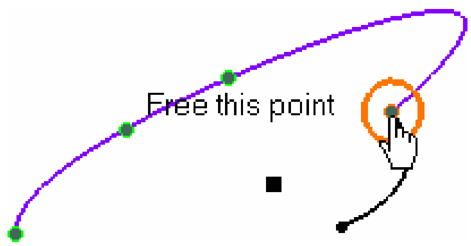


7. Click the **Free or constrain a point** icon  within the dialog box, then select the point.

- If the point is a point in space (free), move the pointer close to the point or a wire to which it should be linked. You can then move the pointer over a geometric element and:
  - move the point to the indicated point by clicking
  - press and hold the Control key (Ctrl) to project this point onto this element according to the shortest distance from the point initial location.



- If the point was lying on another point or a wire (curve, line, spline, and so forth), it is freed from its constraint onto this element, and can be moved to any new location in space.



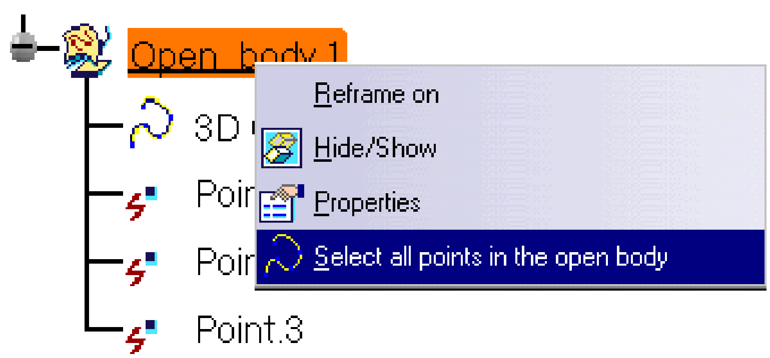
You can snap a point onto a surface using the **Free or constrain a point** icon. The point will be lying onto the surface, but not constrained. It can be moved using the manipulators.

8. Click OK to create the curve.

   A 3DCurve.xxx appears in the specification tree.

- Use the F5 key to move the manipulators into a different plane of the compass. See Managing the Compass.
- You cannot add a point past the end points. To do this, you need to add a point before the end point, move the new point where the end point lies, then move the end point to a new location.
- The creation plane for each free point is defined according to the current plane/compass orientation on the previous point. Therefore you can change creation planes within the same curve, by setting a new current plane/compass orientation on several points.
- Check the **Disable geometry detection** button, when you need to create a point close to a geometric element yet without constraining it onto the existing geometry.
- Check the **Hide previsualisation curve** to hide the previsualisation curve you are creating.

Selecting 3D points

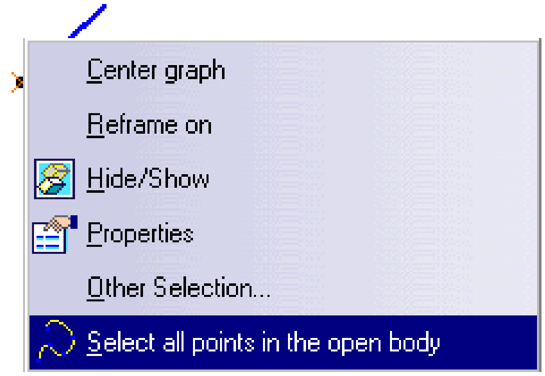


It is possible to select a scan of cloud either in the specification tree or directly in the geometry.

The Select all points contextual menu is available within the 3D curve action only, i.e. it is not available when you select the scan before entering the action but appears when the 3D Curve dialog box is open.

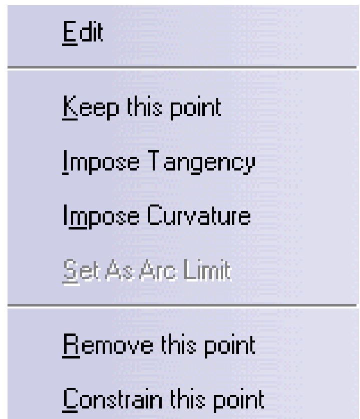
- In the specification tree:
  - select the geometrical set just by clicking it, or
  - right-click the geometrical set and choose Select all points in the geometrical set from the contextual menu, or

- select a point in the geometrical set, right-click it and choose Select all points in the geometrical set from the contextual menu.



- In the geometry: select a point, right-click it and choose Select all points in the geometrical set from the contextual menu.

Contextual Options



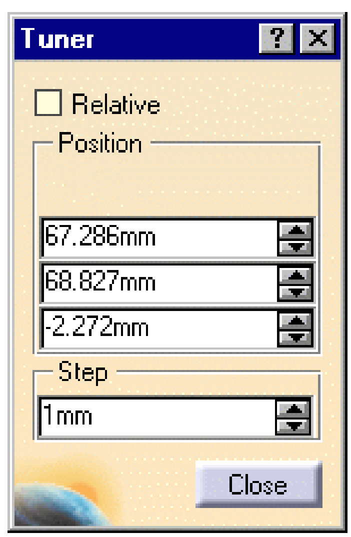
Double-click your curve, right-click on the manipulator to display the contextual menu.

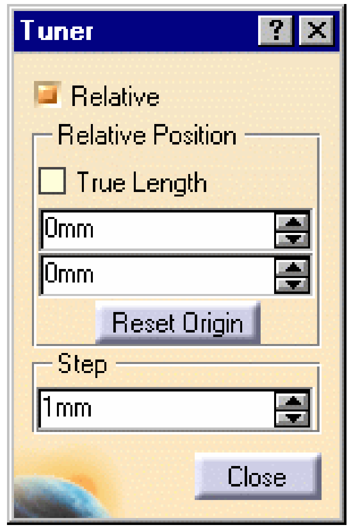
According to the creation type, the following options are available:

	Through Points	Control Points	Near points
Edit	X	X	X
Keep this point	X	X	X
Impose Tangency	X		X
Impose Curvature	X		X
Set as Arc Limit			X
Remove this point	X	X	X
Constrain this point	X	X	X

Editing

1. Right-click any of the manipulators, and choose the Edit contextual menu to display the Tuner dialog box. This option allows you to redefine the tangency position, and its vector's step.





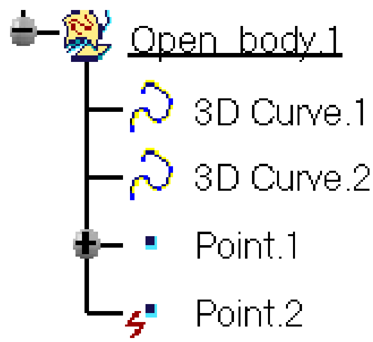
The **Relative** check box enables you to redefine the tangency relative position, and its vector's step.
The **Reset Origin** button allows you to reset the origin of the relative position.

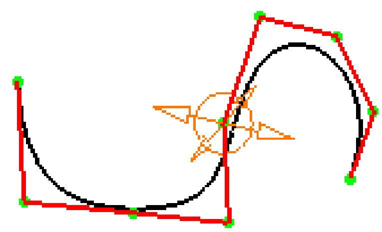
Keeping a point

1. Right-click an existing point and choose the Keep this point menu item to create a point at this location.

   A Point.xxx appears in the specification tree.

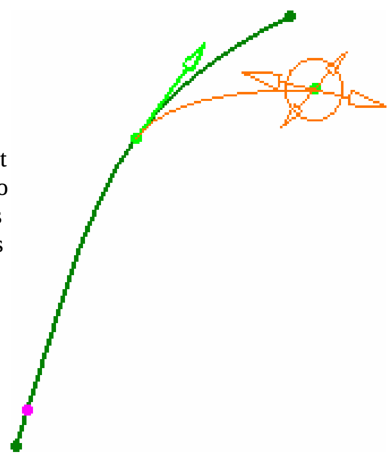
   You can create a Point.xxx either on each control point or on the selected control points.





Imposing a Tangency Constraint

Automatic Constraint



- When a curve is created in **Through points** or **Near points** mode, and its first point is constrained on any point of another curve, the new curve automatically is tangent to the curve on which its first point is constrained. As soon as the curve's second point is created, the imposed tangent is displayed on the new curve.
  To deactivate the default option, uncheck the **Impose Tangency** contextual menu on the tangent vector.

Tangency Constraint on Points

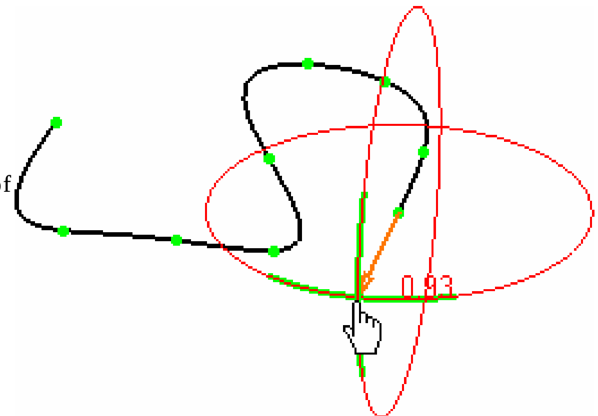
When creating a 3D curve, you may want to impose tangency constraints on specific points of the curve. Then if you move the point at which a tangency constraint has been set, the curve will be recomputed to retain this tangency constraint at the point's new location.
Depending on the creation mode, you can impose this constraints on a limited number of points:

- In **Through points** mode: tangency can be imposed on any point
- In **Near points** mode: tangency can be imposed independently on each end points only
- In **Control points** mode: no tangency constraint can be imposed (end points can be constrained on other elements as described in step 7 above. See also Constraining a Control Points Curve.

Here is how to do it:

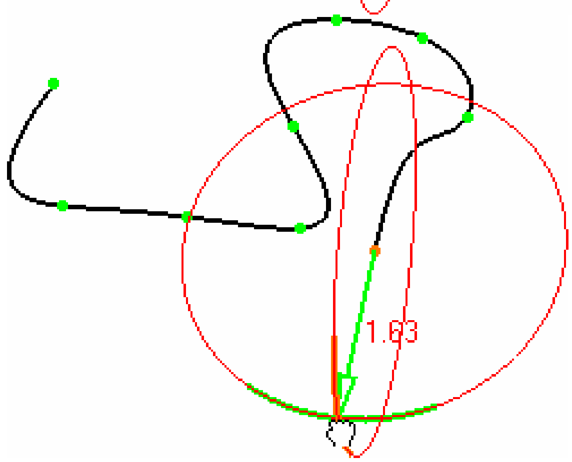
Open the 3DCurve1.CATPart document.

1. Move the pointer over an existing point, double-click it (the 3D curve dialog box appears), then right-click and choose the Impose Tangency menu item.



Two sets of manipulators are displayed:

- two arrows representing the normed directions (vectors) of the tangency
- circles representing manipulators for this vector



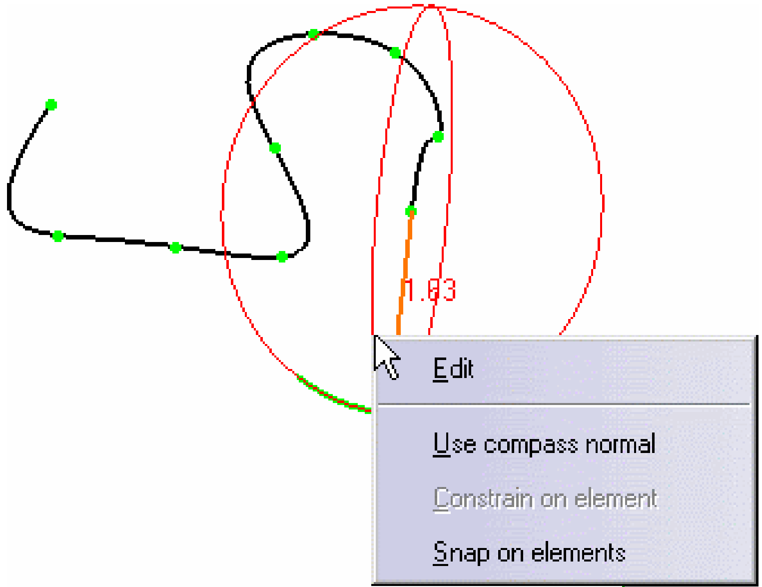
You can also modify the tangency constraint by:

- pulling the arrow
- gliding the circles
- double-clicking the arrow to invert the tangency direction

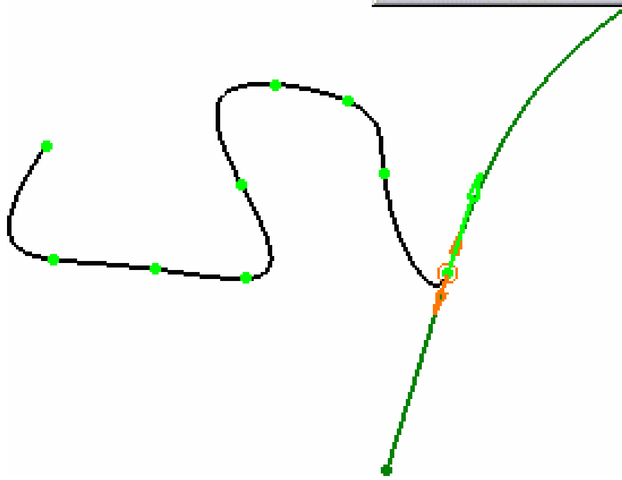
You can set the tangency length by clicking on the arrow then dragging the mouse.

Right-clicking on any of the manipulators, you can also choose to define the constraint according to an external element:

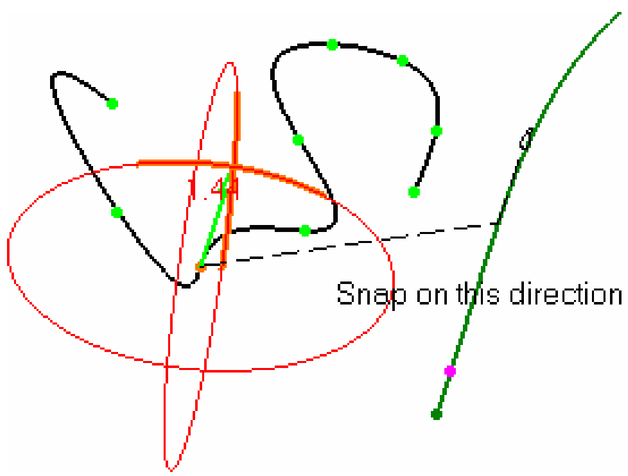
- **Use current plane orientation** (P1)/**Use compass normal** (P2): the tangency constraint is defined in relation to the normal to the current plane, possibly defined by the normal to the compass main plane
  When several points are constrained on the compass, all are modified if the compass settings are changed.
  When this option is checked, the direction cannot be modified directly using the vector manipulator, but only using the compass.



- **Constrain on element**: available only when a point is already constrained on a curve. The curve being created/modified becomes constraint in tangency or curvature to the constraining curve at this point.
  - Tangency constraint: in this case you can only modify the vector's norm using the **Edit** contextual menu, and no longer the vector's direction, the latter being defined by the constraining curve.
    Click the arrow to invert the tangency direction if needed.
  - Curvature constraint: in this case you neither modify the vector's norm using the **Edit** contextual menu, nor the vector's direction, the latter being defined by the constraining curve.

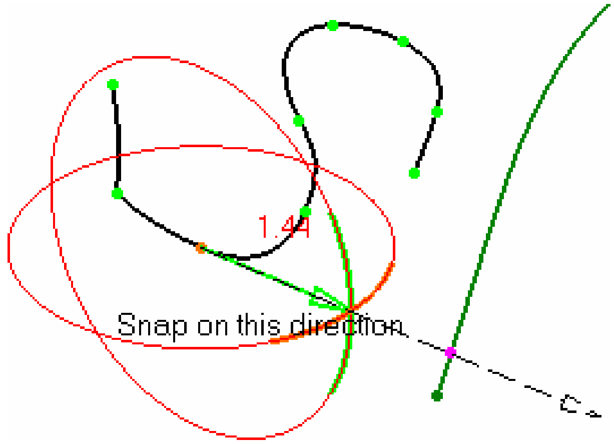


By default, when the tangency vector is constrained onto another curve, its initial direction is retained.



- **Snap on elements**: the vector's direction is defined by an external element. Grabbing a manipulators, you drag the pointer over a curve, and the curve becomes tangent to the curve detected by the pointer.

If the pointer is over a point the direction is computed as the line going from the constrained point and the detected point.
If the pointer is over a plane, the tangency is defined by the normal to this plane.



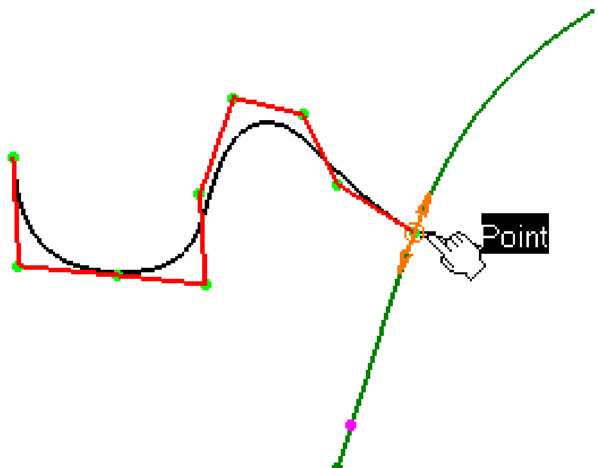
- When snapping on an element, use the Control (Ctrl) key to obtain an exact snap, taking into account both the detected element and the vector's norm.
- Use the Shift key as a shortcut to activate/de-activate the **Snap on elements option when** passing the pointer over geometric elements.

Once you are satisfied with the tangency constraint you imposed, simply release the manipulator and move the pointer around to recover the curve preview indicating that you are ready to create a new point.

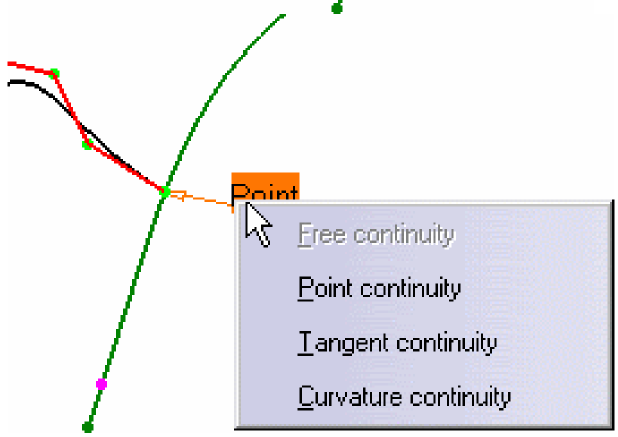
Control Points Curve Constraint

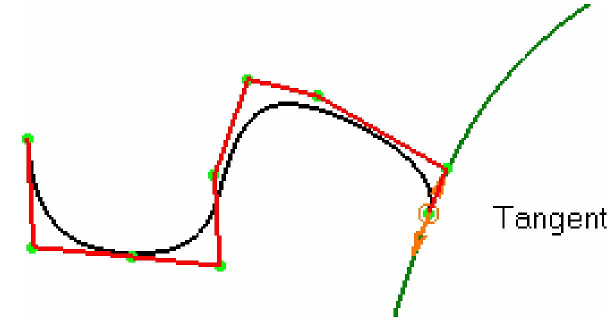
Even though you cannot impose a tangency constraint on a curve created in **Control points** mode, you can constrain its end points on another curve, as described in step 7 above.

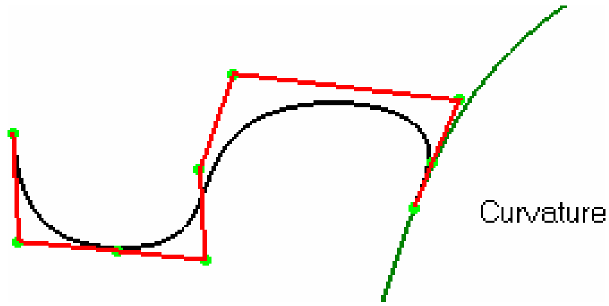
When setting a constraint on an end point, a text is displayed indicating the type of continuity between the two curves.



Right-click the text to display the contextual menu from which you can choose another continuity type: tangency, or curvature.







Note that:

- in **Point continuity**, only the selected point is constrained
- in **Tangent continuity**, the selected point and the next one are constrained
- in **Curvature continuity**, the selected point and the next two points are constrained

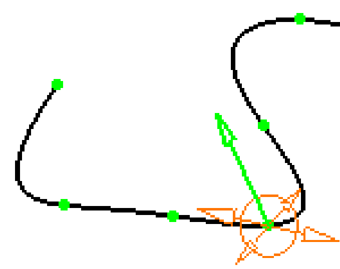
This means that these second and third points will be modified if you move the constrained point along the constraining element, using the manipulators. However, you cannot constrain these points, because they are considered as already constrained. If you try to do so, a warning message is displayed. Nevertheless, you can add/remove points directly after the constrained end point, and the system resets the points as second and third points to be affected by the constraint, where applicable.

A **Continuity** warning is displayed when trying to move the manipulators in a direction that is not compatible with the set constraint.

Imposing a Curvature Constraint

Right-click an existing point and choose the **Impose Curvature** menu item. An arrow representing the curvature direction (vector) is displayed. Modifying the vector direction modifies the curvature direction.

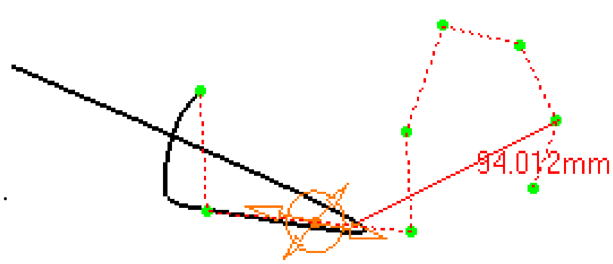
The direction of the curvature is constrained in the plane defined as normal to the tangent vector.



- To impose a curvature continuity, you must ensure that a tangency continuity already exists.
- This option is only available for the **Through points** and **Near Points** creation type.

Setting as Arc Limit

Right-click an existing point and choose the **Set As Arc Limit** menu item to start/stop an arc limit on this point. The curve will pass through this point.



This option is only available for the **Near points** creation type.

You can use standard shortcuts (Ctrl and Shift keys) to select, multi-select, and de-select any combination of control points on these curves.

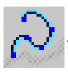
Available capabilities from the Dashboard, and/or specified through the FreeStyle Settings, are: datum creation, temporary analysis, auto detection (except for Snap on Control Point option), attenuation, and furtive display.

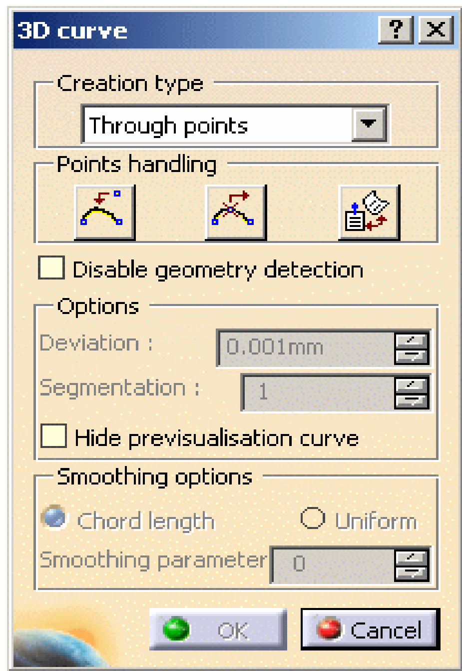
Creating Associative 3D Curves on a Scan of Cloud

This task explains how to create a 3D curve on a scan of cloud:

- either before entering the 3D curve action. In that case, you can select only one scan,
- or after entering the 3D curve action. In that case, you can select one or several scans.
- either graphically. In that case, use the contextual menu Select all points to create a 3D curve on all the points of the scan,
- or from the specification tree. In that case all points of the scan are taken into account even if you do not activate the Select all points menu.

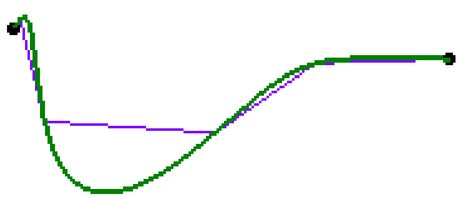
Open the ScanOnCloud1.CATPart document.

1. Click the **3D Curve** icon .
   The 3D curve dialog box is displayed.
2. Choose the curve creation type.

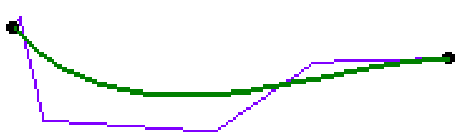


Whatever the chosen type, the curve is previewed in dotted line as you move the pointer.

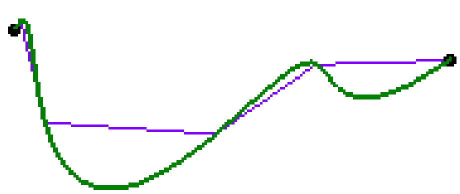
- **Through points**: the resulting curve is a multi-arc curve passing through each selected point.



- **Control points**: the points you click are the control points of the resulting curve



- **Near points**: the resulting curve is a single-arc, with a set degree and smoothed through the selected points.

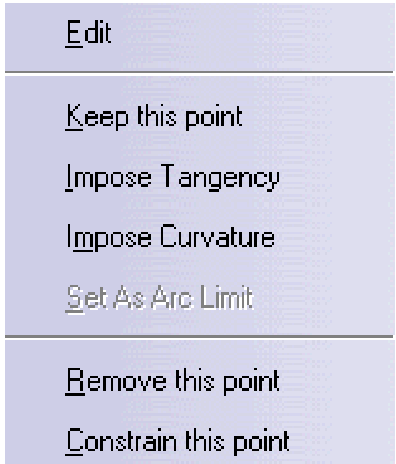


3. Click OK to create the curve.

A 3DCurve.xxx appears in the specification tree.

4. **Double-click your curve, right-click on the manipulator to display the contextual menu.**

Please refer to the Creating Associative 3D Curves to get the corresponding information.



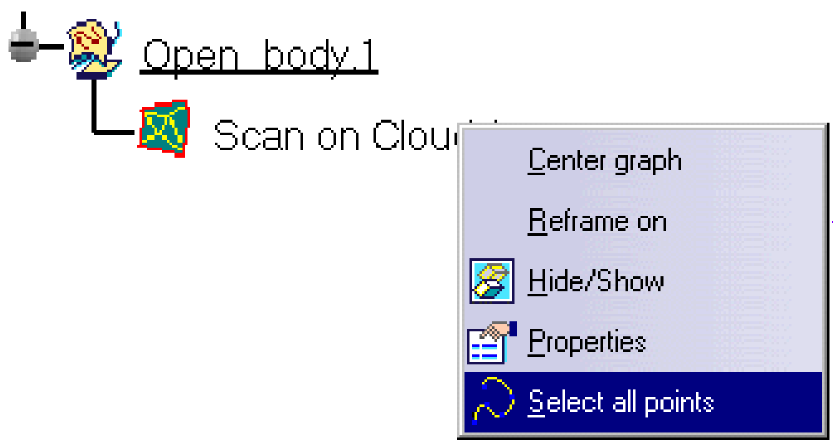
It is now possible to select a scan on cloud **before** entering the 3D curve creation dialog box. To do so, click the Scan on Cloud in the specification tree and click the 3D Curve icon.

Selecting a scan of cloud

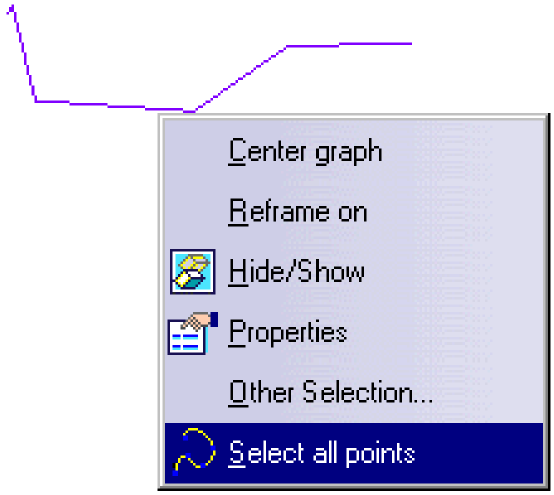
It is possible to select a scan of cloud either in the specification tree or directly in the cloud from the contextual menu:

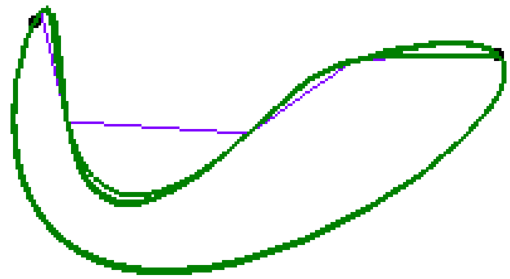
The Select all points contextual menu is available within the 3D curve action only, i.e. it is not available when you select the scan before entering the action but appears when the 3D Curve dialog box is open.

- Right-click the Scan on Cloud.xxx in the specification tree and select **Select all points**.



- Right-click the cloud and select **Select all points**.





Here is an example with Through Points

Only scans of the type "scan on cloud" can be selected since other types of scans might contain too many points.

- Use the F5 key to move the manipulators into a different plane of the compass. See Managing the Compass.
- The creation plane for each free point is defined according to the current plane/compass orientation on the previous point. Therefore you can change creation planes within the same curve, by setting a new current plane/compass orientation on several points.
- Check the **Disable geometry detection** button, when you need to create a point close to a geometric element yet without constraining it onto the existing geometry.
- Check the **Hide previsualisation curve** to hide the previsualisation curve you are creating.

For further information on the proposed Options, please refer to Creating Associative 3D Curves.

You can use standard shortcuts (Ctrl and Shift keys) to select, multi-select, and de-select any combination of control points on these curves.

Available capabilities from the Dashboard, and/or specified through the FreeStyle Settings, are: datum creation, temporary analysis, auto detection (except for Snap on Control Point option), attenuation, and furtive display.

Curves from Scans

This task shows how to create curves from a scan or a set of scans.

The **Curve from Scans** action tries to create curves

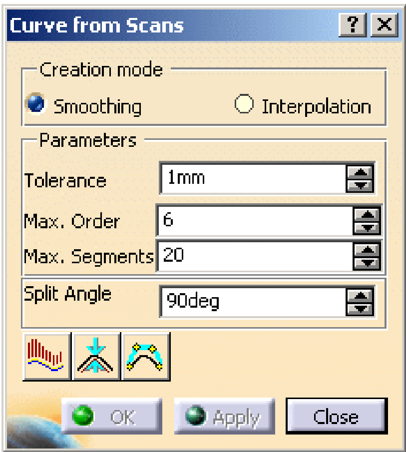
- with the defined tolerance,
- with the least possible number of segments of the least possible order.

The **Curve from Scans** action proposes a dynamic definition of split points.

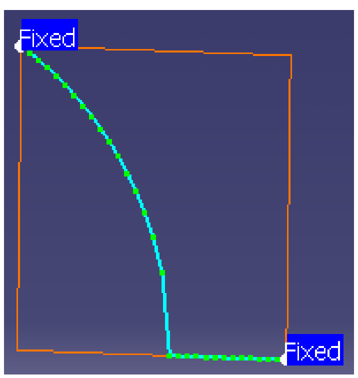
Open the CurveFromScan1.CATPart model from the samples directory.

For a better understanding, some images below show only one scan.

1. Select the Curve from scans icon and a set of scans. The **Curve from Scans** dialog box is displayed.



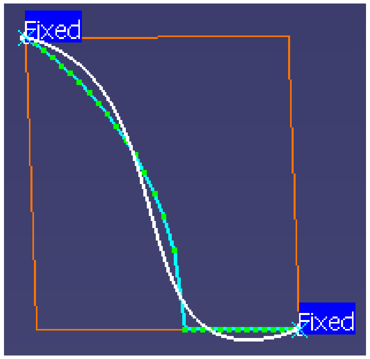
The scan is displayed in the "Polyline+Point" mode with the current graphic symbol. By default, the end points are fixed.



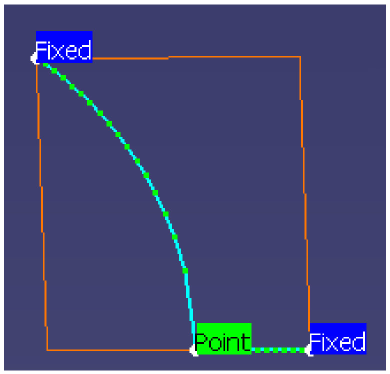
- Scan modifications are not dynamic. You have to click on **Apply** to take new parameter values into account.
- The scans can be selected in the specification tree.

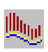
2. Make sure the **Smoothing Creation mode** is checked.

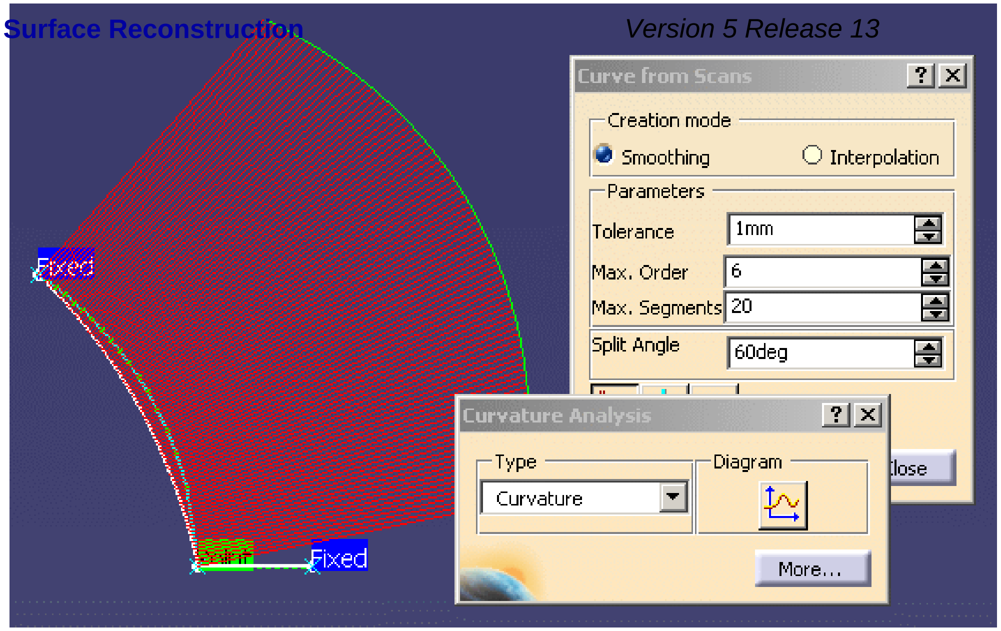
3. Click Apply. A temporary curve is displayed in white, indicating the tolerance is met.



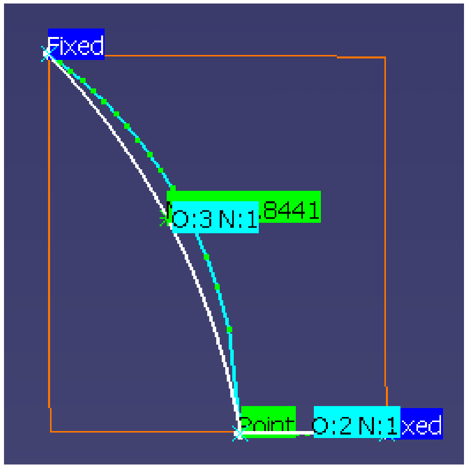
4. Change the **Split Angle** value to 60. A Split point is automatically inserted at the angle.



5. Push the  icon to display the curvature analysis:



6. Release the icon and now push the icon to display the maximum deviation and the icon to display the order and the number of segments:



7. Click **OK** to exit the action and create the curve(s). Curve.x elements are created in the specification tree. The segmentation display is erased.

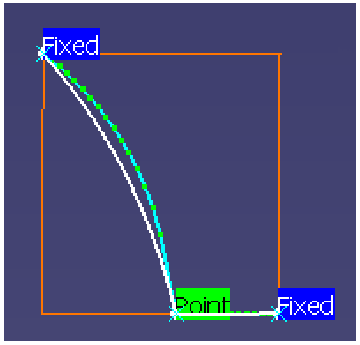
Parameters

Creation mode

Two creation modes are proposed:

- **Smoothing** or
- **Interpolation**.

In the **Smoothing** creation mode, the curve is created by smoothing all the points between two split points.

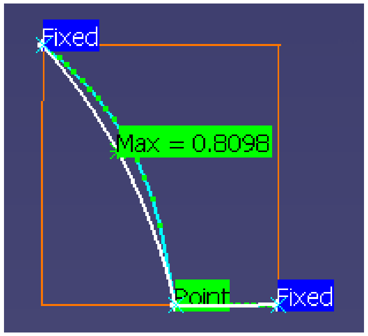


Three parameters are available in this mode:

- Tolerance,
- Max. Order,
- Max. Segments

Tolerance is the maximum distance between the curve and the points.

- Decrease the value of **Tolerance** to reduce that distance.
- You can check the distance between the curve and the points with the  icon.
- If the tolerance is met, the computed curve is displayed in white. If the maximum deviation display is activated, the maximum error is displayed in green.



- Otherwise, the computed curve is displayed in red. If the maximum deviation display is activated, the maximum error is displayed in red.

Max. Order is the maximum order of the curves created, i.e. the number of control points of those curves.

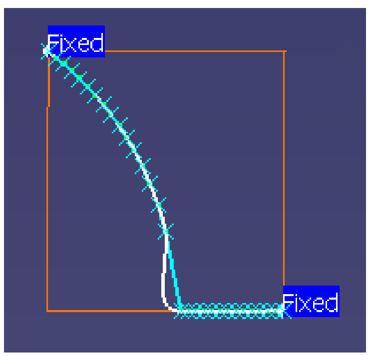
Max. Segments which is the maximum number of spans between two cutting points.

The **Curve from Scans** action tries to create curves with the defined tolerance, with the least possible number of segments of the least possible order. I.e.:

- the action tries to create a curve with one segment of order 2.
- If the tolerance is not reached, the order of the segment is increased up the the **Max. Order** value.
- If the tolerance is not yet reached, the number of segments is increased, the segments having the least possible order. This order is increased to meet the tolerance, then the number of segments, and so on until both the maximum order and the maximum number of segments are reached.

You can check the segmentation and the order of the curves with the  icon.

In the **Interpolation** mode, the curve is created by interpolating the points of its support scan.



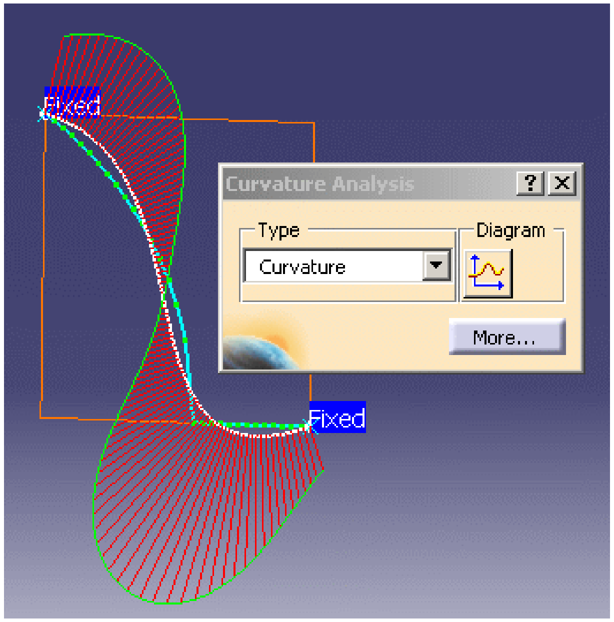
No parameters are proposed for this mode

Visualization and Analysis icons

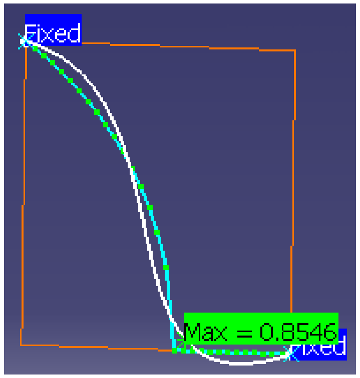
Three visualization icons are available: 

 analyzes the curvature of the resulting curves:

- This is a temporary analysis, no analysis element is created in the specification tree.
- Click on **More** to display more analysis options.
- Click on **Less** to display the quick analysis options
- More information is available in the Curvature Analysis section.



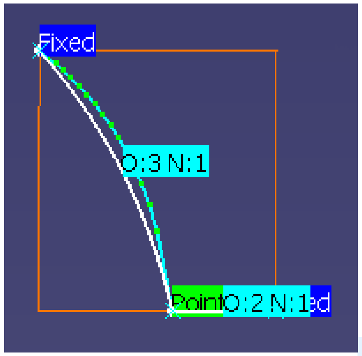
 displays the maximum deviation (not available for the Interpolation mode):



The label is green if the tolerance is met, otherwise it is red.

 displays the order and the number of segments (not available for the Interpolation mode):

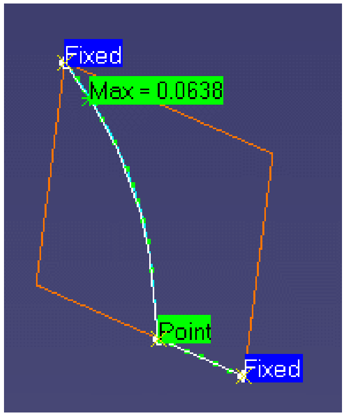
When the curve computed is segmented, the segmentation is displayed with blue x symbols. This color and symbol are not editable.



Push the icons to activate the display, release the icon to erase the display. Those three displays can be combined together.

Split angle

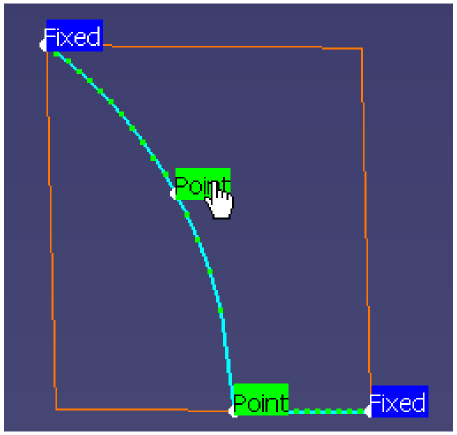
A split angle is proposed by default at 90 degrees. This value is editable. Whenever the computed curve forms an angle greater than this value, it is split automatically into two curves. The split spot is displayed and two curves are created.



If you modify the split angle value after having computed curves and before having validated them, the display of the computed curves is updated.

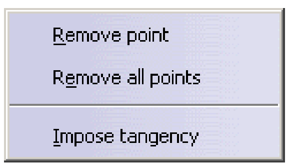
You can also:

- add split points by picking points of the scan.
- or move a split point to another point of the scan:
  - pick the split point you want to move: press the control key and the left-click on the label of the split point,
  - with the control key and the left button of the mouse still pressed, drag the split point to the required scan point and drop it there.



The default constraint on a split point is "Point", i.e. passage. Click on the green label to change it to "Tangent". A second click will return it to "Point".

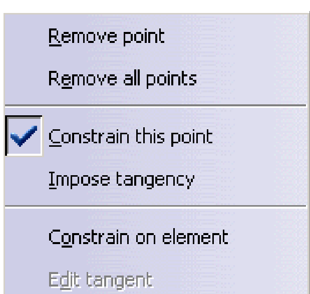
You can also use the Impose tangency of the contextual menu of the constraint.



This contextual menu can also be used to remove one split point or all split points.

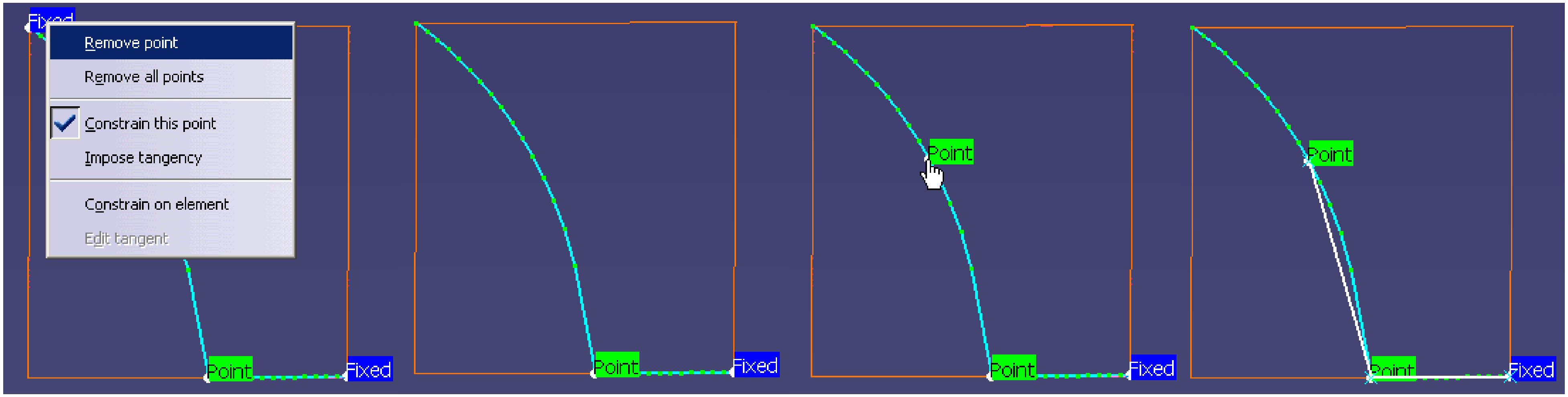
End points

End points propose more items in their contextual menu.

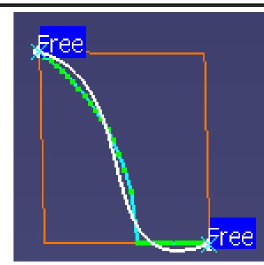
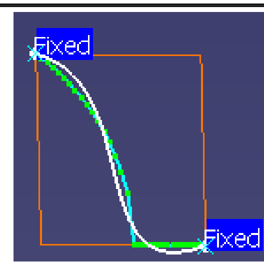


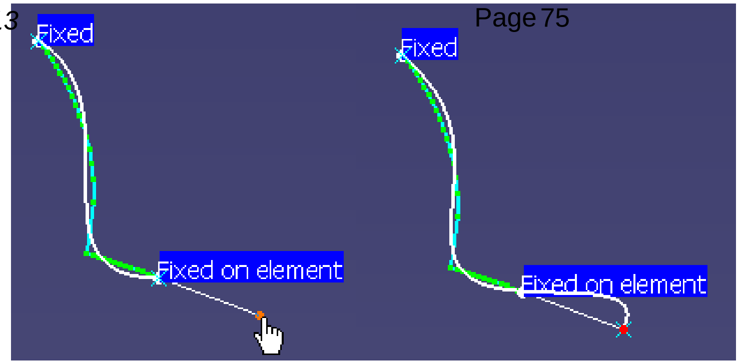
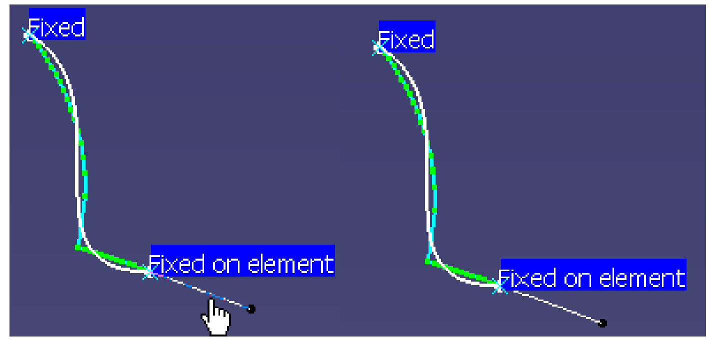
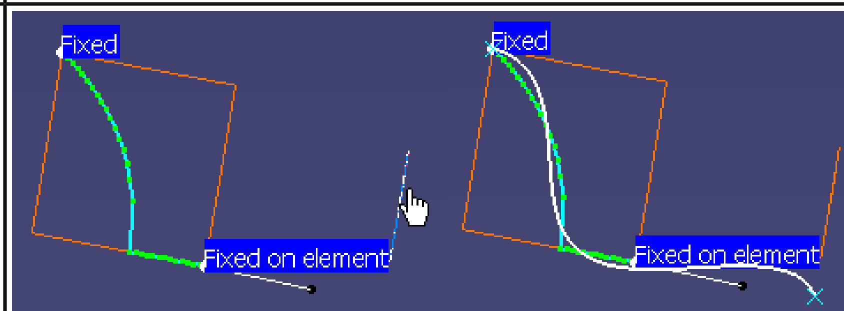
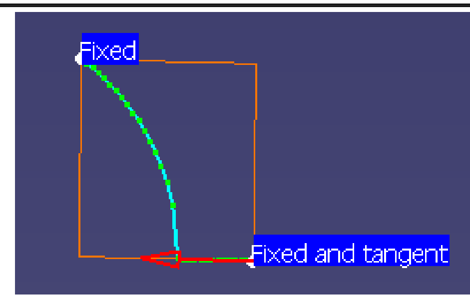
You can remove the extremity point of a computed curve, and replace it with a new or existing split point.

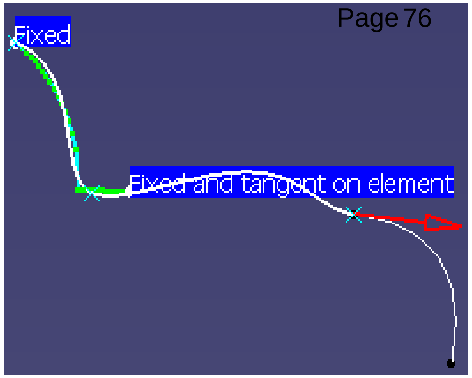
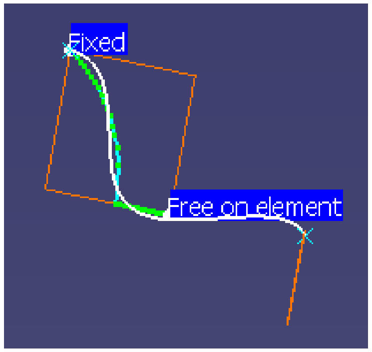
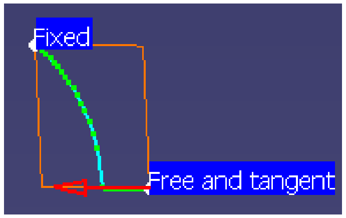
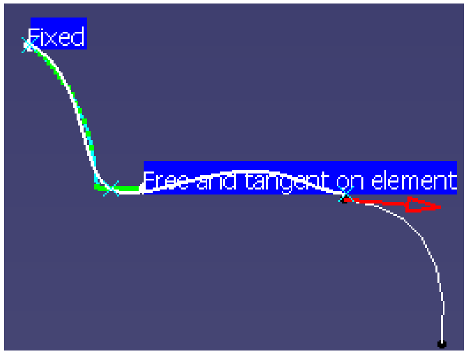
Remove the end point: The end point is removed: Pick a point on the scan: Click **Apply** to visualize the new curve:



You can free or constrain the end points by checking the appropriate option in the contextual menu.

Constrain this point	Impose tangency	Constrain on element	Edit tangent	Label	Meaning	
O	O	O	Not available	Free	The extremity is free	
X	O	O	Not available	Fixed	The extremity is fixed, it is the scan extremity (this is the option by default)	

X	O	X	Not available	Fixed on element	If the element is a point, the extremity of the curve is this point, not the scan extremity See constrain on element	
					If the element is a curve, the curve extremity is the nearest extremity of the constraining curve See constrain on element	
					If the element is a plane, the curve extremity is the point of the plane nearest to the scan extremity See constrain on element	
X	X	O	Available	Fixed and tangent	The tangency direction is given by a vector going through the first (or the last) two points of the scan.	

X	X	X	Available	Fixed and tangent on element	The extremity is fixed and tangent to the constraining element. See constrain on element	
O	O	Y	Not available	Free on element	The extremity is free and depends on the element. See constrain on element	
O	X	O	Available	Free and tangent	The extremity is free. The tangency direction is given by a vector going through the first (or the last) two points of the scan.	
O	X	X	Available	Free and tangent on element	The extremity is free and depends on the element. The tangency direction is given by the element (plane or curve). See constrain on element	

Constrain on element:

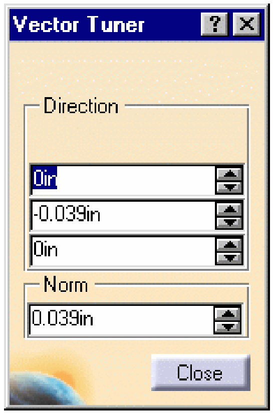
The extremity of the curve computed is the scan extremity, unless the **Constrain on element** option is checked.

- If the constraining element is a point, the curve extremity is that point,
- if the constraining element is a curve, the computed curve extremity is the nearest extremity of the constraining curve,
- if the constraining element is a plane, the curve extremity is the orthogonal projection of the scan extremity on to the plane.

If an extremity is constrained on an element, and if a tangency is imposed, the tangency direction is given by:

- the tangent vector of the curve on the constraining curve extremity,
- or the normal to the constraining plane.

Whenever the **Impose tangency** is checked, you can modify the tangent vector with the following box (Edit tangent in the contextual menu):



- It is useful to impose a tangency constraint at the extremities if you intend to reconstruct a part in two steps:
  - reconstruction of the first half of the part,
  - recovery of the whole part by performing a symmetry.

The tangency constraint will ensure that the two halves fit perfectly.

- Only the tangency direction is taken into account, the norm is not.

This option is available in the Curve from Scans action only.

Creating Intersections

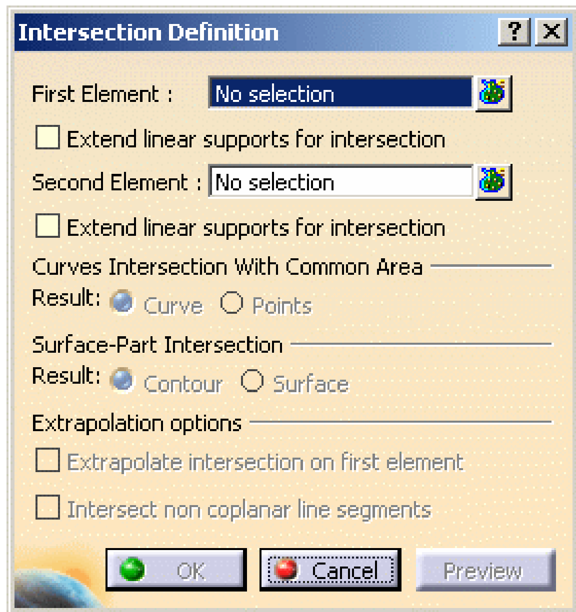
This task shows you how to create wireframe geometry by intersecting elements.

You can intersect:

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

Open the Intersection1.CATPart document.

1. Click the **Intersection** icon .



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

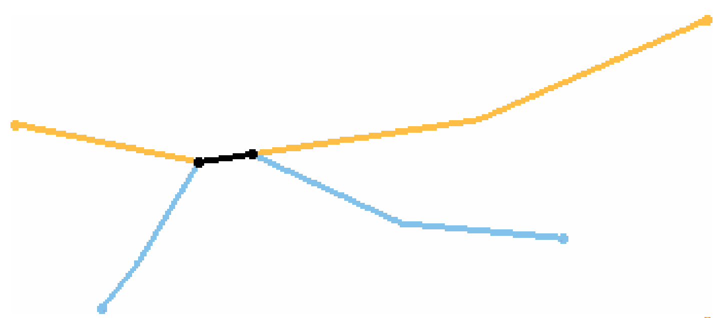
2. Select the two elements to be intersected.

   The intersection is displayed.

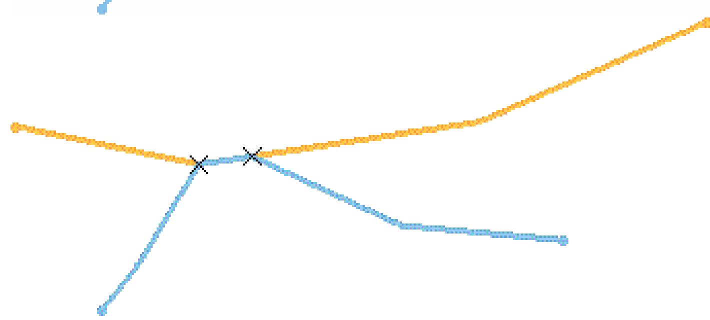
Multi-selection is available on the first selection, meaning you can select several elements to be intersected, but only one intersecting element.

3. Choose the type of intersection to be displayed:

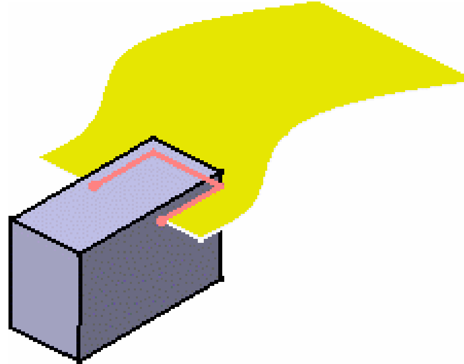
- A **Curve**: when intersecting a curve with another one



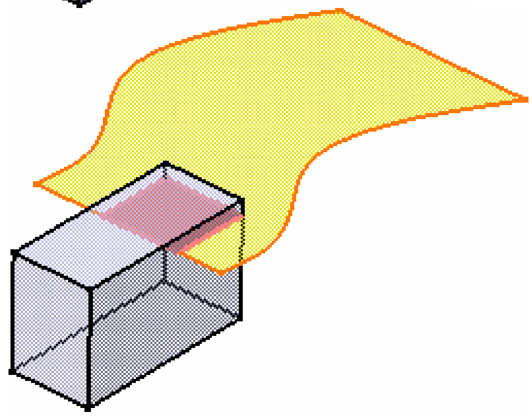
- **Points**: when intersecting a curve with another one



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



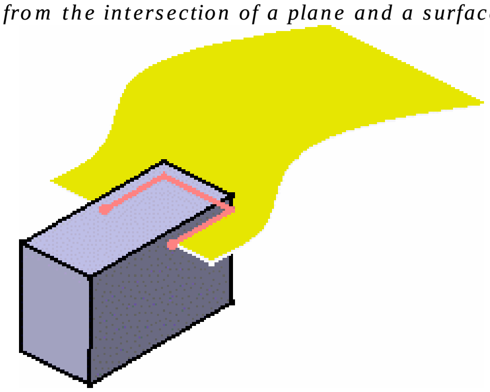
- A **Face**: when intersecting a solid element with a surface (we increased the transparency degree on the pad and surface)



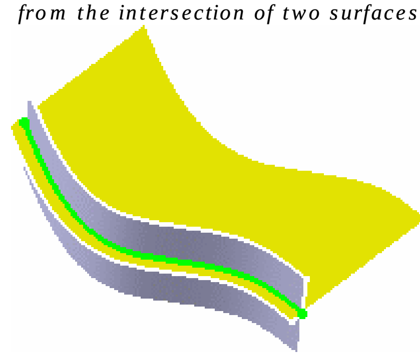
4. Click OK to create the intersection element.

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

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



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



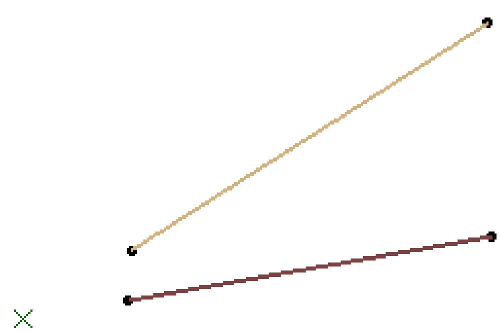
Several options can be defined to improve the preciseness of the intersection.

Open the Intersection2.CATPart document.

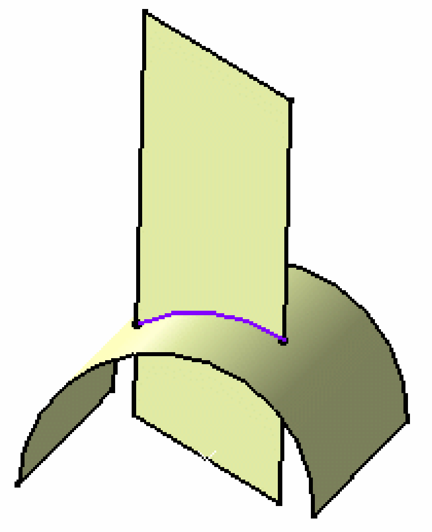
- The **Extend linear supports for intersection** option enables you to extend the first, second or both elements.

Both options are unchecked by default.

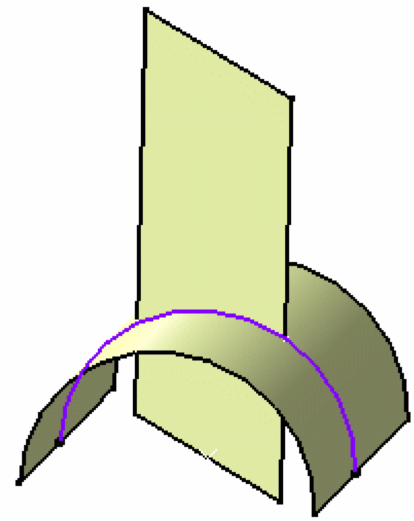
Here is an example with the option checked for both elements.



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



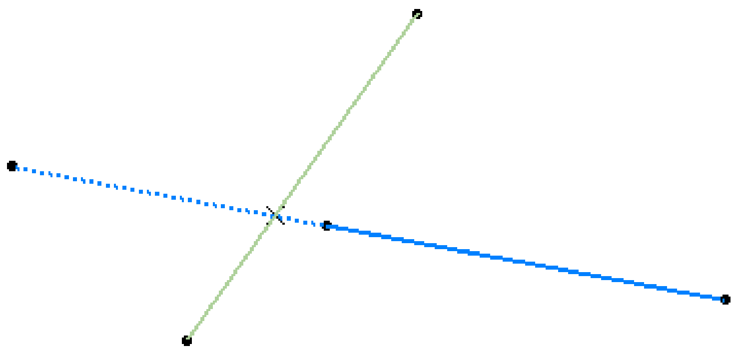
Intersection without the Extrapolation option checked



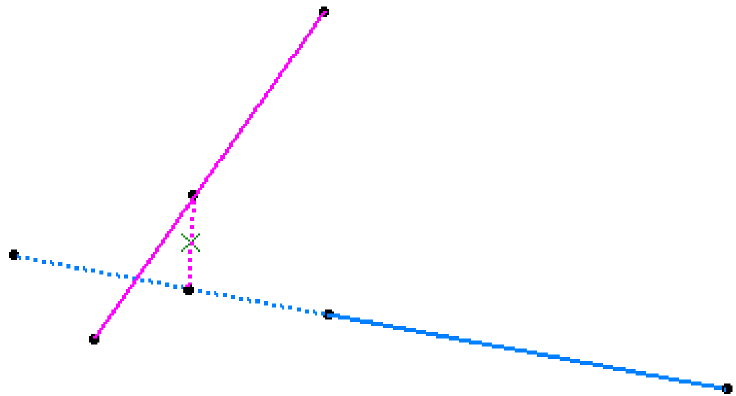
Intersection with the Extrapolation option checked

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

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



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



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

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

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

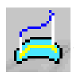
Creating Projections

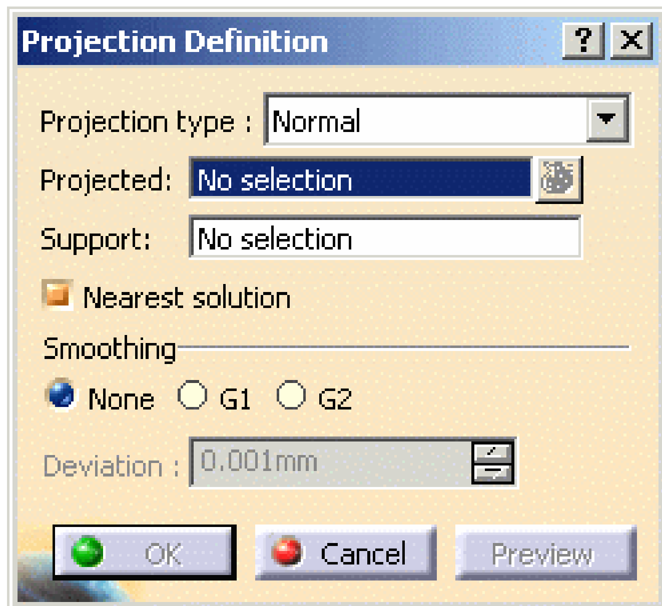
This task shows you how to create geometry by projecting one or more elements onto a support. The projection may be normal or along a direction.
You can project:

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

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

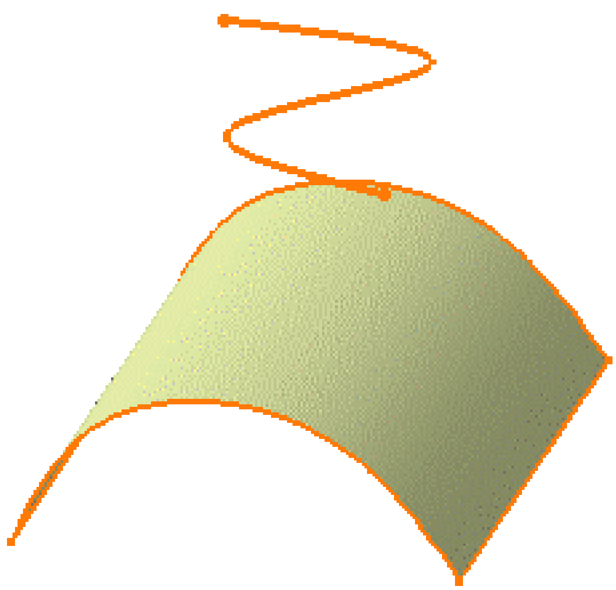
Open the Projection1.CATPart document.

1. Click the **Projection** icon .



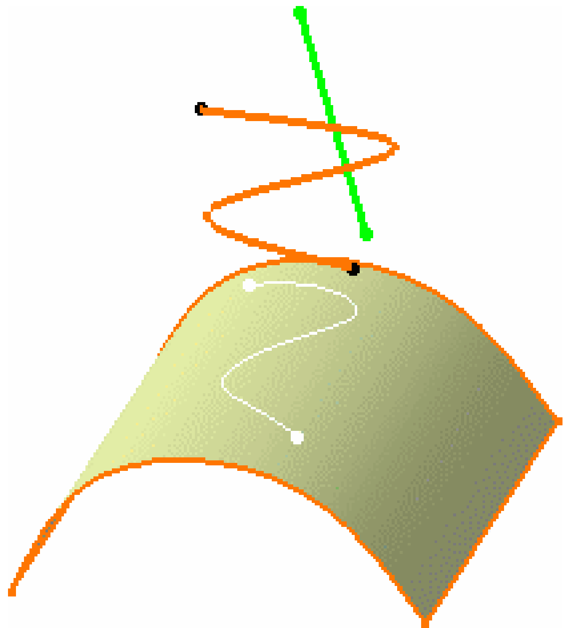
The Projection Definition dialog box appears as well as the Multi-Selection dialog box allowing to perform multi-selection.

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



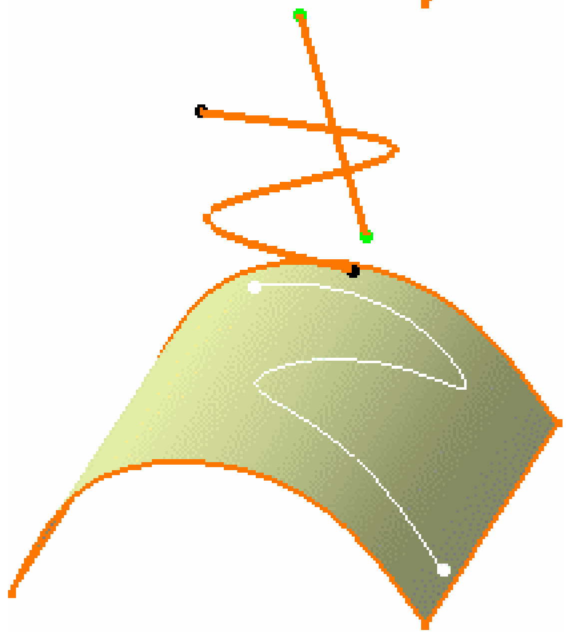
You can select several elements to be projected. In this case, the **Projected** field indicates: **x elements**

3. Select the **Support** element.



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

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

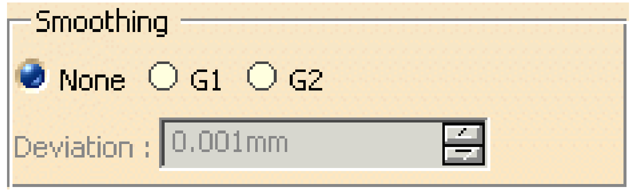


- **Along a direction**: you need to select a line to take its orientation as the translation direction or a plane to take its normal as the translation direction.

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

- Whenever several projections are possible, you can select the **Nearest Solution** check box to keep the nearest projection.
The nearest solutions are sorted once the computation of all the possible solutions is performed.

- You can smooth the element to be projected by checking either:
  - **None**: deactivates the smoothing result
  - **G1** : enhances the current continuity to tangent continuity
  - **G2** : enhances the current continuity to curvature continuity



- You can specify the maximum **deviation** for G1 or G2 smoothing by entering a value or using the spinners.

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

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

5. Click OK to create the projection element.

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

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

Selecting Using Multi-Output

This capability enables to keep the specification of a multi-selection input in a single operation. It is available with the following functionalities:

- Intersections
- Projections
- All transformations: translation, rotation, symmetry, scaling, affinity and axis to axis
- Developed wires (Generative Shape Optimizer)

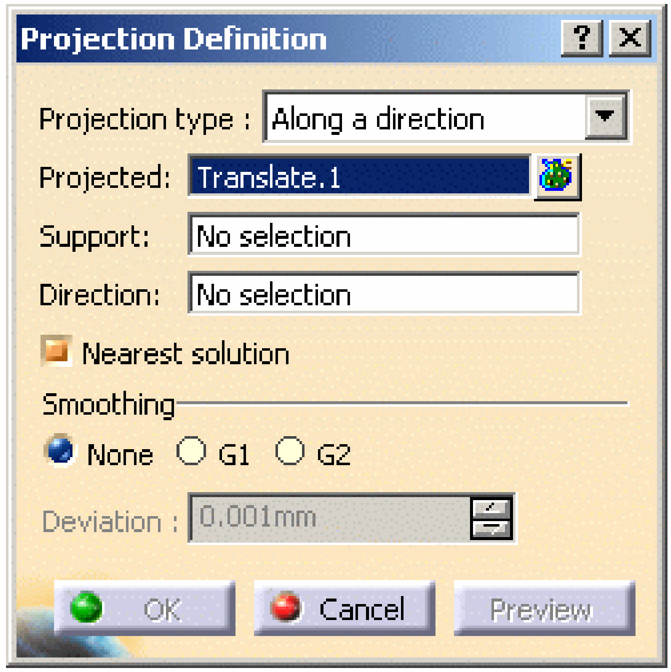
Let's take an example using the Projection and Translation functionalities.

Open the Multi-Output1.CATPart document.

1. Click the **Projection** icon.

The Projection Definition dialog box appears, as long as the Tools Palette toolbar.

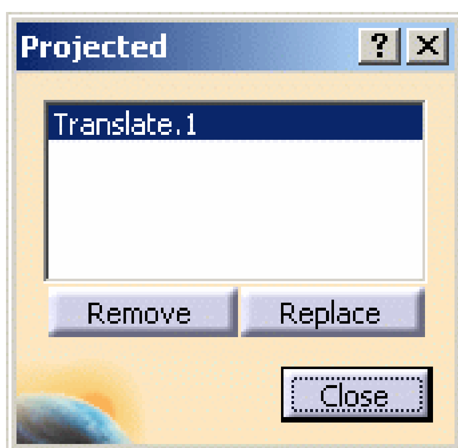
2. Select Translate.1 as first element to be **Projected**.



3. Click the bag icon to display the elements list.

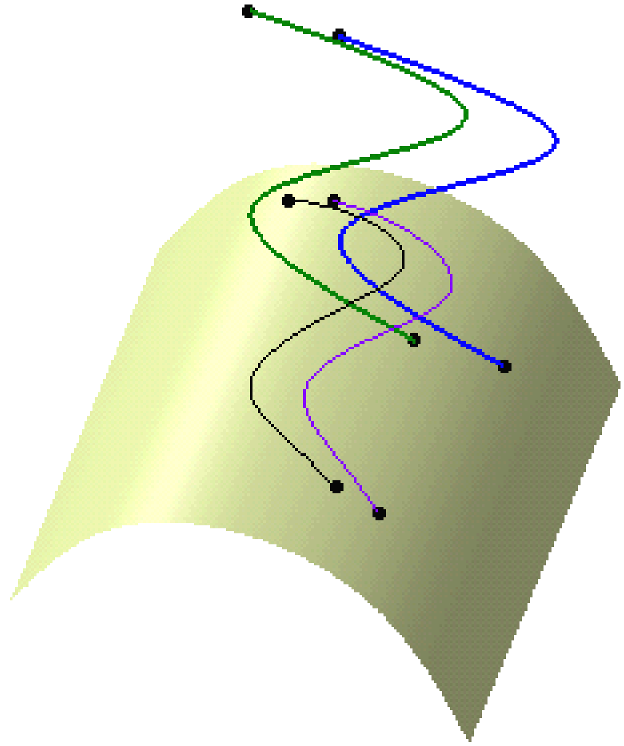
   The Projected dialog box opens.

4. Select as many elements as you need for your projection.
5. Click **Close** to return to the Projection Definition dialog box.



The number of selected elements is displayed **Projected** field.

Use the **Remove** and **Replace** buttons to modify the elements list.

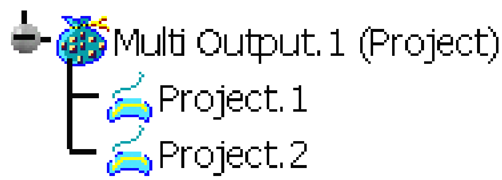


6. Select Extrude.1 as the **Support** element.

7. Select **Normal** as Projection type.

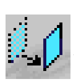
8. Click **OK** to create the projection elements.

The projection is identified as Multi Output.1 (Project) in the specification tree.

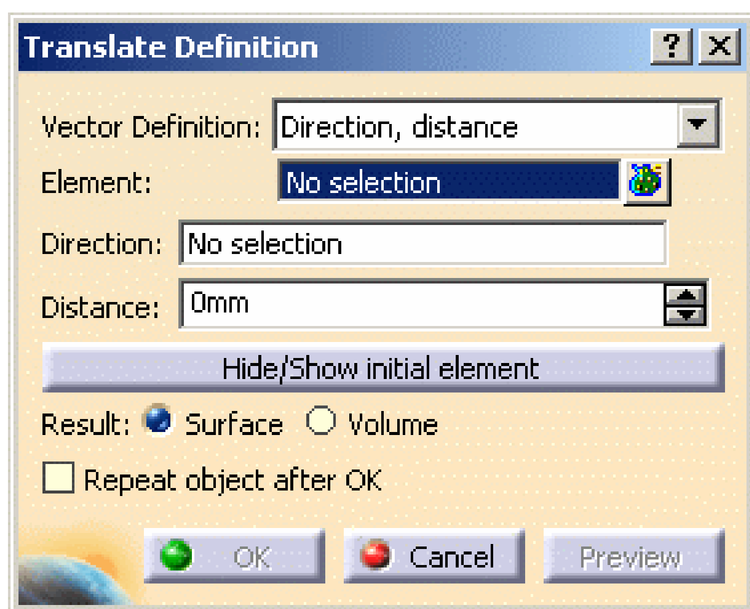


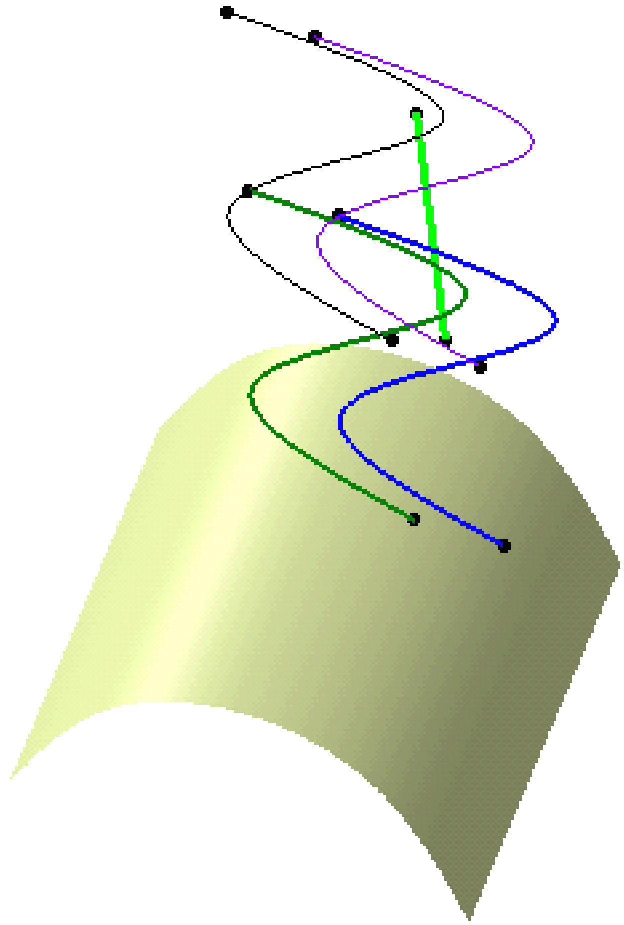
The created elements are aggregated under Multi Output.1.

You can create several multi-outputs in the specification tree, each one grouping one type of elements.

9. Click the **Translate** icon .

The Translate Definition dialog box appears.





10. Select Translate.1 and Translate.2 as the **Elements** to be translated.

11. Select Direction, distance as the **Vector Definition**.

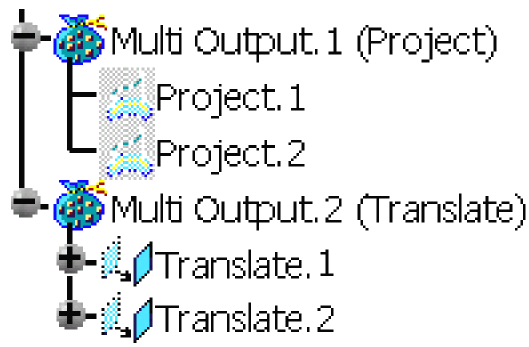
12. Select Extract.2 as the **Direction**.

13. Select -50mm as the **Distance**.

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

The translation is identified as Multi Output.2 (Translate) in the specification tree and appears below Multi Output.1.

The created elements are aggregated under Multi Output.2.



When one or several features are in error under a multi-output (during creation or edition), an error message is issued after clicking **Preview** or **OK** and displays all features in error.

You are able to manually delete or deactivate the feature(s) in error. When editing the multi-output, deactivated features are not displayed.

You can now deactivate the all the elements of a multi-output. As a consequence, the multi-output disappears from the 3D geometry and no more features in error can be generated.
Similarly, you can activate all the elements of a deactivated multi-output.
To have further information on deactivation, please refer to the Deactivating Features chapter.

- Multi-selection is available when editing a single feature: double-click it in the specification tree and click the bag icon to replace it or add new elements.

- Multi-outputs and elements aggregated under a multi-output can be edited separately, simply by double-clicking it in the specification tree. Elements can be modified (added, replaced, or removed): the corresponding multi-output automatically updates.

Unshared features are aggregated under the parent command that created them and put in no show in the specification tree.
Shared features are not aggregated under the parent command.

- The Datum capability is available. If an element is in error, it cannot be created as a datum element; only elements that could be generated from the multi selection are created.

Domain Creation

You can use the following tools to create domains:

Clean Contour
Adjust nodes

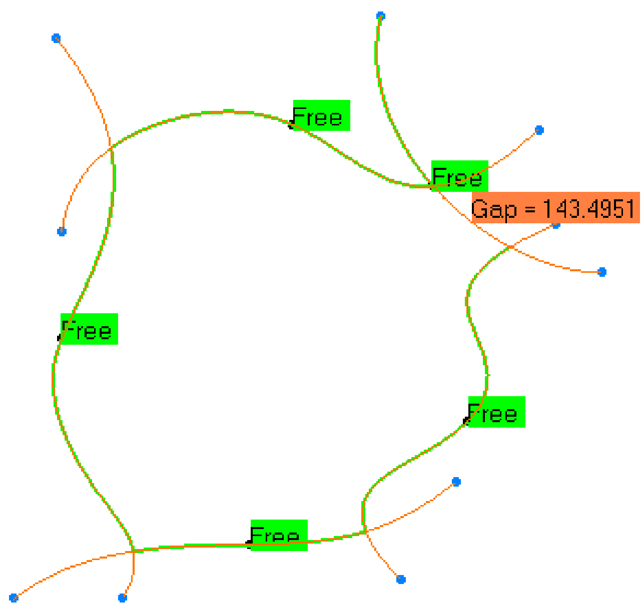
CleanContour Creation

This task shows how to create a CleanContour.

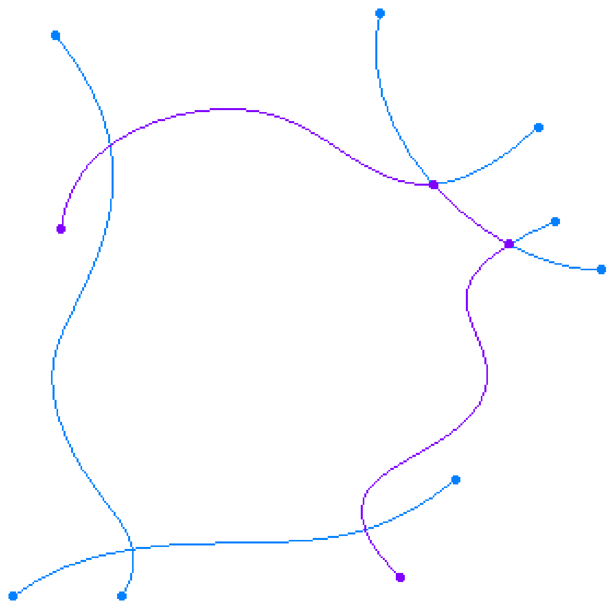
A CleanContour is created from selected edges or curves, ordered or not, that are chained into an open or closed contour:

- Each input element is cleaned (burred or trimmed),
- Continuity or fixity constraints are applied to those elements and taken into account during the cleaning,
- "Free" curves are sent to the NoShow and a new curve is created in the CleanContour.
  "Fixed" curves (by default face edges or curves belonging to another CleanContour or a join) are not replaced by new ones and are integrated directly in the CleanContour.
- These curves are then chained to produce the CleanContour.

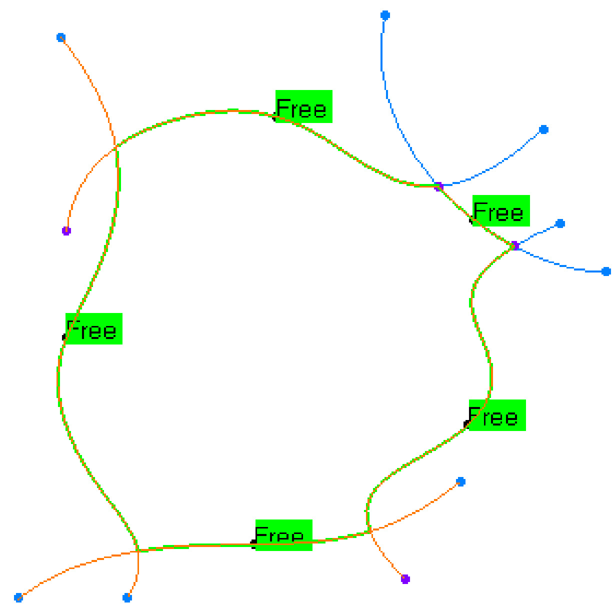
The CleanContour action sets the chaining order of the curves to create a contour. In some cases (especially with long curves) the chaining may lead to an unexpected result. You may need to slice curves or edges in order to solve this chaining incompatibility. For further information, see "Curves Slice ".



original curves: CleanContour impossible



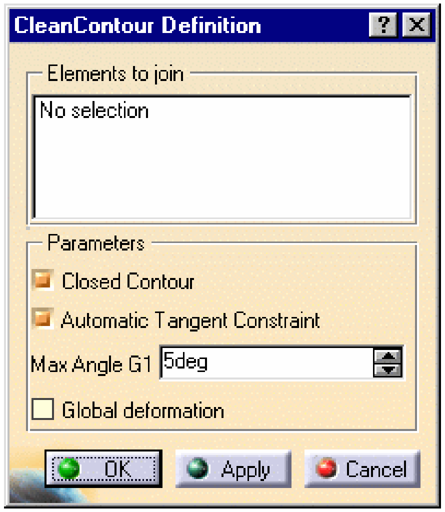
sliced curves



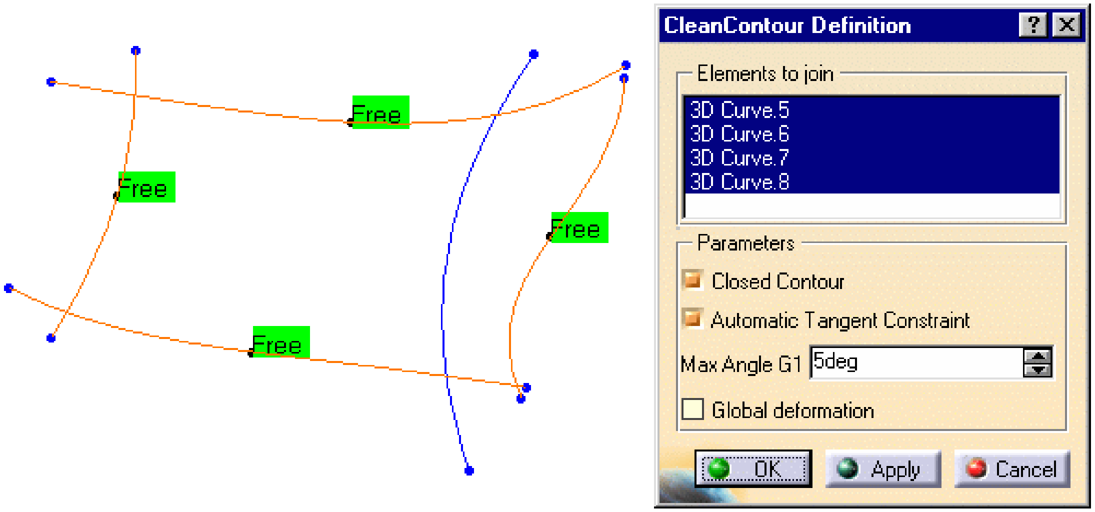
CleanContour

Open the CleanContour1.CATPart from the samples directory

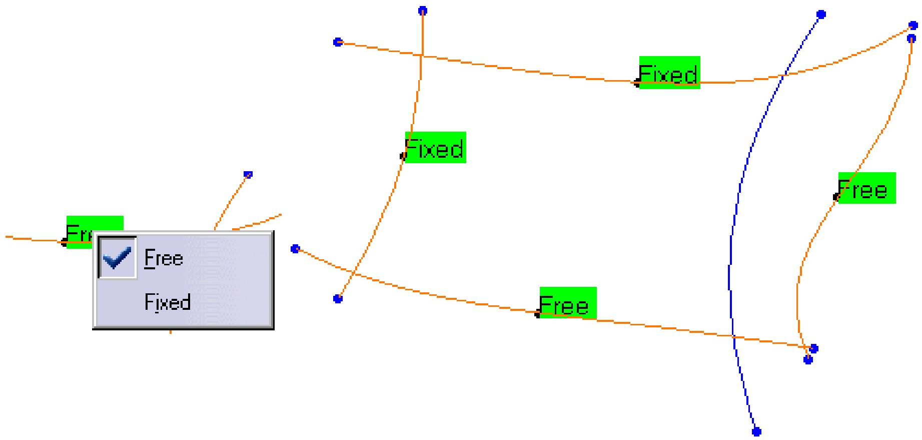
1. Click the **CleanContour** icon . The **CleanContour** dialog box is displayed.



2. Select the curves. The default constraints are displayed on each curve. The curves are listed in the dialog box :



You can change them by simply clicking on the text or using the contextual menu.



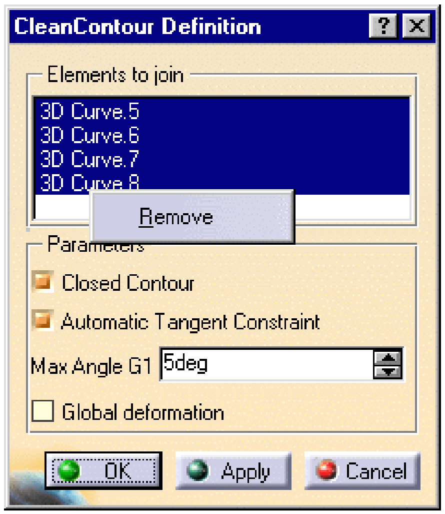
- If you select a sliced surface edge, the constraint existing on the edge is not taken automatically into account. You have to restore it with the contextual menu.
- You may want to select a join. If this join contains a sliced surface edge, or a split CleanContour that contains a sliced surface edge, with a tangency constraint that you want to keep, pick the curves one by one, graphically, i.e. do not select a join by picking one vertex, nor select the elements in the specification tree.

If you want to replace one curve by another, pick the curve to replace and then the new curve.

There are several ways to modify the list of the elements selected:

a. Pick the name of the curve you want to delete in the list of the elements selected in the dialog box.

b. Call the contextual menu and click **Remove**. The curve is removed from the list, and is no longer highlighted in the graphic zone. This is useful when the curve to remove is too short to be selected graphically.



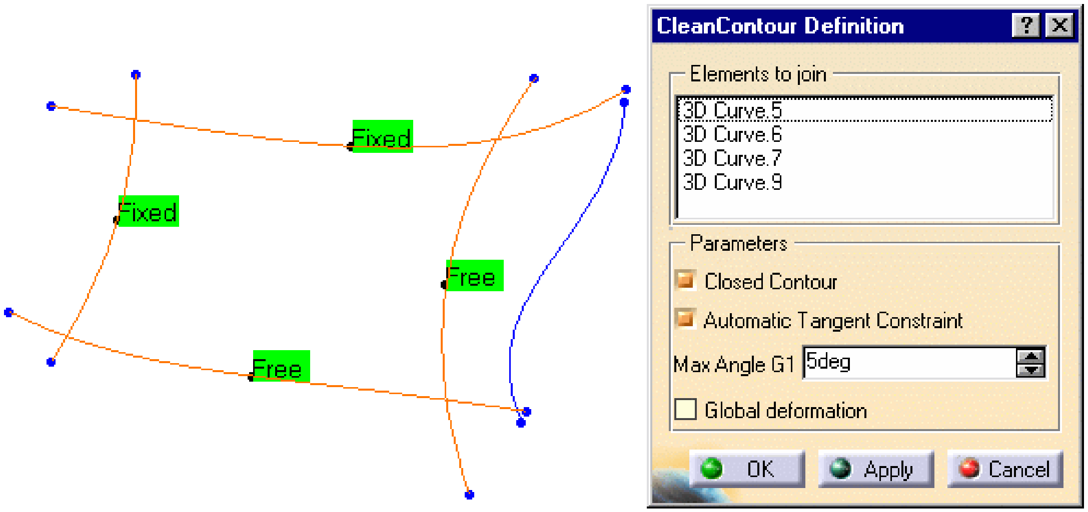
or

a. Use the **Undo/Redo** function to deselect the curves you have previously selected.

or

a. Pick the curve in the graphic zone. It is removed from the list of the dialog box and is no longer highlighted.

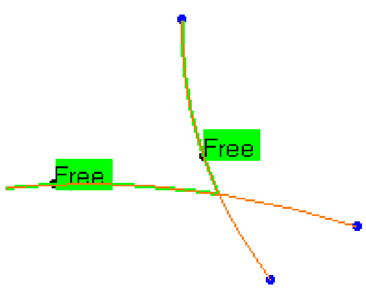
If necessary, you can pick further curves to complete the selection. The dialog box is updated accordingly:



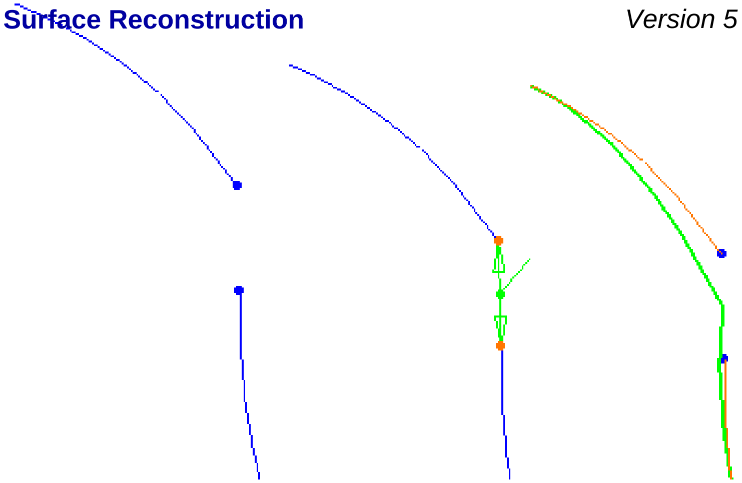
The **Undo/Redo** function is no longer available once you have used one of the two other methods.

The CleanContour computation is based on the minimum distance of the curves. The deformation allowed may not exceed the hole size. Priority is given to the parametric restriction over the deformation.

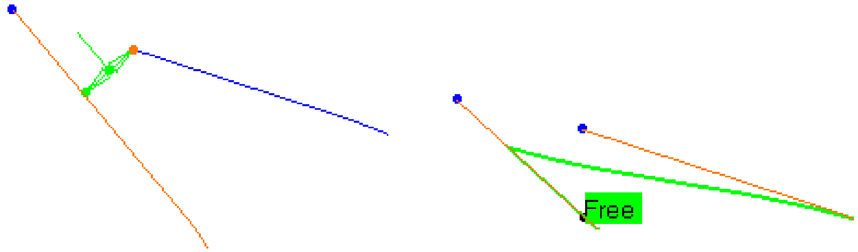
- if the points of the minimum distance between two curves are no end points of the curves, the curves are restricted.



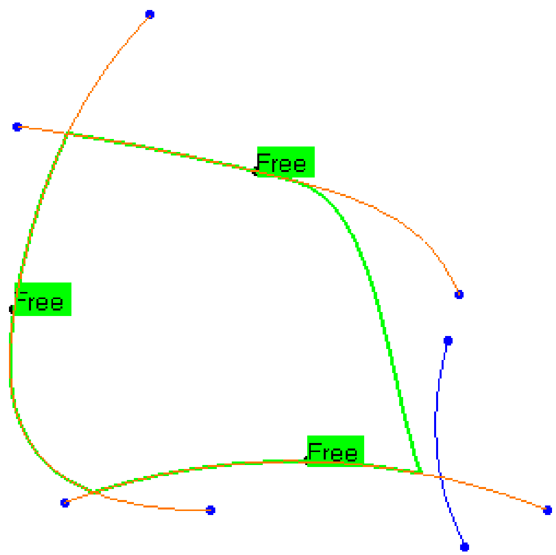
- if the points of the minimum distance between two curves are end points of the curves, a point is computed on the segment corresponding to that minimum distance, weighted by the length of each curve :

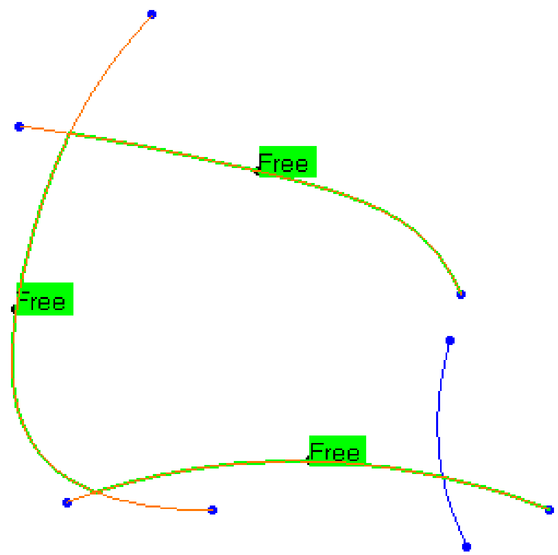


- if only one end point of a curve is a point of minimum distance, this end point is moved to the point of minimum distance on the other curve.



3. Check the **Closed Contour** option to close the contour, according to the rules above:

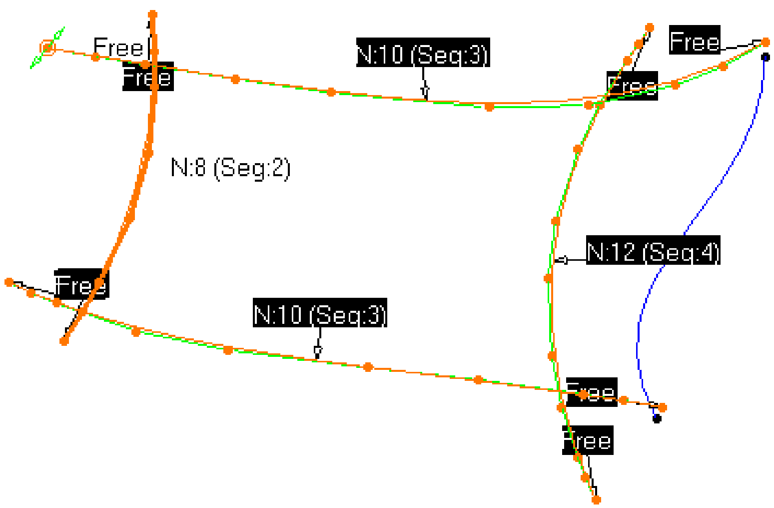




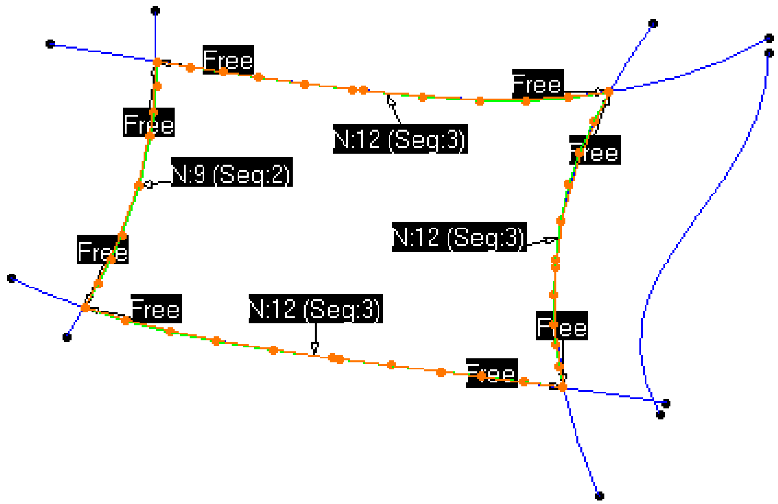
4. Check the **Automatic Tangent Constraint** to force a tangency constraint on the curve end points when the angle of the tangents at those ends is lower than the **Max Angle G1** value.

5. The curves are deformed to achieve a CleanContour. By default, this deformation is local:

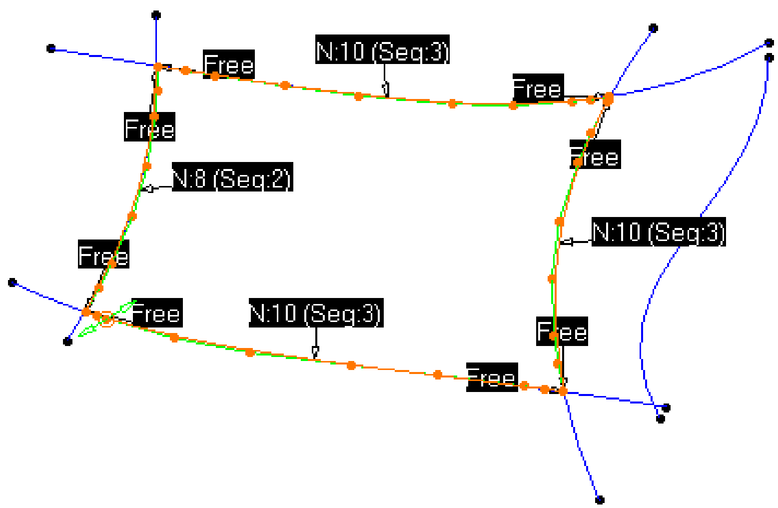
Input curves



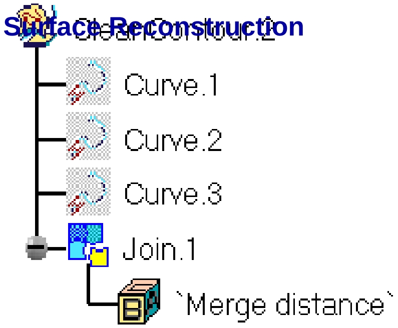
The deformation is local and not distributed along the whole curves.



If you check the Global deformation option, the deformation is more evenly distributed on the whole curves. The degree and the structure of the curves are kept.



6. Click **Apply**: a proposed CleanContour is displayed in green. Click **OK** to validate. A **CleanContour.X** body containing a **Join.x** element, the editable merging distance parameter and the hidden modified or deformed curves (input of the join) is created in the specification tree. The CleanContour is created, the input curves are hidden.


Curve.1
Curve.2
Curve.3
Join.1
`Merge distance`

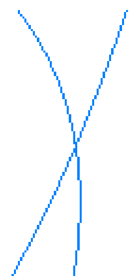
Curves Network

This task will show you how to create a network of constrained curves to be used in the surfaces network command.

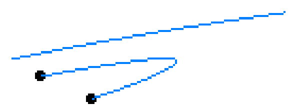
On a first step, you have created characteristic curves on the cloud. These curves are often approximate and require some preparation:

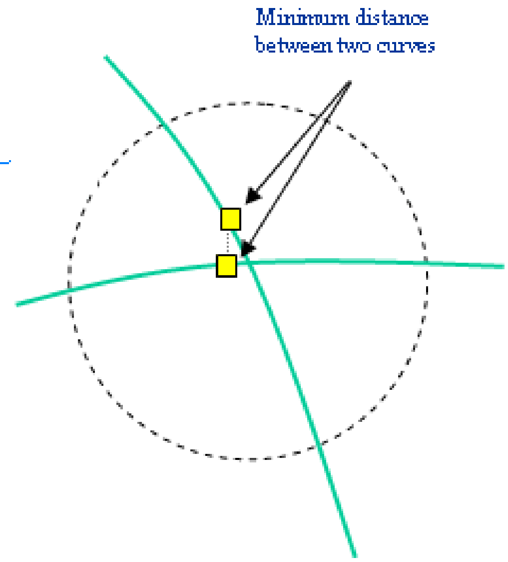
- Slicing that cuts curves or edges in several pieces, according to a pseudo-intersection: there is a pseudo-intersection between two curves if they intersect each other in the view direction (but not really), and if the mini 3D distance between them at this cutting point is lower than the parameter **Max. Distance**.

Pseudo-intersection of two curves in the view direction



in another view.



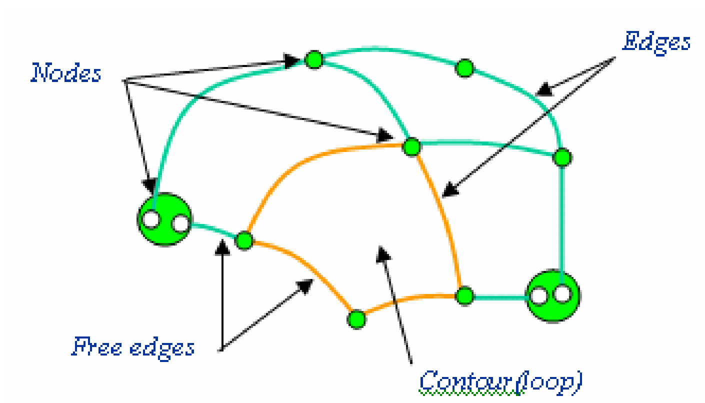


- Deburring to remove undesirable small curve pieces, based on a Min length criterion.

Then, you can create a network from these cleaned curves according to the following:

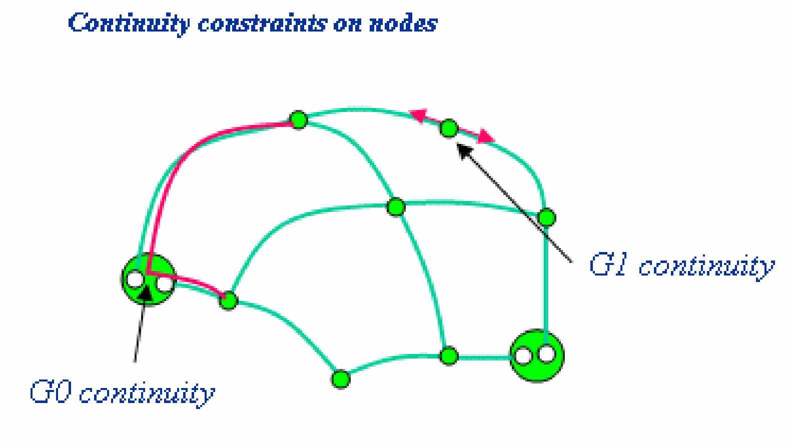
- A network is a set of closed and connected contours, named wires,
- a contour is a set of connected edges, it is necessarily closed but not limited to 3 or 4 sides,
- an edge belongs to one contour (free border edge) or 2 contours (common edge),
- if there are no free border edges, the network is closed,
- a node is the topological encounter point of two or more edges. These edges may belong to different contours.

Once the action has detected the topology (node, contours, free border edges),



the curves are adapted to the network internal constraints such as:

- Connection constraints on nodes (all edges ending on a common node must be connected on this point).

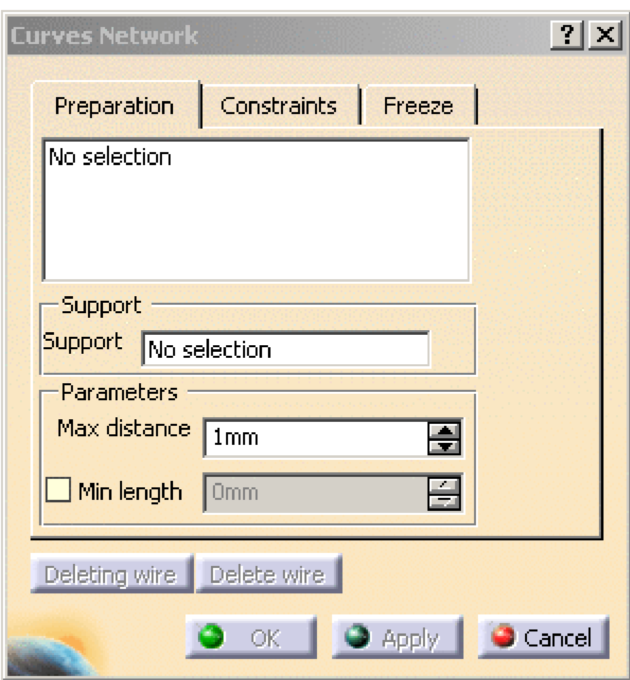


- Tangency constraints between two edges on a node.

Open the SurfNetwork.CATPart from the samples directory.

1. Click the **Curves Network** icon. The dialog box is displayed. It consists of 3 tab pages:

    - Preparation,
    - Constraints,
    - Freeze.

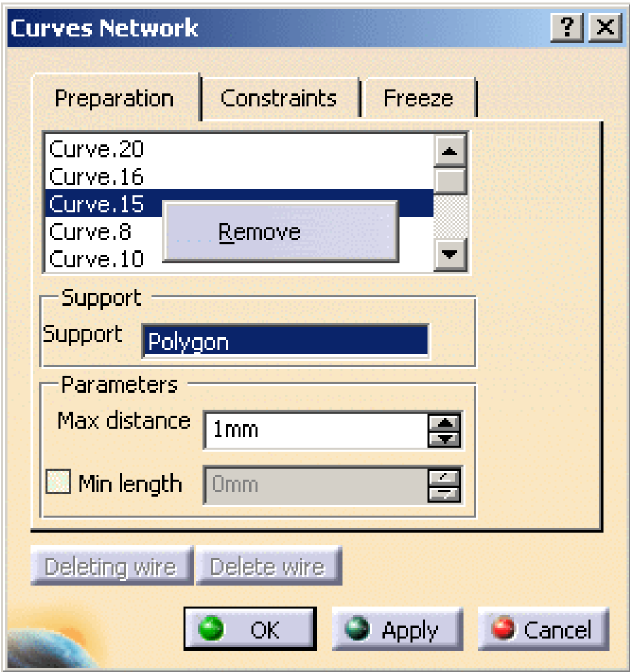


2. Stay in the **Preparation** tab and select the curves to clean. They are displayed in the dialog box.

You can select a CleanContour, a Curves network or a Join of curves, their curves will be added to the list.

To remove a curve from the list:

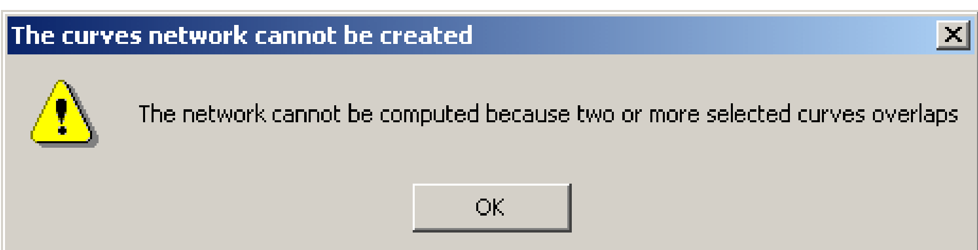
- pick the curve again in the graphic area, or
- pick the curve again in the specification tree, or
- pick the curve you want to remove in the list of the dialog box and use the contextual menu **Remove**.



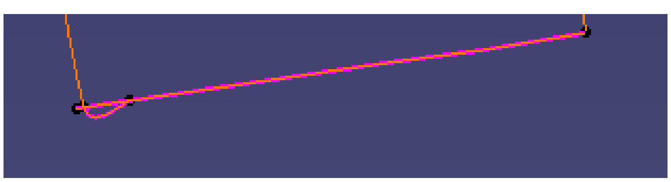
You can also select additional curves, they will be added to the list.

3. Select the mesh as the **Support**. Click Apply.

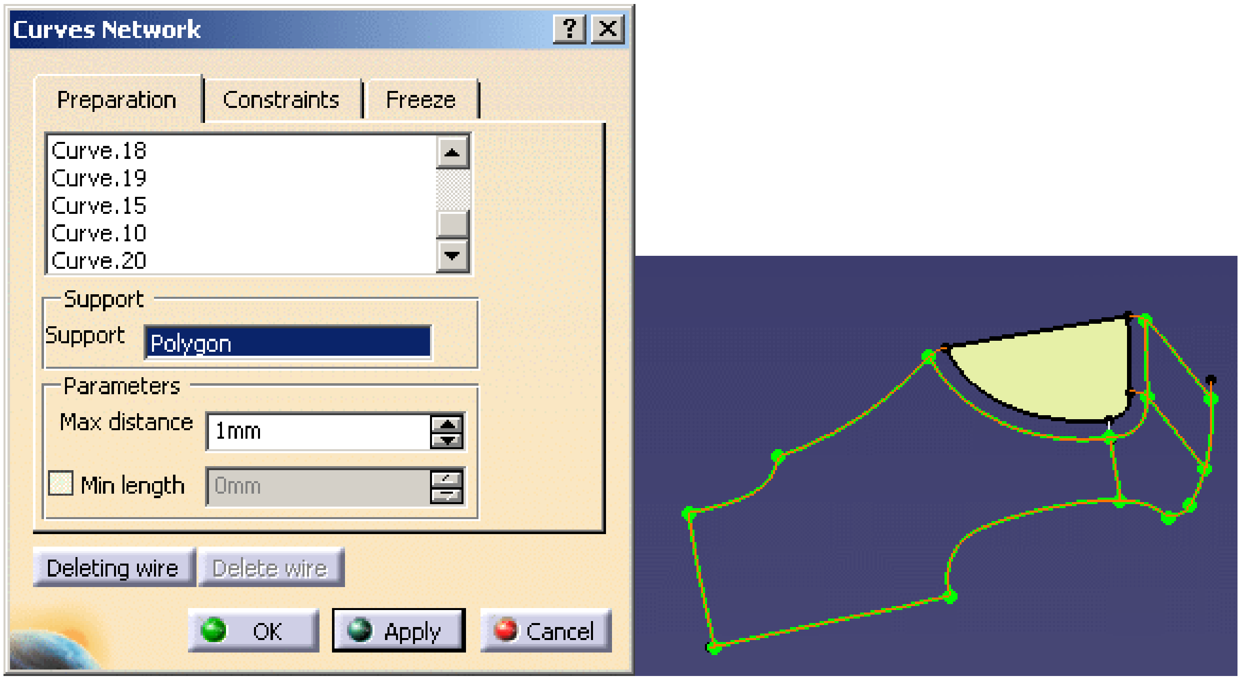
Before computing the network, the action searches overlapping curves. If some are detected, a message is displayed:



The overlapping curves are displayed in magenta:

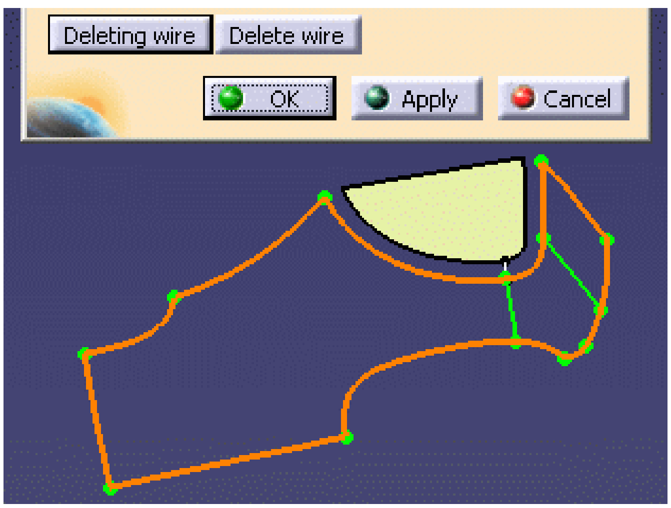


The wires are then computed and displayed. The push button **Deleting wire** is now available.

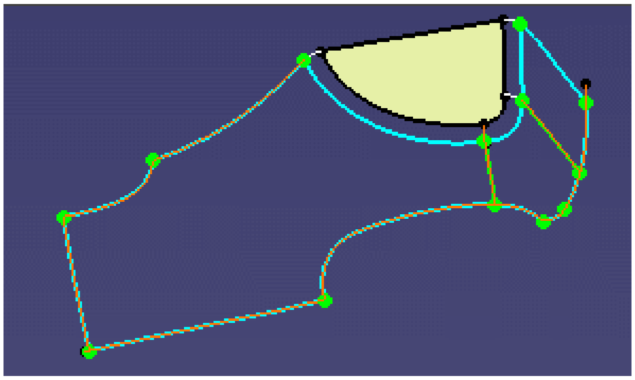


A wire is computed on each "loop" of curves, plus a large one on the "loop" formed by the external curves because the network is considered as closed by default. If your network does not form a closed volume, you usually do not need this large wire (it would create an additional surface as you create surfaces from the curves network). In the same way, you can remove wires corresponding to holes in your part.

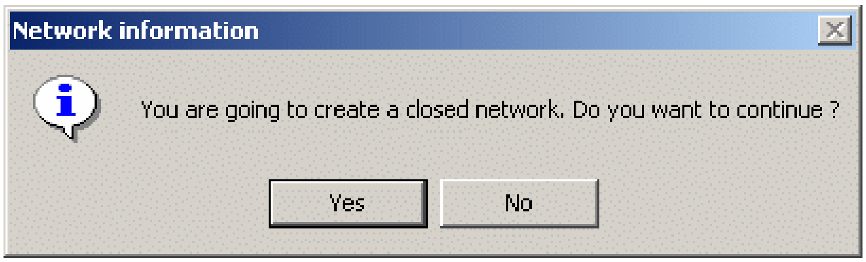
4. Push the **Deleting wire** button to activate the wire selection. The large wire is highlighted and a **Delete wire** button becomes available.



5. Push the **Delete wire** button to remove this wire.

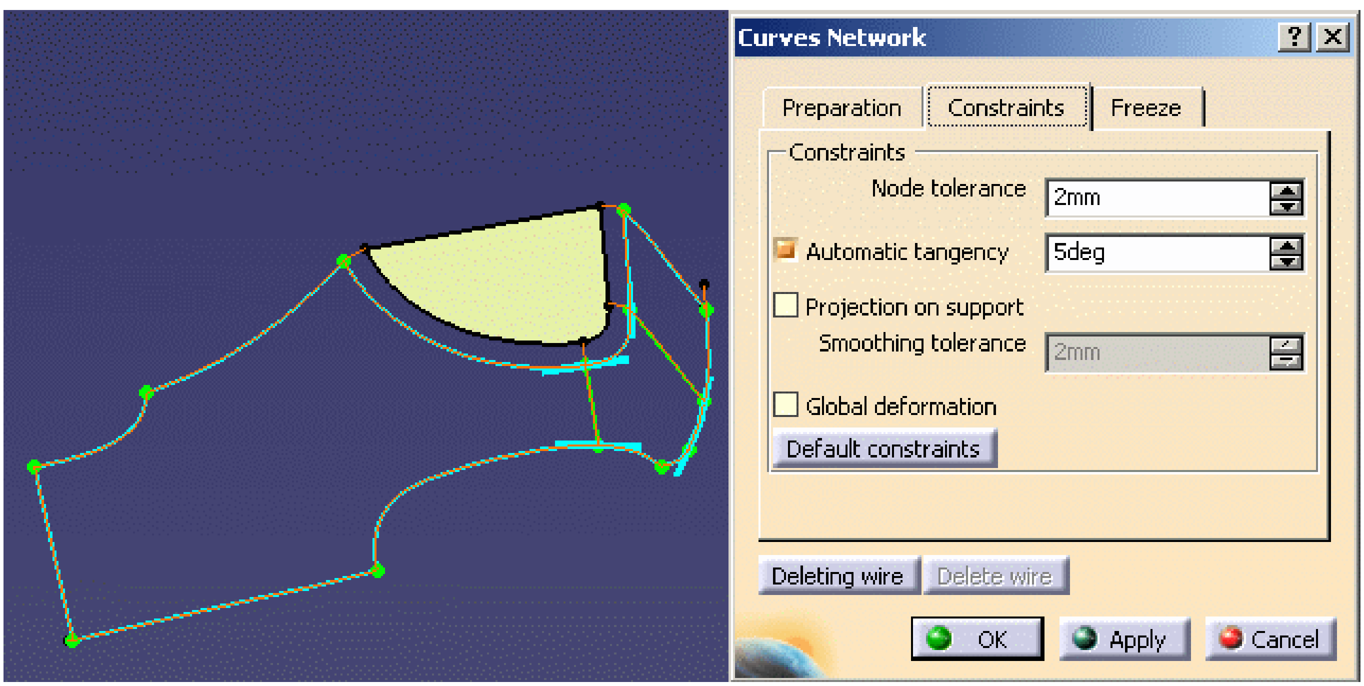


- In the cases where you want to create a surface on a network of curves with a hole in it, i.e. without filling one or several wires, use the **Deleting wire** button to activate the wire selection, select the wire by picking two of its edges, and push the **Delete wire**.
- If you do not remove the large wire, an information message is displayed as you push OK.



The parameters Max distance and Min length are those used in the Curves Slice command.

6. Go to the **Constraints** tab to define constraints. The existing constraints are highlighted, as a green dot for continuity constraints on nodes, as a blue line for tangency constraints, as a red curve for fixed curves.



The constraints are applied globally on the whole network.

Continuity constraints on nodes are implicit and can not be removed.

The **Node tolerance** defines the maximum distance between edges extremities to consider these edges connected on a node. You can modify this value, but it can not be smaller than the **Max distance** parameter.

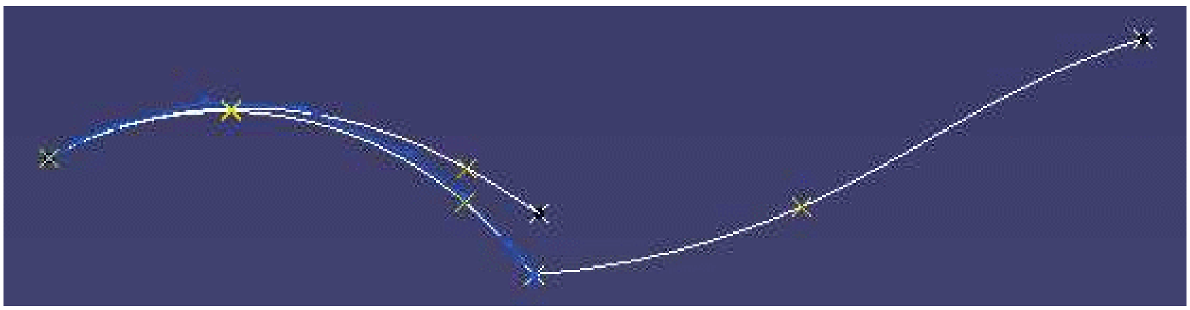
The **Node tolerance** is visualized as a green sphere whose radius is equal to the **Node tolerance**. The size of this sphere is updated when you modify the **Node tolerance** value. You can move the sphere using the cursor.

Tangency constraints are activated by the **Automatic tangency** check box. You can edit the value of the tangency angle. Push **Apply** after each modification.

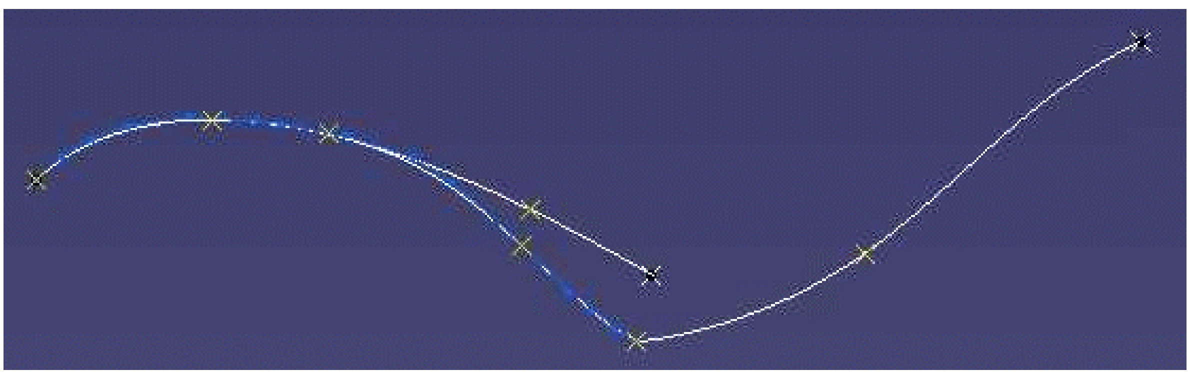
Check **Projection on support** to compute the network from the projections of the curves on the polygon. The curves projected are smoothed, with the **Smoothing tolerance**. This parameter can be edited. It is applied to all curves.

You can choose between a global or a local deformation constraint (available for all deformable edges). This constraint is applied during the geometric adaptation step. If the **Global deformation** checkbox is set, edges deformations are done along the entire length of the edges.

Example of global deformation:



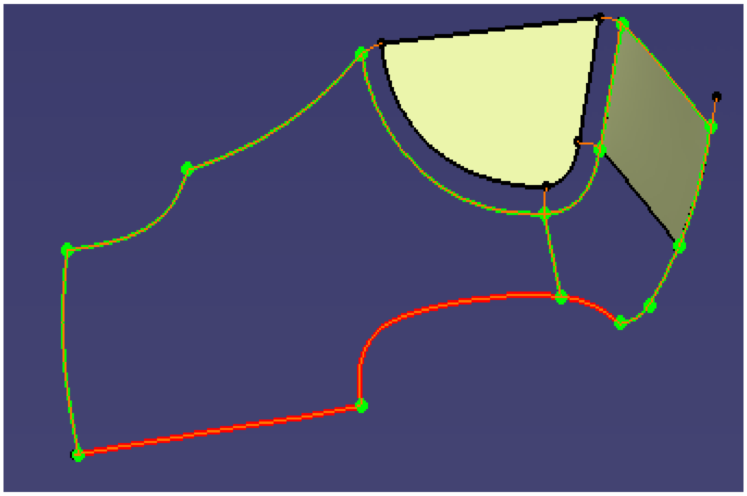
Example of local deformation:



Push the **Default constraints** to revert to the default constraints.

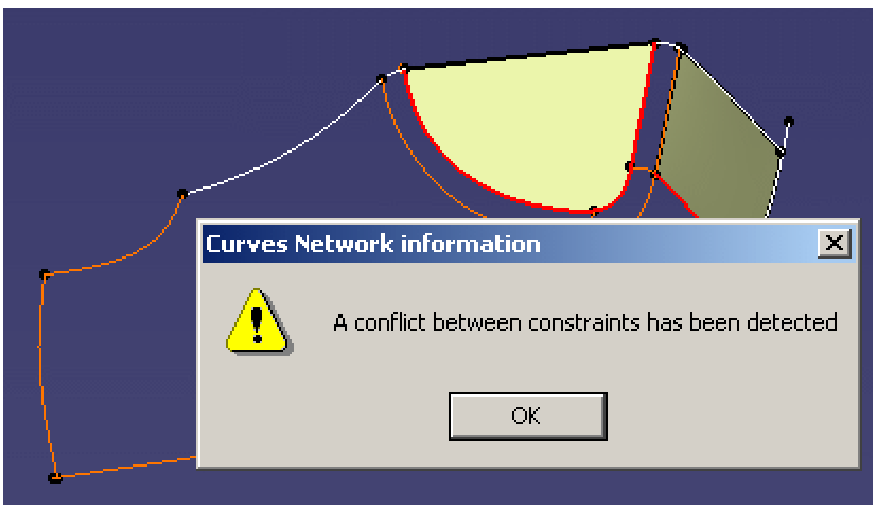
7. Go to the **Freeze** tab page to select curves that must remain fixed.

Pick a curve to fix it. It is displayed in red in the graphic area and displayed in the dialog box. Pick it again or use the contextual menu **Remove** to free it.

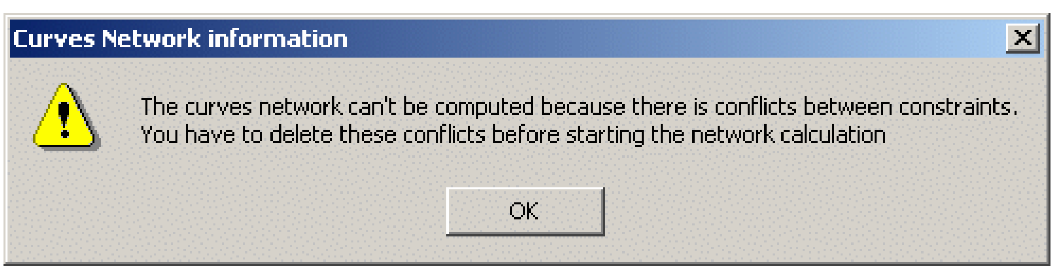


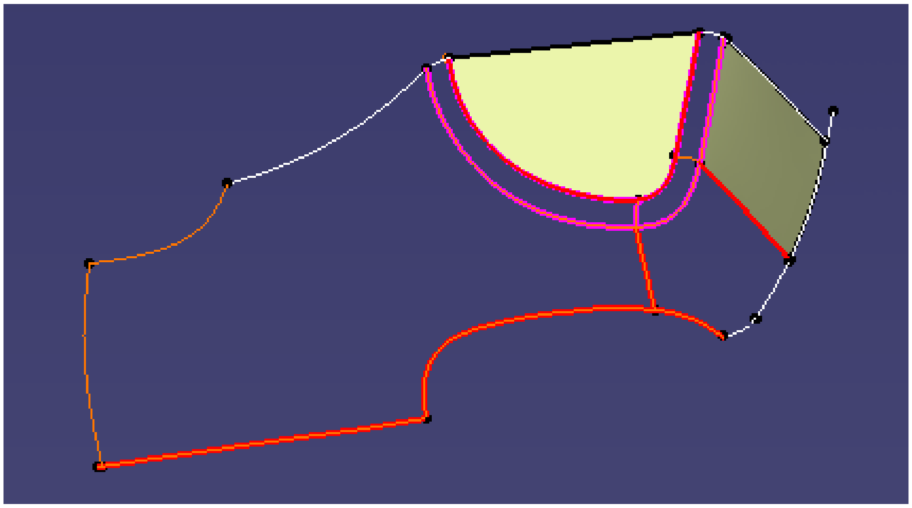
- You can freeze only curves that have been selected in the preparation tab.
- An edge of a face is automatically and definitely fixed.
- Free border edges are fixed by default but you can free them.

The action provides a constraint solver to search conflicts between constraints.



A message is displayed at the end of the search. When all constraints have been checked, conflicts are highlighted in magenta.





Modify the constraints to solve the existing conflicts before creating the wires.

7. Push OK to validate the network and exit the dialog box. A **CurvesNetwork** element is created in the specification tree.

Surface Creation

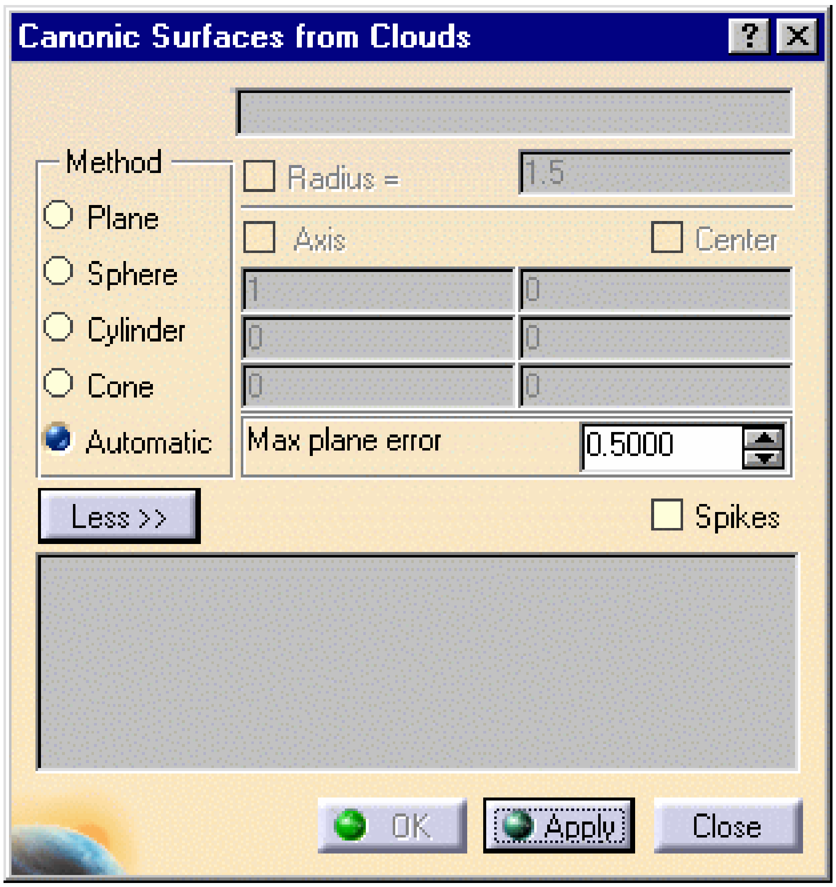
This chapter deals with the creation of surfaces in Quick Surface Reconstruction.

Basic Surface Recognition

This task shows you how to recognize the basic shapes of a part and how to create the corresponding editable surfaces.

Open the Canonic1.CATPart from the samples directory.

1. Click on the **Basic Surface Recognition** icon . The **Basic Surface Recognition** dialog box is displayed.



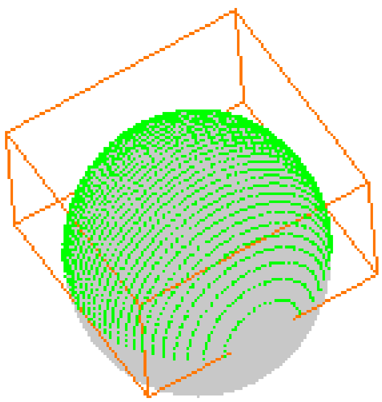
2. First identify visually the portions of the part made of the basic surfaces, i.e. planes, cylinders, spheres, cones. Then activate each of those portions in turn and let the application recognize and create the surface. The name of the cloud selected is displayed at the top of the dialog box.

The **Activate** action is available within the **Basic Surface Recognition** action:

- Pick the cloud of points or polygon as you enter the **Basic Surface Recognition** action.
- Start the **Activate** action and pick a cloud of points or polygon (the one you have previously picked or another if need be).
- Activate the area you want to work on.
- Exit the **Activate** action.
- Pick the area you have just activated and start the Basic Surface Recognition.

The portion of the cloud of points or polygon to recognize can also be activated before entering the **Basic Surface Recognition**.

3. Select the type of surface to detect or choose the **Automatic** option and click **Apply** to visualize the shape.



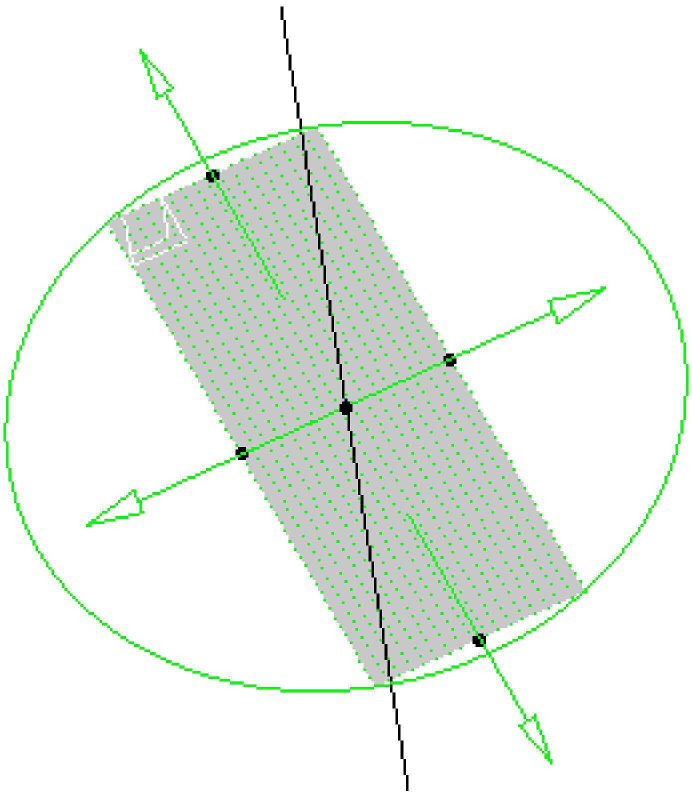
For each type of surface, you can choose to let the application compute the surface or you can set some data:

- For a plane: the normal to the plane and its passing point,
- For a sphere: the center and the radius,
- For a cylinder: the radius, the axis and the center,
- For a cone: none,
- For the automatic option: the maximum plane error.

Those data are also displayed in the dialog box when the surfaces are computed by the application.

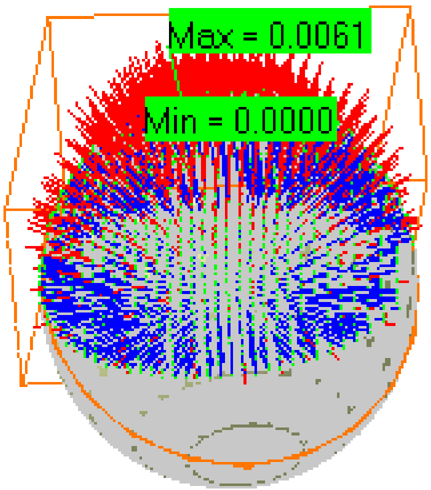
You can activate the corresponding fields by checking their names and edit the values as necessary before creating the surfaces.

- If you choose a shape type to recognize, you can edit its geometric properties (Axis, Center, Radius) by activating the corresponding check boxes and entering new values.
- If the shape recognized is a plane, you can edit it graphically, using the extension (arrows) and rotation (circle) manipulators.



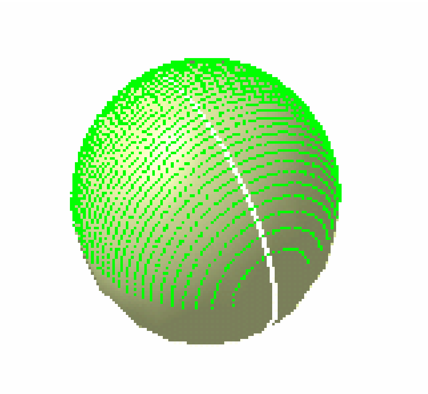
- If you choose the **Automatic** option, you can not modify the geometric properties of the shape directly. Once the shape has been recognized, check the corresponding shape type to make the corresponding properties editable.
- Click **Apply** to take those modifications into account.

4. Check the **Spike** option to display the deviation between the canonic surface and the original part.

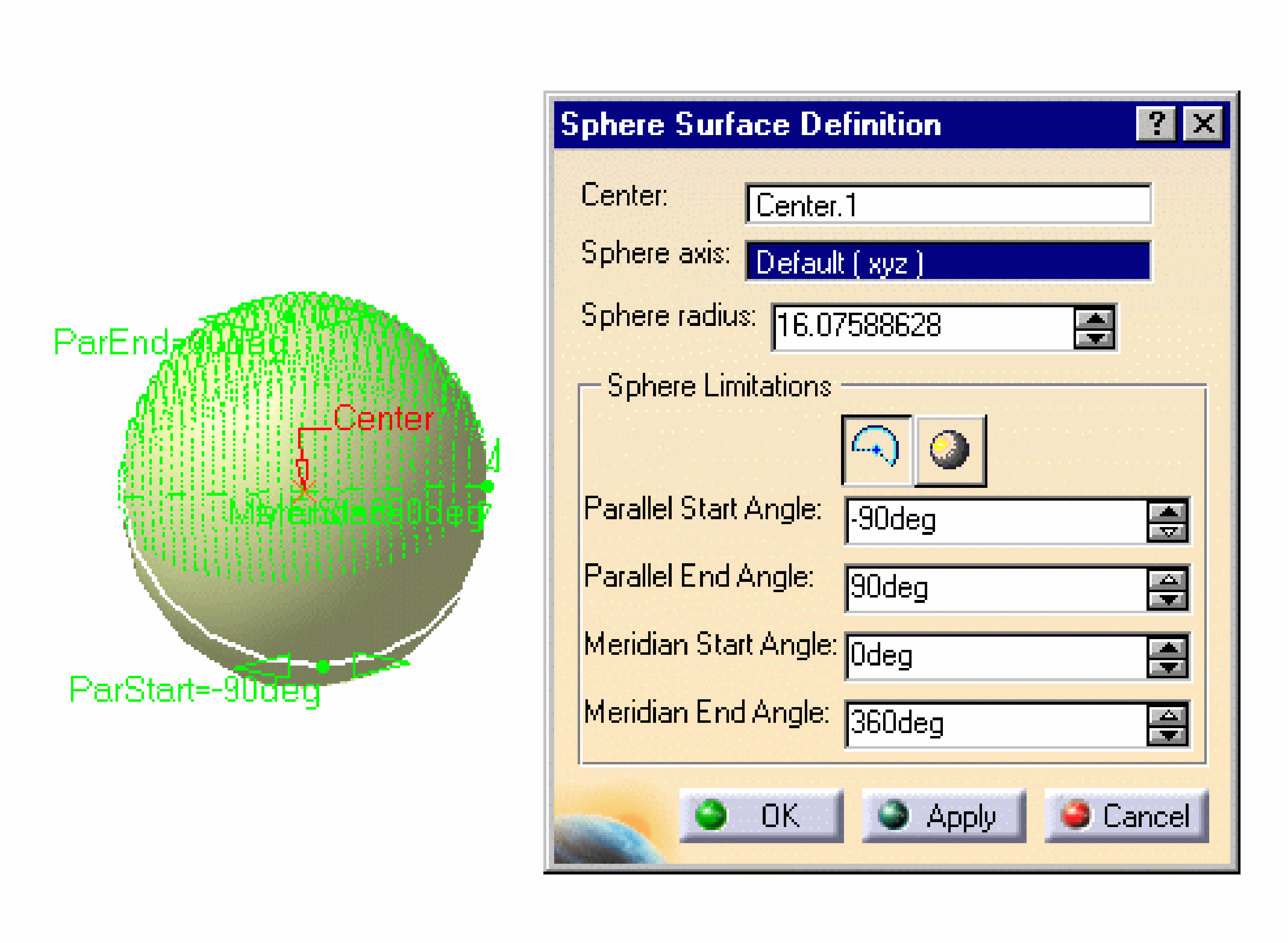


5. Push the **More>>** button to display the statistics on the action.

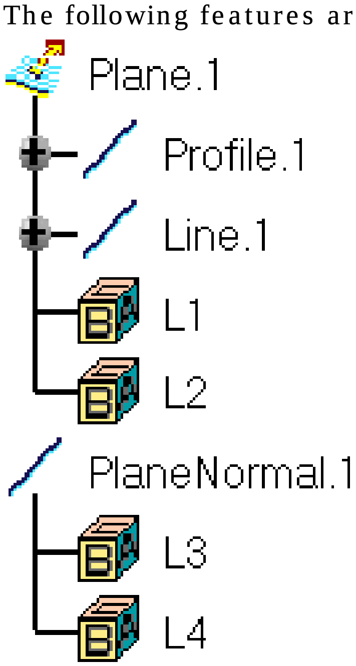
6. Click **OK** to create the shape.



7. Double-click the shape to edit it, if required:

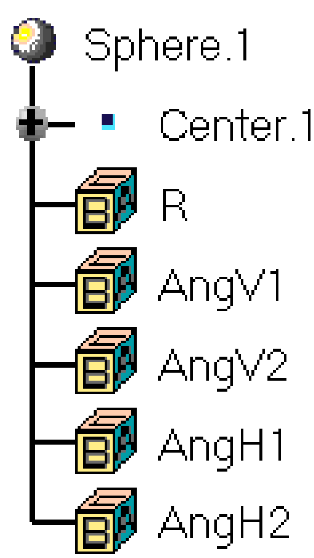


The following features are created in the specification tree:

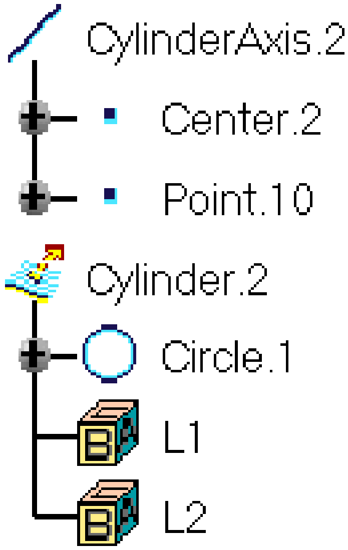


An extrusion element named **Plane.x** with its components,

The normal to that plane named **PlaneNormal.x** and its components

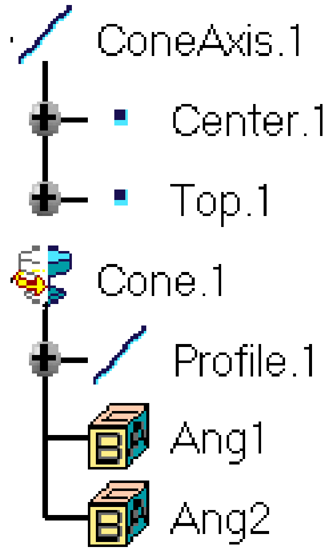


A feature sphere named **Sphere.x** and its components



An extrusion element named **Cylinder.x** with its components,

The axis of that cylinder named **CylinderAxis.x** and its components.



A rotation element named **Cone.x** and its components,

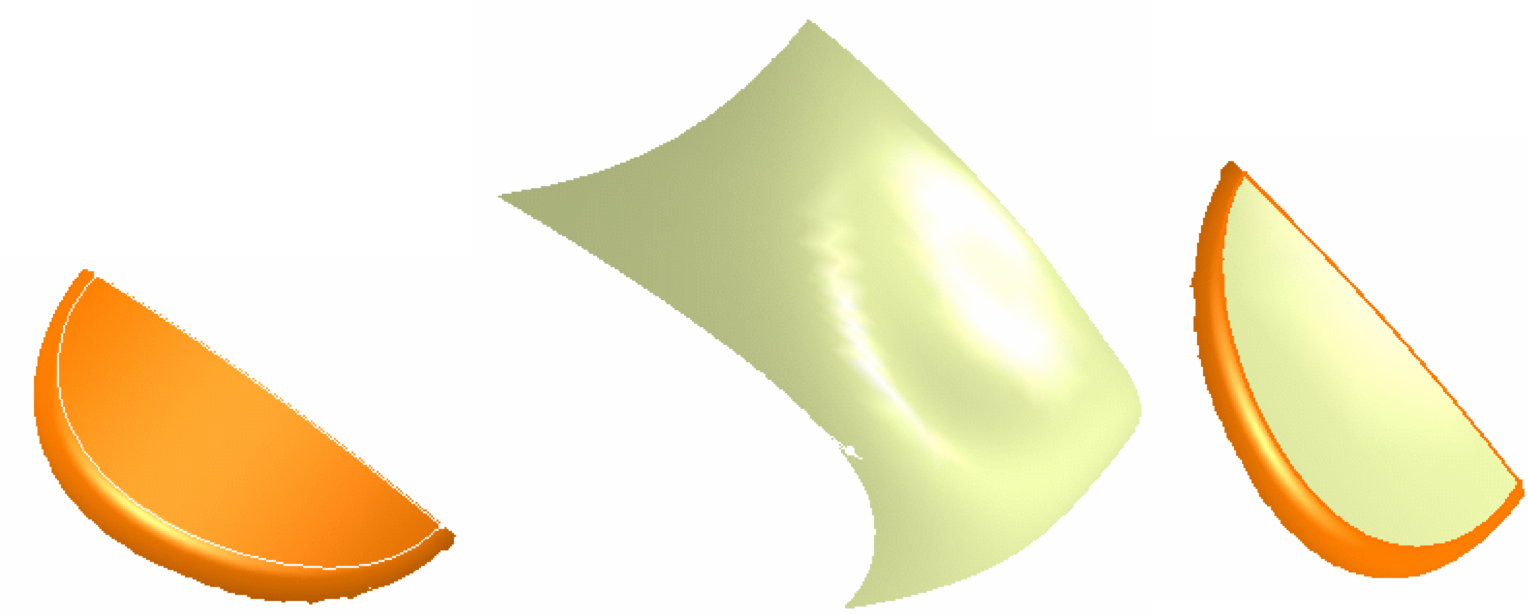
The axis of that cone named **ConeAxis.x** and its components.

PowerFit

This task shows how to use PowerFit.

PowerFit is used to create:

- an untrimmed surface supported by one cloud of points or one polygon and/or curves,
- a surface supported or not by one cloud of points or one polygon and trimmed by an external boundary.

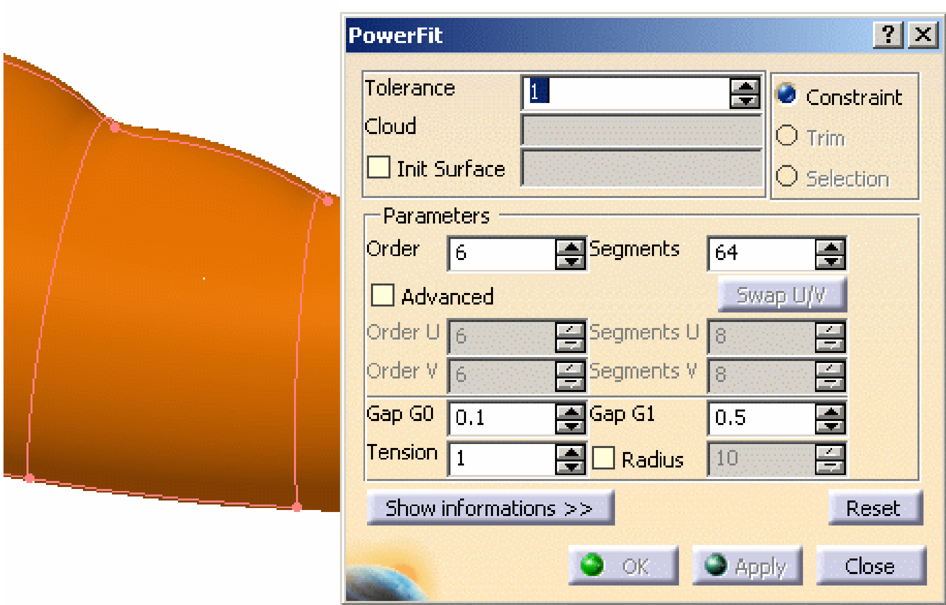


Original polygon and boundary | Untrimmed surface supported by the polygon | Surface supported by the polygon and trimmed by the boundary

The dialog box labels have been updated: Point Gap has been replaced by Tolerance and Number by Segments.

Open the PowerFit1.CATPart from the Sample directory. This model consists of a polygon (Polygon) and a join (Join.2).

1. Click on the **PowerFit** icon . The dialog box is displayed:



This action is modal: the values used are re-displayed the next time you open this dialog box.

2. Enter the polygon and/or the curves to process, set the constraints and parameters, click **Apply** to preview the result, and **OK** to validate it. A

Surface.x element is created in the specification tree. The segmentation display is erased.

Below, you will find explanations on:

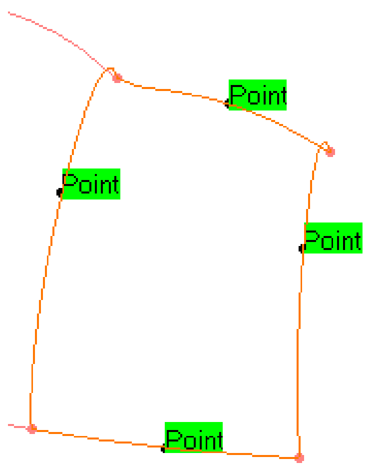
- Input elements,
- Constraints,
- Parameters,
- Information,
- Counterdraft detection.

Input elements

- Select the element(s) to be processed:
  - either a cloud of points or a polygon or a portion of these,
  - or curves,
  - or both.

Clouds:

You may process only one cloud or one polygon at a time. Once selected, it is sent in the NoShow space.



Curves:

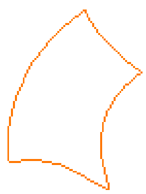
The curves form either a closed set of curves or an open set of curves with only one hole in it. You can make the curves continuous either during their construction or using the clean contour action.

You can either select a whole clean contour from the specification tree, or by picking one of its vertex (the clean contour is highlighted), or pick curves (the curves are highlighted).

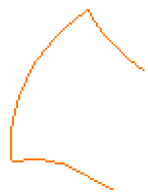
If one curve of the clean contour is linked to a face, the tangency continuity is not proposed if you pick the curve. It is proposed if you pick the clean contour.

Those input curves are not modified and are sent to the NoShow space.

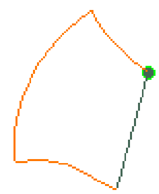
If a trimmed surface is created, the edges of the face will be the curves computed from the input curves.



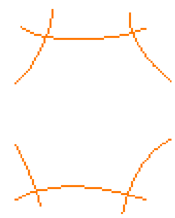
Closed set of curves



Open set of curves



Open set of curves with its topological closing line (no curve is created as such, but as the edge of the trimmed surface, if any)



This type of open set of curves is not accepted:

- Check the **Init Surface** box if you want to enter one. The init surface helps the computation by giving the shape of the result surface. You can either enter it yourself or let the application compute it in the direction of the largest curve. If you select an init surface, its name is displayed in the field **Init Surface**.

The init surface must be larger than the domain to process.

- **Tolerance** is the mean maximum deviation between the surface created and the cloud of points or polygon, i.e. the deviation may be higher at some places. This field is editable.

Constraints

- Choose the way the input curves are taken into account:

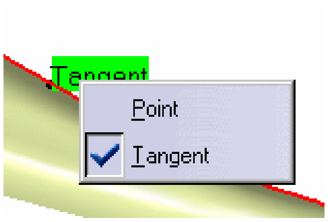
  - as **Constraint**: the computed surface will go through them,
  - **Trim**: the surface is computed, then the curves are projected onto it to trim it.
  - or **Selection**: the computation is based on the points located inside the curves.

Continuity may be requested on the input curves regarding the surface to create:

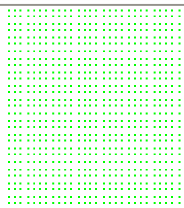
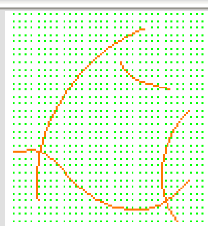
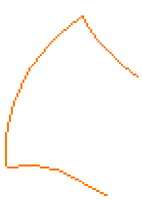
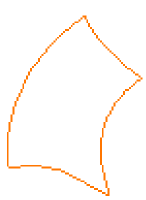
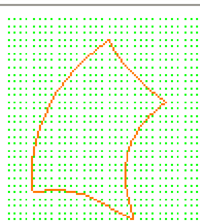
- G-1: free. Applies to the **Trim** or the **Selection** options.
- G0: point continuity. Applies to the **Constraint** option.
- G1 : tangent continuity. Applies to the **Constraint** option.

By default, the continuity requested in the Join action are proposed by the **PowerFit** action.

You can change them by simply clicking on the text or using the contextual menu.



- You may want to select a join. If this join contains a sliced surface edge, or a split CleanContour that contains a sliced surface edge, with a tangency constraint that you want to keep, pick the curves one by one, graphically, i.e. do not select a join by picking one vertex, nor select the elements in the specification tree.

In short:	Input	Information	Output
	Points or polygons		The surface is computed on the points. It is not trimmed.
	Miscellaneous curves	No outer boundary. No points.	This case is not dealt with.
	Miscellaneous curves + points	No outer boundary. Points.	This case is not dealt with.
 or 	Curves (Outer boundary). No points.	Option: **Constraint** Possible constraints: G0, G1.	The surface is computed on the curves. The curves become the edges of the surface.
 or 	Curves + points	Option: **Selection** Possible constraint: G-1	The surface is computed on the points located inside the curves. The surface is not trimmed.
		Option: **Trim** Possible constraint: G-1	The surface is computed on the points. The curves are then projected on the surface to trim it.
		Option: **Constraint** Possible constraints: G0 or G1	The surface is computed on the points and the curves. The curves become the edges of the surface.

Parameters

- **Order, Segments:**

These parameters apply globally to the surface computed. They are maximum values. The actual values are computed automatically by the action.

PowerFit creates a NURBS surface, controlled by the tolerance (i.e. **Tolerance**), the number of segments and their order. Whenever possible, this surface consists of one single segment, otherwise, it is made of several segments. This surface may then be trimmed by the curves.

You can increase the order of the segments, thus reducing their number, or vice-versa.

If the number of segments is x, this means that the surface computed will consist of a maximum of x segments, or less. The default number of segments is 64, the maximum number is 2048.

If the order of segments is y, this means that each segment will have a maximum number of y control points in each direction, or less. The segment order may vary from 3 to 15.

- **Advanced**:

You may want to impose an order and a number of segments in both U and V direction. To do so, check **Advanced**. The **Order** and **Segments** fields above are no longer available. You can edit the fields below to :

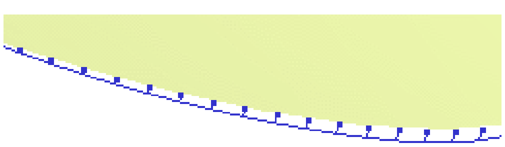
- enter the number of segments in each direction,
- enter the order of segments in each direction,
- swap the values in U and V.

Click **Apply** to restart the computation.

- Increasing the order of the segments may result in an oscillating surface, even if this is not visible.
- Push the **Show Information** button and check the **Segmentation** option to display the segmentation of the computed surface.

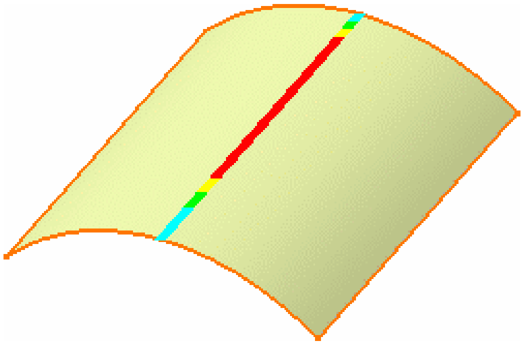
- **Gap: G0:**

this is the distance between the surface and the boundary curves. Since there is more noise on points than on curves, the **Tolerance** may be higher than the **GO Gap**. The default value is 1.



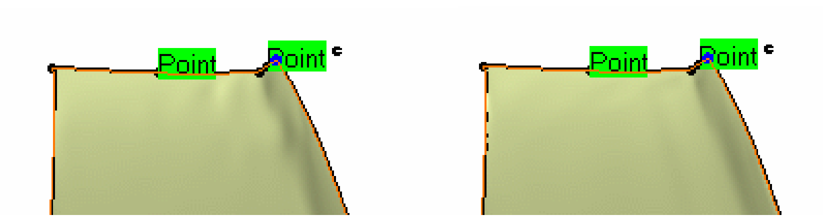
- **Gap: G1:**

this is the tangency tolerance between two contiguous surfaces (in blue below). The default value is 0.5.



- **Tension:**

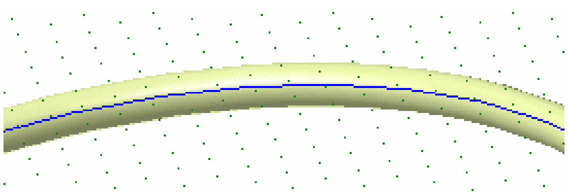
Possible values are between 0 and 4. Use a higher value to have a smoother (but less tense) surface.



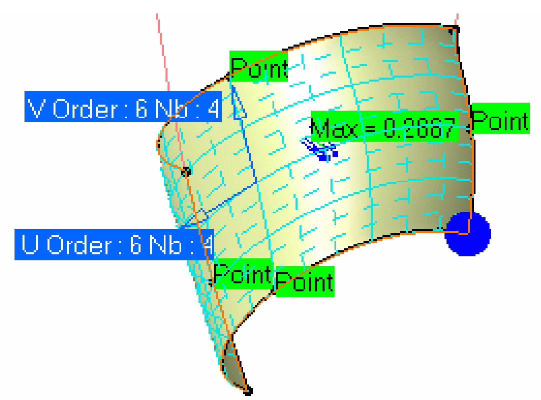
Please note that since the shape is constrained by the points, the effect of this parameter is limited.

- **Radius:**

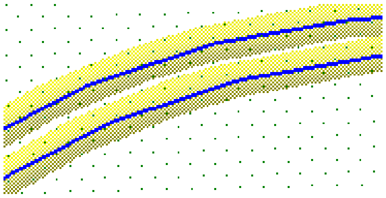
when the cloud of points is noisy, it is difficult to have the surface going through all the points and the curves (risk of undulations). The points inside a circular pipe centered on the curve are deleted, and you may want to set the radius of that pipe.



When you check this option, a blue sphere is displayed on the extremity of the first curve, representing this radius (if you have selected at least one curve and a cloud of points or a polygon).



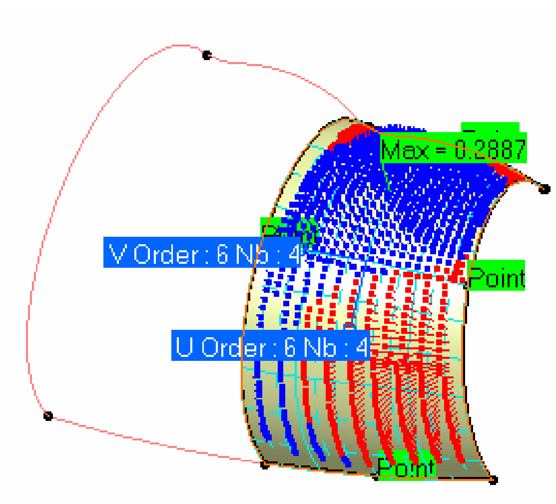
If two curves are not distant enough, all the points between them may be deleted, making the computation of the surface impossible.



Information

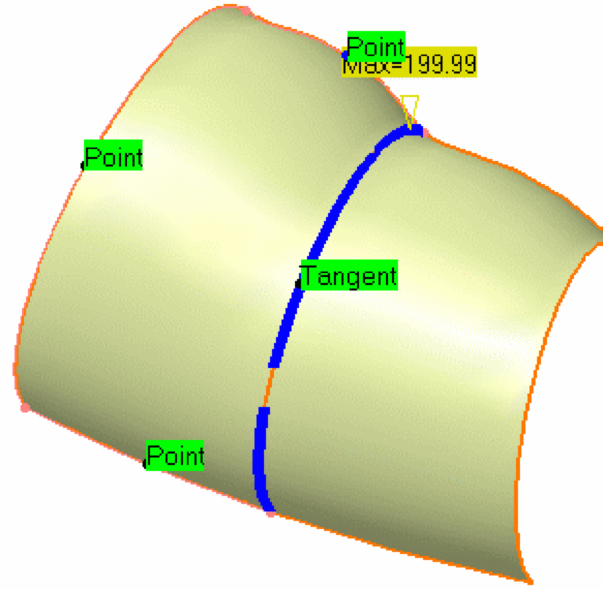
Push the Show information button to check the required options and display statistics.

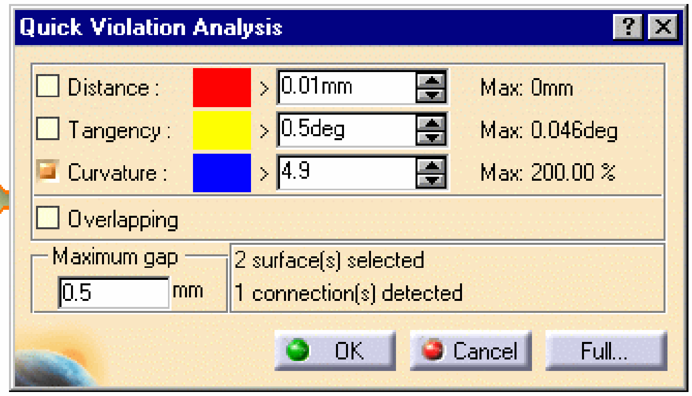
- Check the **Spikes** option to display the deviations.
- Check the **Segmentation** option to display
  - the segmentation on the surfaces computed,
  - the number of segments and order in U,
  - the number of segments and order in V.
- Use the **Deviation** field to enter the value above which the deviation spikes will be displayed. When you first enter the action, the **Deviation** value is the same as the **Tolerance.** Once a surface has been computed, the **Deviation** value is the computed one.



You must first click **Apply** to compute the surface and to display the deviation spikes.

- You can use one or more edges of an existing surface to compute a new surface with PowerFit. Check the **Connect Checker** option to display the connection analysis between the existing surface and the new surface.





The **Connect Checker** is not available on surface edges that have been previously sliced.

- You must first click **Apply** to compute the surface and to display the connection analysis.
- Click on **OK** or **Cancel** to exit the **Quick Violation Analysis** and return to the PowerFit dialog box .
- Click on **Full** to display more analysis options.
- More information is available in the Connect Checker section.

Information on the points for the parameters taken into account by the computation are available in the box at the bottom of the dialog box (no dynamic display):

Max Deviation = 2.643
Mean Deviation = 0.1803
Standard Deviation = 0.2409
For 98% points, deviation < 1

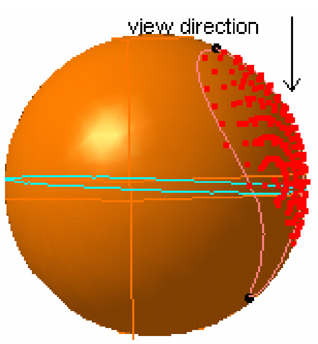
- the maximum deviation found between the points of the cloud and the surface,
- the mean deviation found between the points of the cloud and the surface. This deviation should be as small as possible.
- the standard deviation, i.e. the dispersion of the points around the mean deviation. A small standard deviation indicates that most points are within the mean deviation, i.e. that there are only few outliers.
- the percentage of points of the cloud that are below the mean deviation.

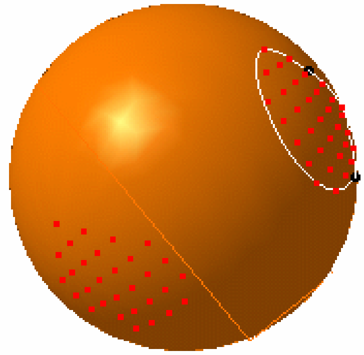
Counterdraft detection

- Generally, when a counterdraft or opposite points are detected, a message is displayed in the dialog box, and no surface is computed. You can solve the problem by activating a portion of the cloud of points and restarting the computation.
- When PowerFit is used to compute a trimmed surface from the points of a cloud of points and curves that form the surface boundaries, these curves define a prismatic trap with infinite height and a view direction. The points used for the computation are all the points contained in this trap. Therefore, counterdraft or opposite points may alter the computation.

Counterdraft:

Opposite points:





PowerFit can select the required points up to a certain level:

- This is possible with polygons, not with cloud of points.
- All triangles considered belong to the same polygon and their orientation is coherent.
- The points are separated into two zones, according to the direction of the normal of the triangles. This selection is easy in the above cases (the equator line is the separation between the two zones for the counterdraft, for the opposite points, the zones are already well delimited). The incorrect zone is not taken into account in the computation.
- The selection is harder in such cases:

According to the normal of their triangles, the points belong to two zones, shown in black and in yellow in our example. The points surrounded in black form one zone, the points surrounded in yellow form another zone, both zones contain points that should not be taken into account. Such cases are not yet dealt with successfully.

- In short, the points that PowerFit recognizes as unwanted for the computation are correctly eliminated. However, some points may still be taken into account, whereas they should not. This may lead to a defective result. In such cases, you should activate yourself the requested zone, with the Activation command.

Creating Multi-sections Surfaces

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

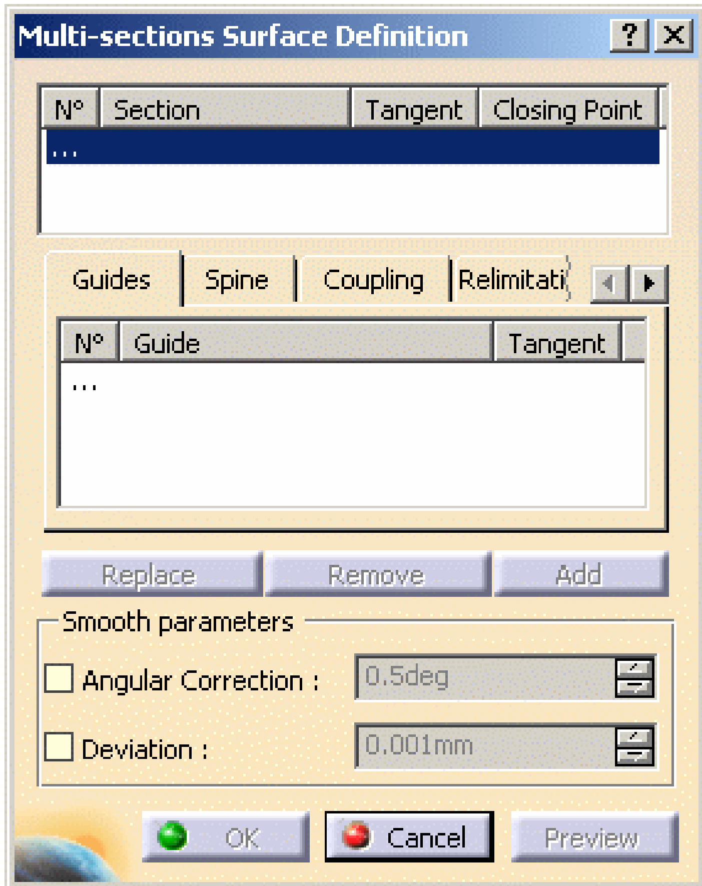
- Relimitation
- Planar Surface Detection
- Coupling

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

Open the Loft1.CATPart document.

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

   The Multi-sections Surface Definition dialog box appears.



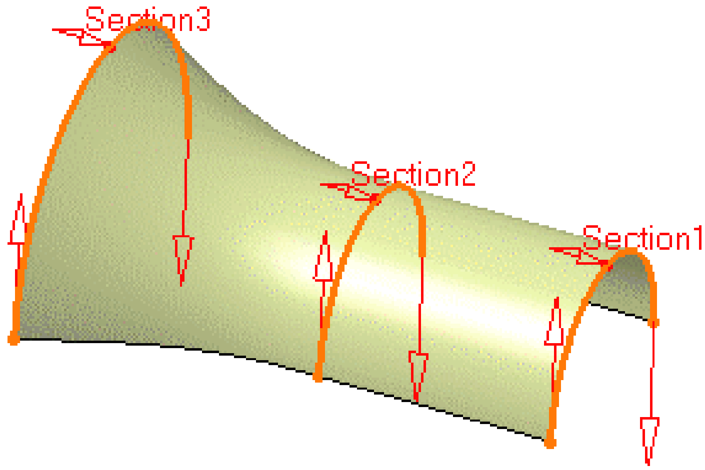
2. Select two or more planar section curves.

   The curves must be continuous in point.

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

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

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



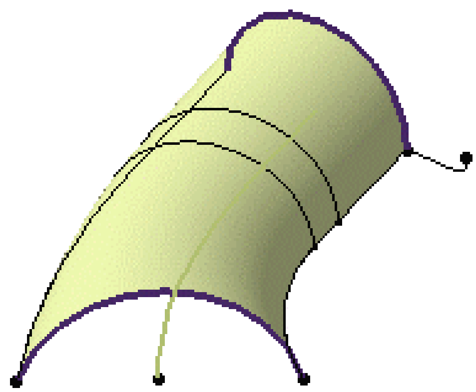
Example of a multi-sections surface defined by 2 planar sections and 2 guide curves:

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 multi-sections surface if it intersects the first extremity of each sections curve.

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



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

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

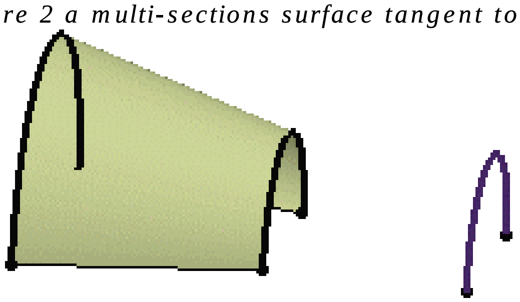


Figure 1

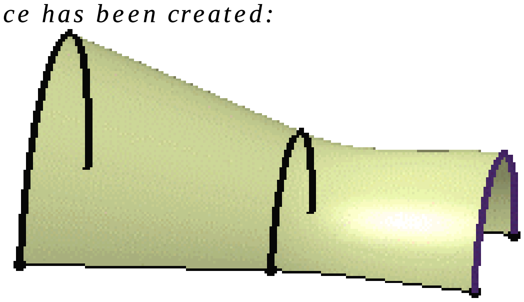


Figure 2

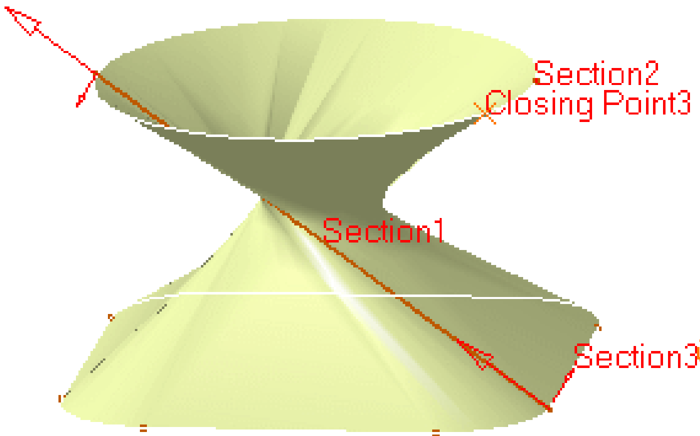
You can also impose tangency conditions by specifying a direction for the tangent vector (selecting a plane to take its normal, for example). This is useful for creating parts that are symmetrical with respect to a plane. Tangency conditions can be imposed on the two symmetrical halves.
Similarly, you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

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

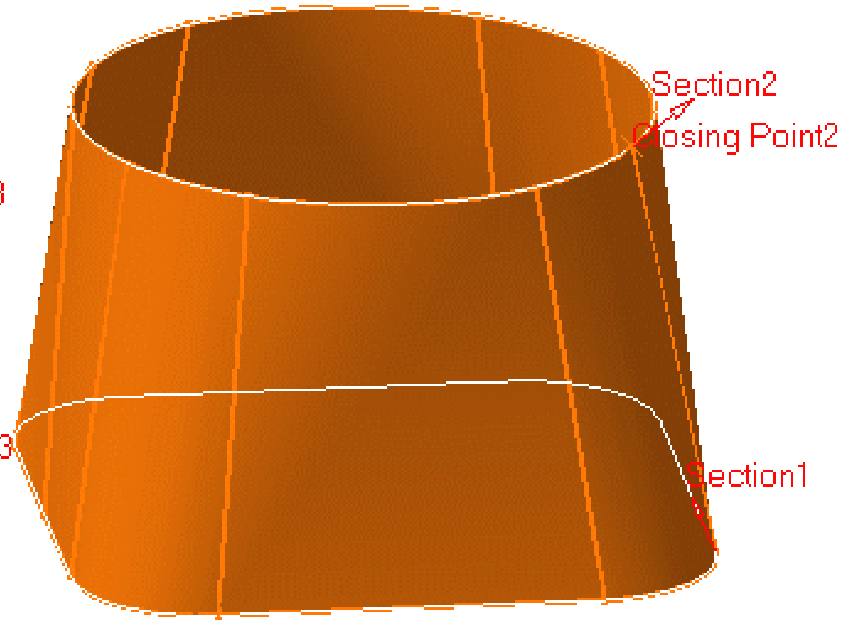
   Note that the spine curve must be normal to each section plane and must be continuous in tangency.

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

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

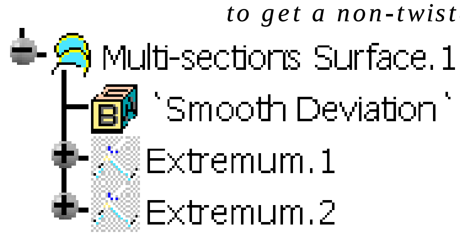


The surface is twisted



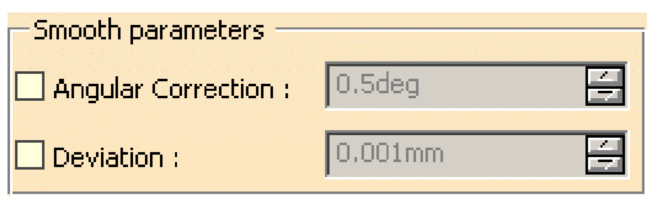
A new closing point has been imposed to get a non-twisted surface

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



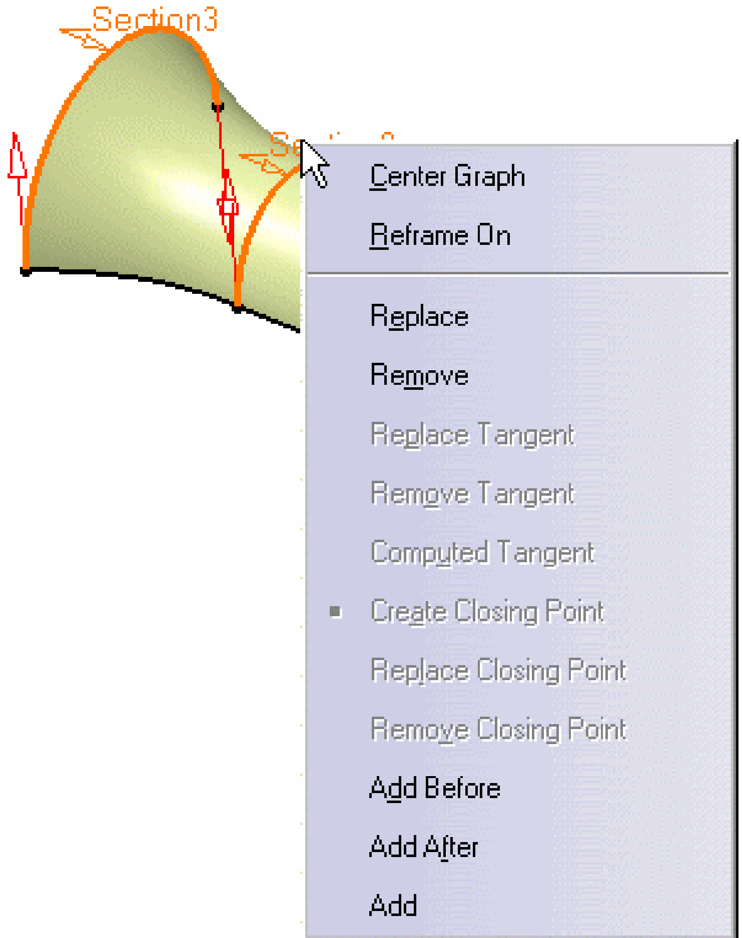
In the Smooth parameters section, you can check:

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

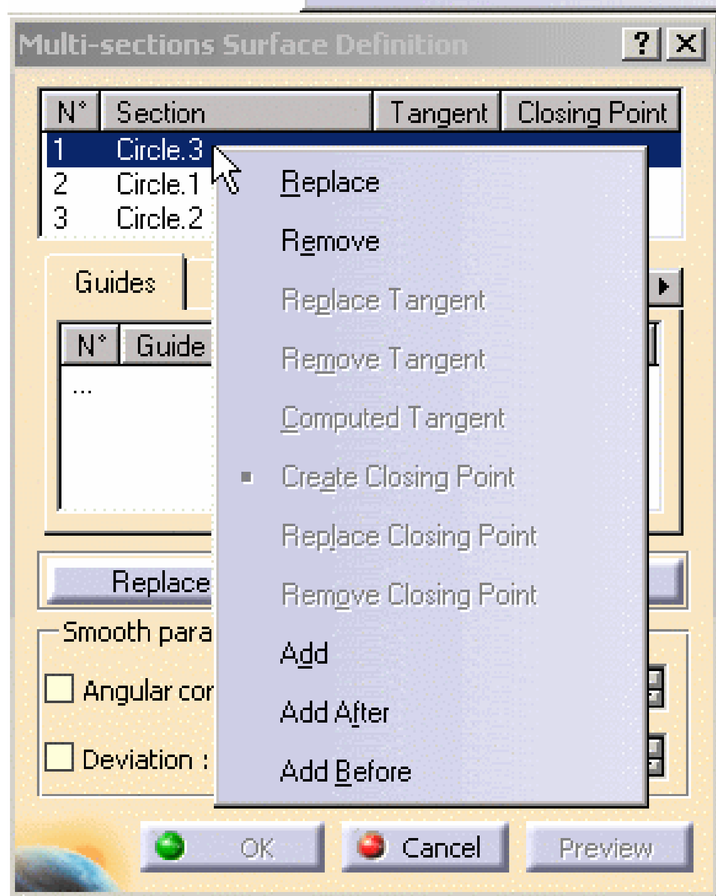


5. It is possible to edit the multi-sections surface reference elements by first selecting a curve in the dialog box list, or by selecting the text on the figure, then choosing a button to either:

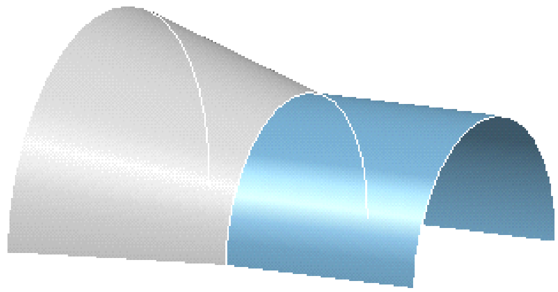
- remove the selected curve
- replace the selected curve by another curve
- add another curve



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



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



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

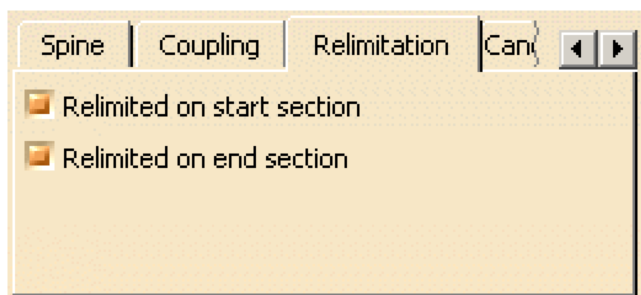
The surface (identified as Multi-sections Surface.xxx) is added to the specification tree.

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

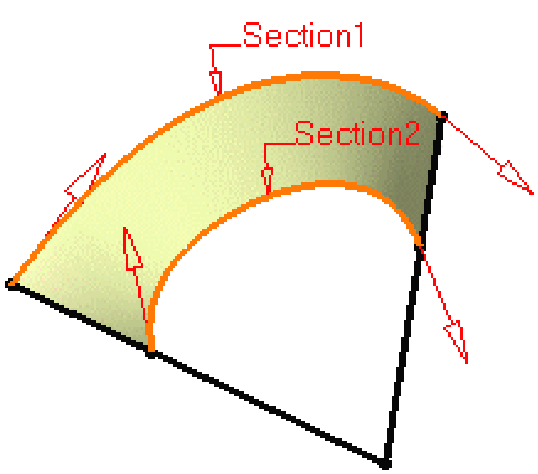
Relimitation

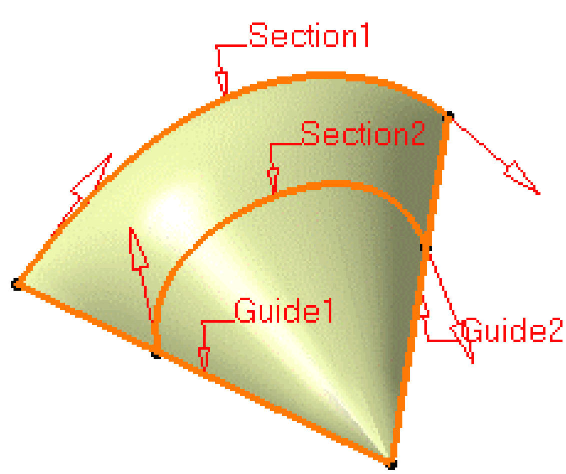
The Relimitation tab lets you specify the relimitation type. (Open the Loft3.CATPart document).

You can choose to limit the multi-sections surface only on the Start section, only on the End section, on both, or on none.



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





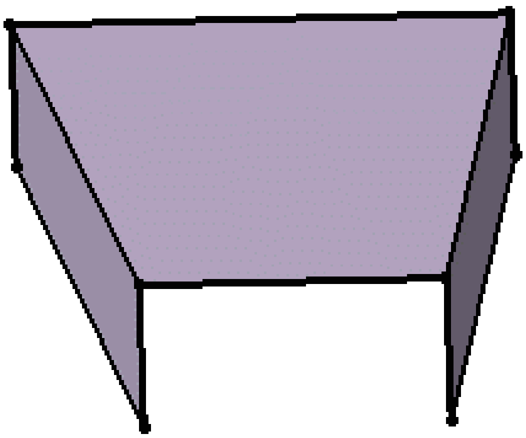
Multi-sections surface relimitation option checked on both Start and End section

Multi-sections surface relimitation option unchecked on End section only

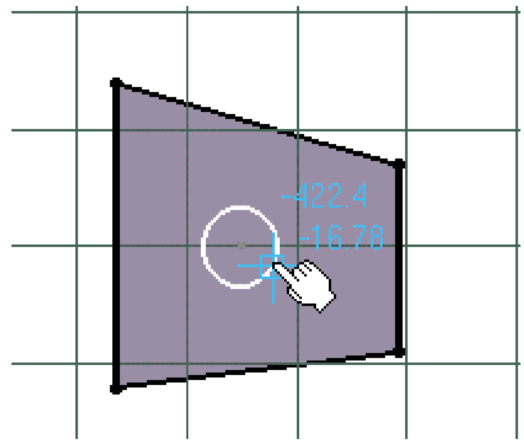
After the multi-sections surface is relimited, the following constraint needs to be fulfilled: the plane normal to the spine defined at the relimitation point must intersect the guide(s) and the point(s) resulting from this intersection must belong to the section.

Planar Surface Detection

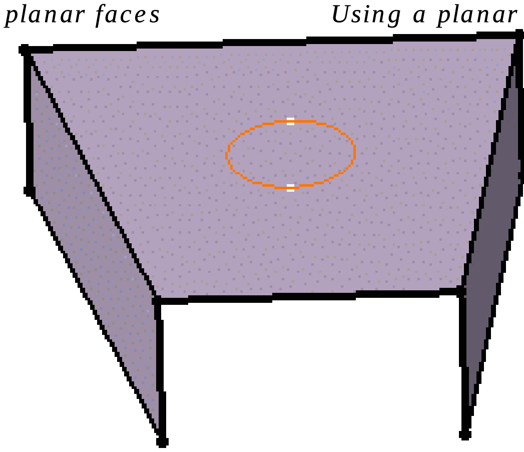
- Use the **Planar surface detection** check button (Canonical Surfaces tab) to automatically detect planar surfaces to be used as planes for features needing one in their definition.



Initial multi-sections surface with planar faces



Using a planar face as reference for a sketch



Resulting sketch

Coupling

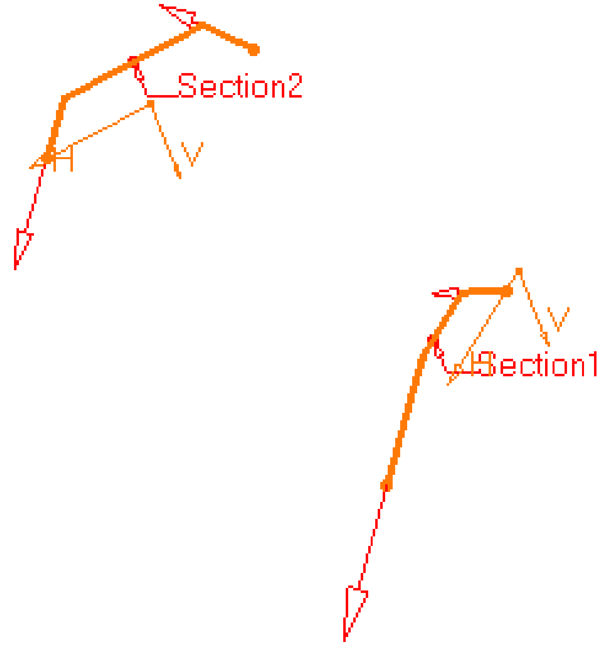
This task presents the two kinds of coupling during the creation of the multi-sections surface surface:

- coupling between two consecutive sections
- coupling between guides

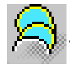
These couplings compute the distribution of isoparameters on the surface.

Open the Loft2.CATPart document.

Coupling between two consecutive sections



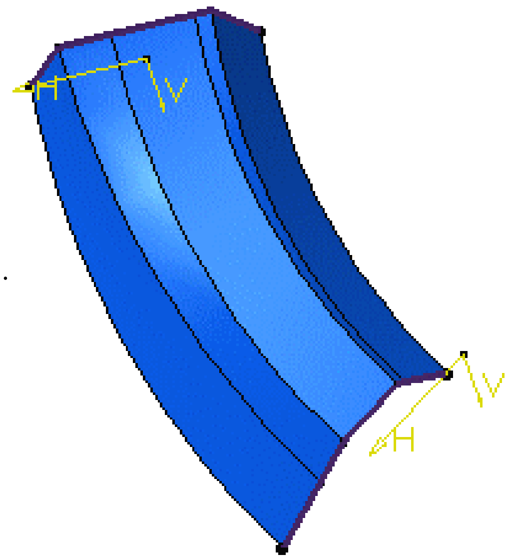
This coupling is based on the curvilinear abscissa.

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

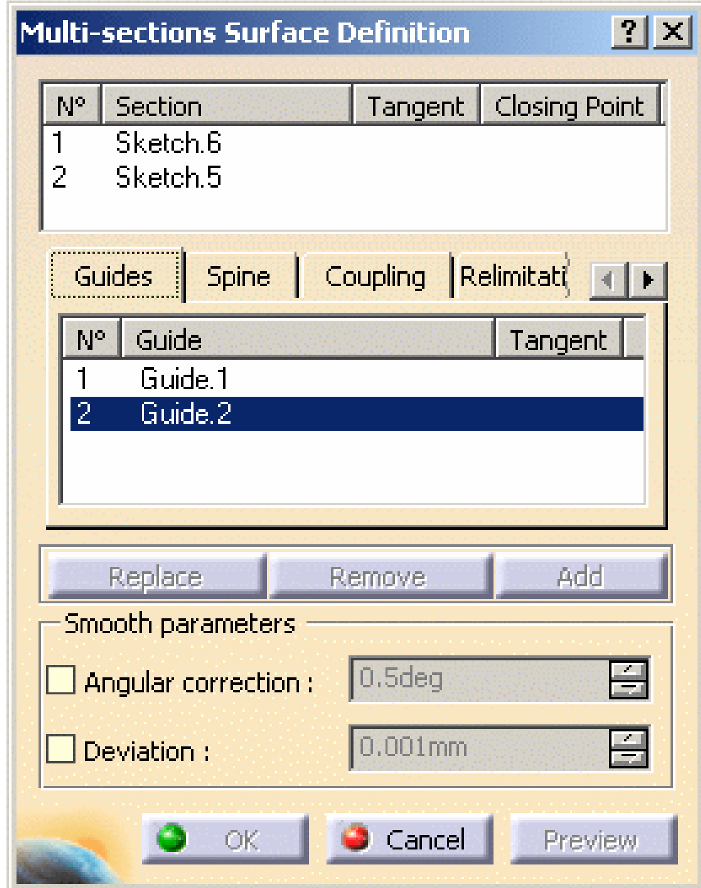
   The Multi-sections Surface Definition dialog box appears.

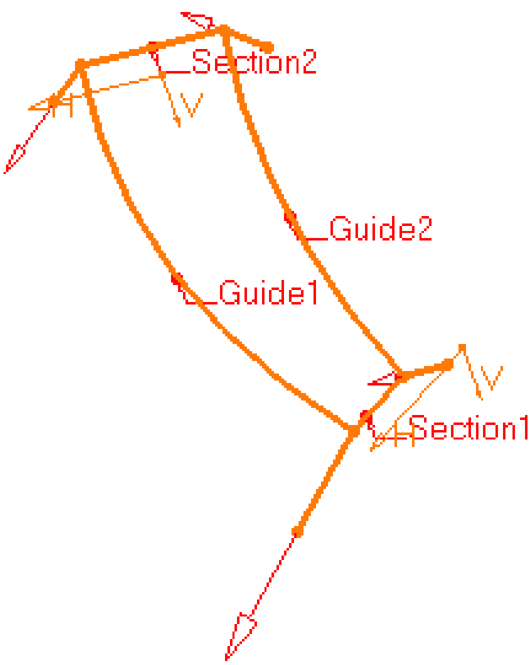
2. Select the two consecutive sections.

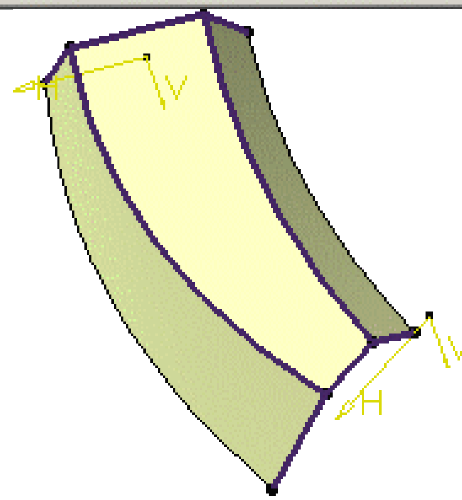
3. Click **OK** to create the multi-sections surface.



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

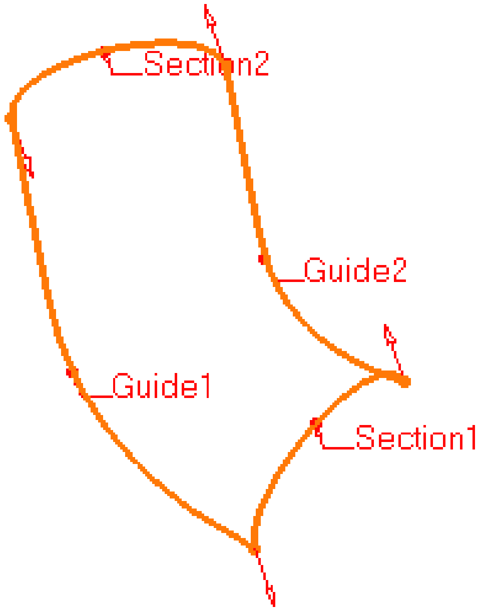


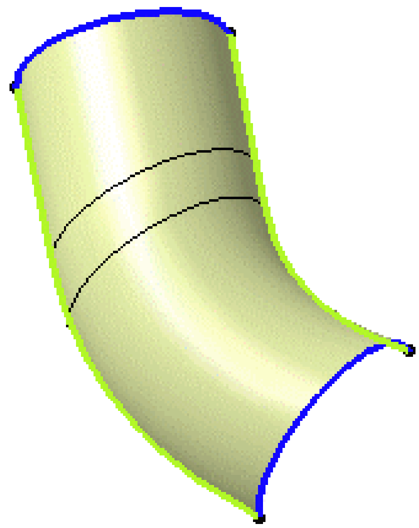




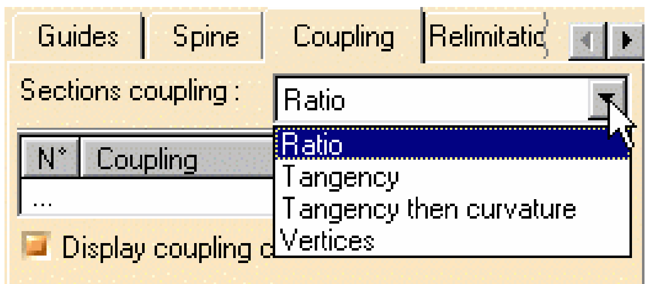
Coupling between guides

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

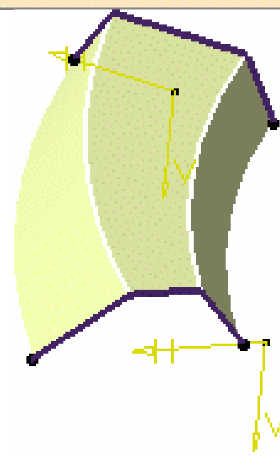




Several coupling types are available, depending on the section configuration:



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

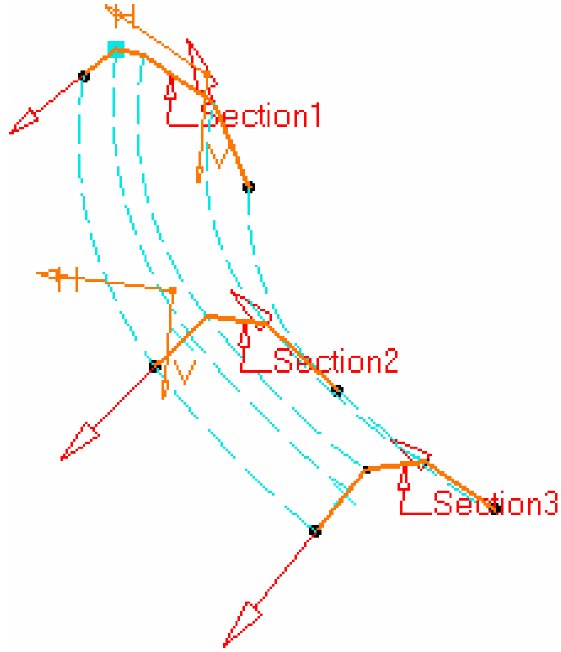


- **Tangency then curvature**: the curves are coupled according to their tangency continuity first then curvature discontinuity points. If they do not have the same number of points, they cannot be coupled using this option.
- **Vertices**: the curves are coupled according to their vertices. If they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

Manual Coupling

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

1. Select the sections for the multi-sections surface, and check their orientations.

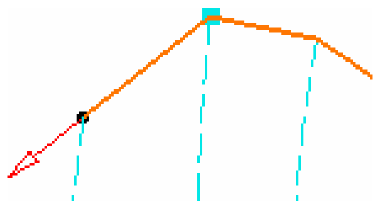


2. In the **Coupling** tab, choose the **Tangency** option and click **Apply**.

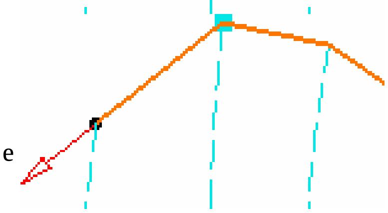
   An error message is displayed as the number of discontinuity points on the first section is greater than on the other two sections.

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

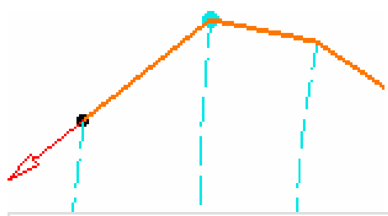
- In **Tangency** mode: uncoupled tangency discontinuity points are represented by a square



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

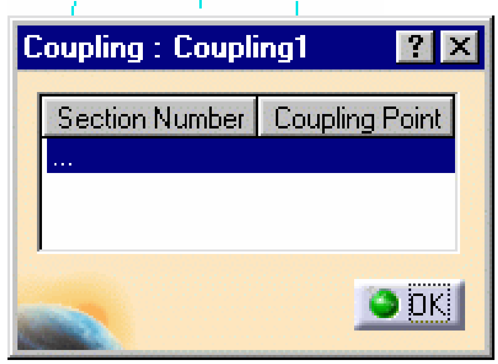


- In **Vertices** mode: uncoupled vertices are represented by a full circle

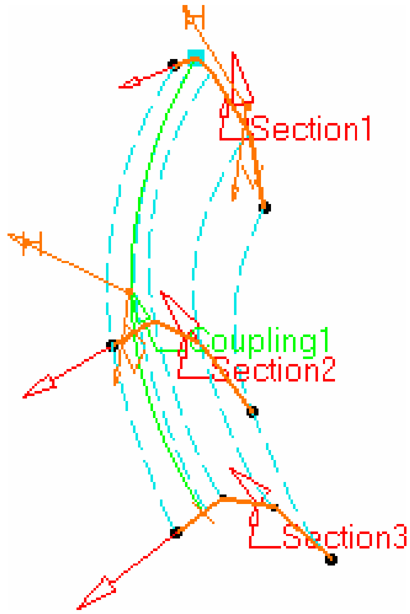


3. Click in the coupling list, or choose **Add coupling** in the contextual menu, or using the **Add** button, and manually select a point on the first section.

   The Coupling dialog box is displayed.



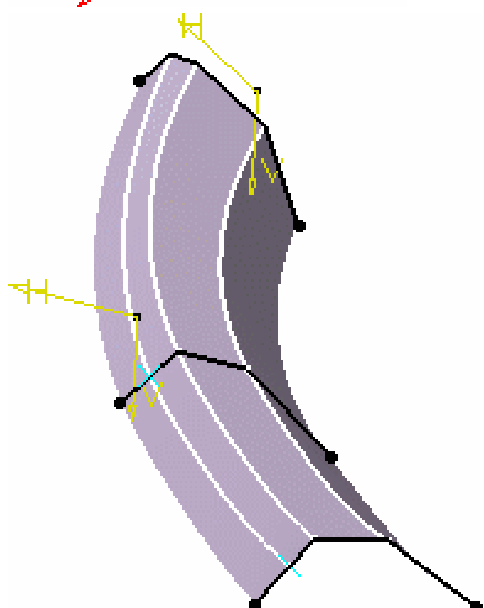
4. Select a corresponding coupling point on each section of the multi-sections surface.



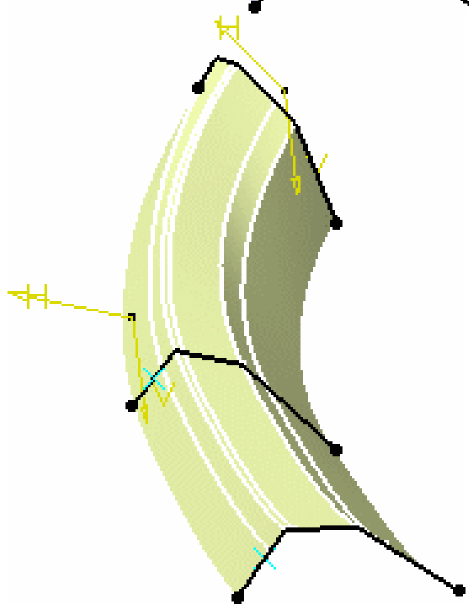
The Coupling dialog box is updated consequently, and the coupling curve is previewed, provided the **Display coupling curves** option is active.

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

5. Click OK.



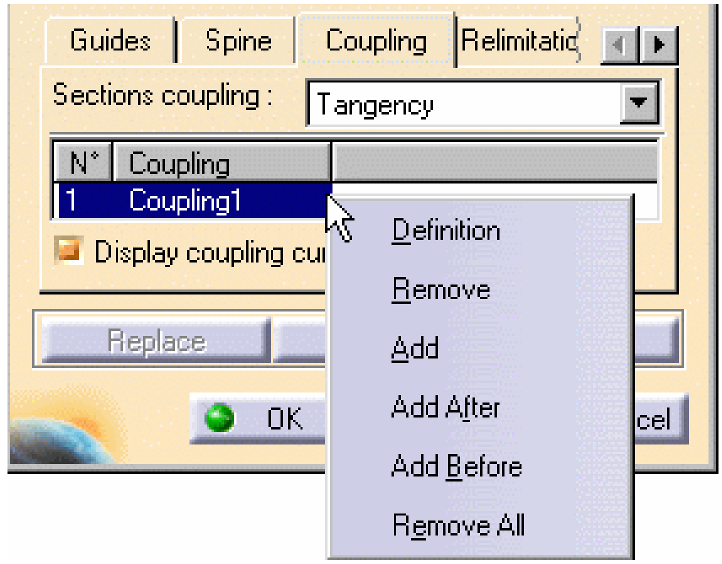
The multi-sections surface is created as defined with the coupling specifications.



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

Note the increased number of generated surfaces.

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

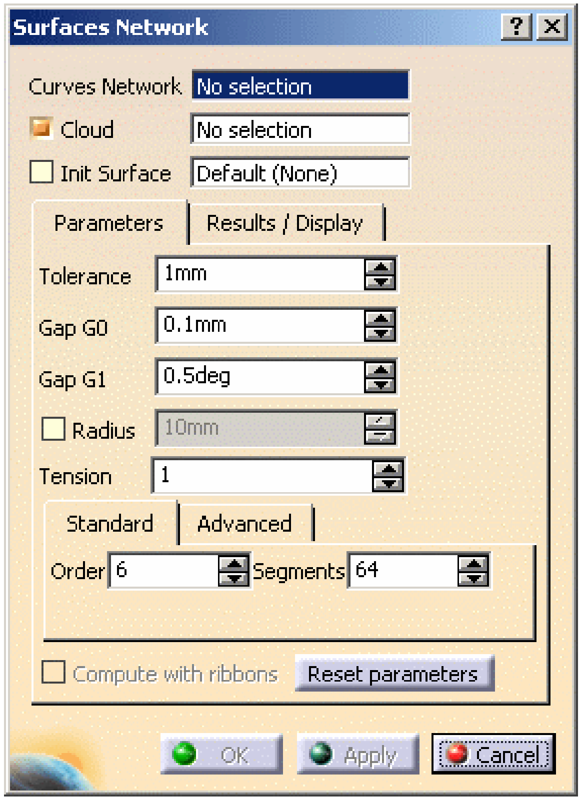


Surfaces Network

This task will show you how to create a surface on a curves network.

Create a network of curves from the SurfNetwork.CATPart from the samples directory.

1. Click the **Surfaces Network** icon . The dialog box is displayed.

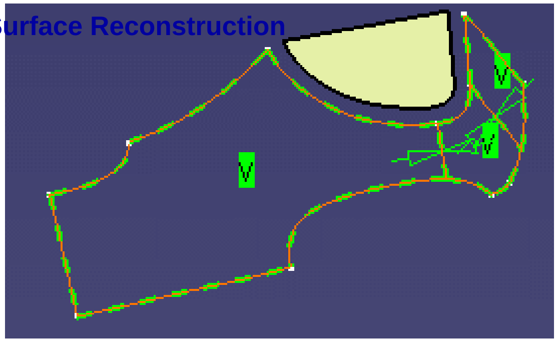


2. Select the curves network you have created (mandatory), as well as the mesh as the Cloud.

It is not necessary to use a support **Cloud** or a **Init Surface**, but they may improve the output.

Check the **Init Surface** box if you want to enter one. The init surface helps the computation by giving the shape of the result surface. You can either enter it yourself or let the application compute it in the direction of the largest curve. If you select an init surface, its name is displayed in the field **Init Surface**.

The init surface must be larger than the domain to process.



The curves network is displayed, with a green V marker on each wire, meaning the wire will be filled.

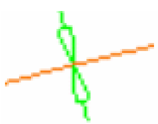
You can fill all the wires automatically (this is the proposed default) or you can fill the wires one by one.

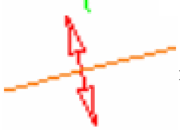
If you do not want to fill a wire:

- either click on its marker, it will turn into a red X marker. The wire will not be filled. (click the marker again to reselect the wire, if necessary),
- or place the cursor on a marker and use the contextual menu:

Selected and **Not selected** apply to a single wire, **Select all** and **De-Select all** apply to the whole network, **Swap Selection** reverses the selection. **Remove Surface** allow you to suppress a surface you are not happy with or that you prefer to fill later.

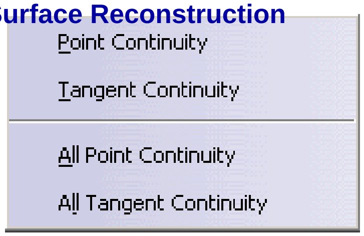
Constraints are set on edges shared by two wires.

 represents a point continuity

 represents a tangency continuity

To change the type of a constraint:

- either click on its marker. This will act as a toggle,
- or place the cursor on a marker and use the contextual menu:



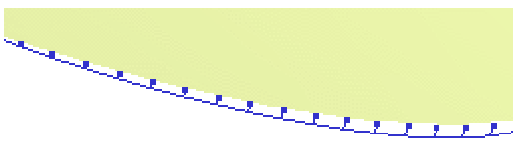
Point Continuity and **Tangent Continuity** apply to a single constraint, **All Point Continuity** and **All Tangent Continuity** apply to the whole network.

You can set the following **Parameters:**

Tolerance is the mean maximum deviation between the surface created and the cloud of points or polygon, i.e. the deviation may be higher at some places. This field is editable.

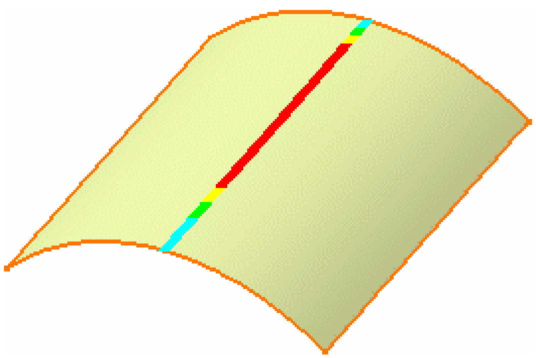
Gap: G0:

this is the distance between the surface and the boundary curves. Since there is more noise on points than on curves, the **Tolerance** may be higher than the **GO Gap**. The default value is 1.



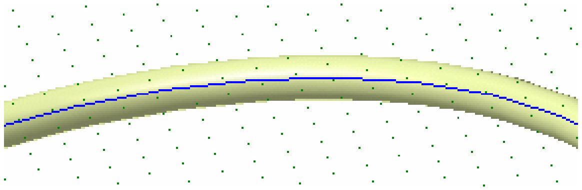
Gap: G1:

this is the tangency tolerance between two contiguous surfaces (in blue below). The default value is 0.5.

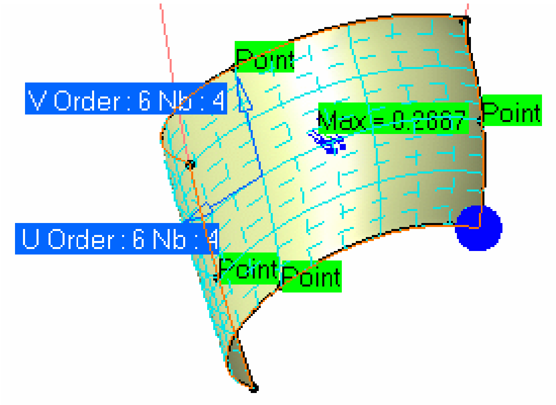


Radius:

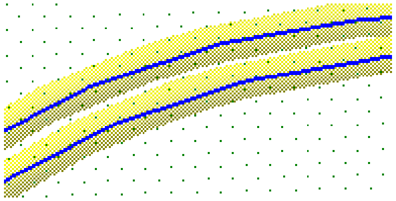
when the cloud of points is noisy, it is difficult to have the surface going through all the points and the curves (risk of undulations). The points inside a circular pipe centered on the curve are deleted, and you may want to set the radius of that pipe.



When you check this option, a blue sphere is displayed on the extremity of the first curve, representing this radius (if you have selected at least one curve and a cloud of points or a polygon).

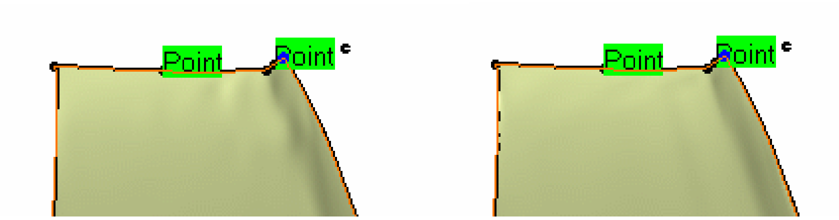


If two curves are not distant enough, all the points between them may be deleted, making the computation of the surface impossible.



Tension:

Possible values are between 0 and 4. Use a higher value to have a smoother (but less tense) surface.



Standard: Order, Segments:

These parameters apply globally to the surface computed. They are maximum values. The actual values are computed automatically by the action.

Surfaces Network creates a NURBS surface, controlled by the tolerance (i.e. **Tolerance**), the number of segments and their order. Whenever possible, this surface consists of one single segment, otherwise, it is made of several segments. This surface may then be trimmed by the curves.

You can increase the order of the segments, thus reducing their number, or vice-versa.

If the number of segments is x, this means that the surface computed will consist of a maximum of x segments, or less. The default number of segments is 64, the maximum number is 2048.

If the order of segments is y, this means that each segment will have a maximum number of y control points in each direction, or less. The segment order may vary from 3 to 15.

Advanced: Order and Segments in U and V

You may want to impose an order and a number of segments in both U and V direction. To do so, go to the **Advanced** tab. You can edit the fields to :

- enter the number of segments in each direction,
- enter the order of segments in each direction.

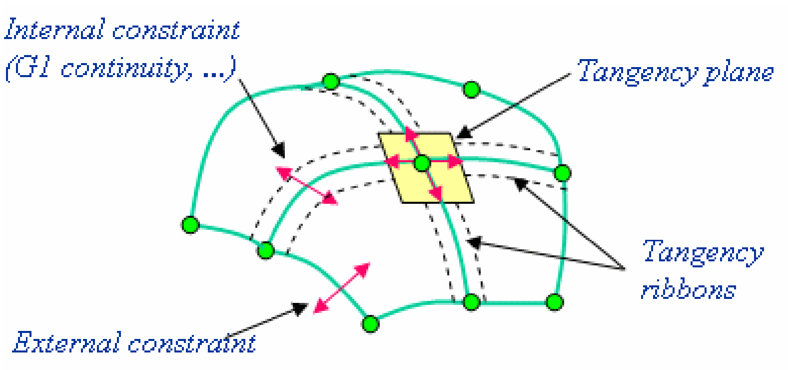
Click **Apply** to restart the computation.

- Increasing the order of the segments may result in an oscillating surface, even if this is not visible.

Compute with ribbons:

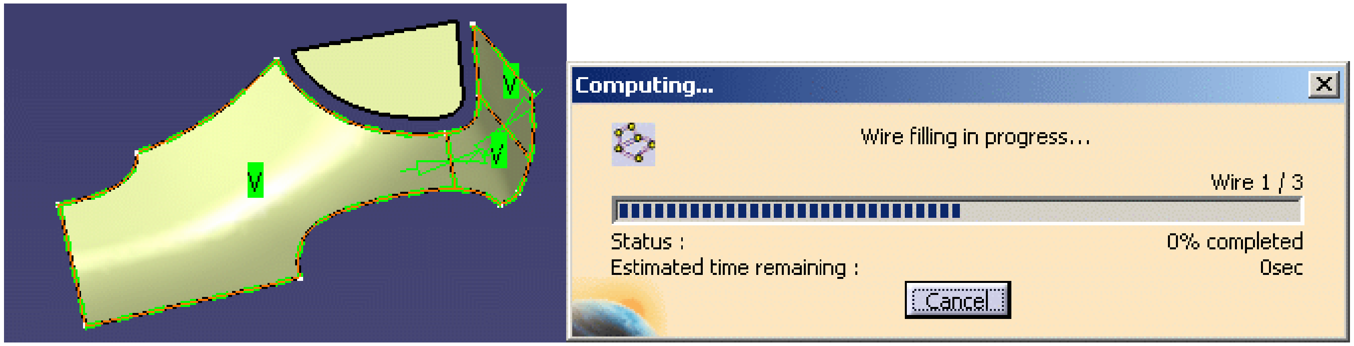
This option is only available if a mesh is selected and is used when a tangent continuity is required between the wires.

The wires curves are projected on the mesh and a tangency ribbon is computed on the mesh around the curve projection, and then taken into account for the computation of the filling surface.

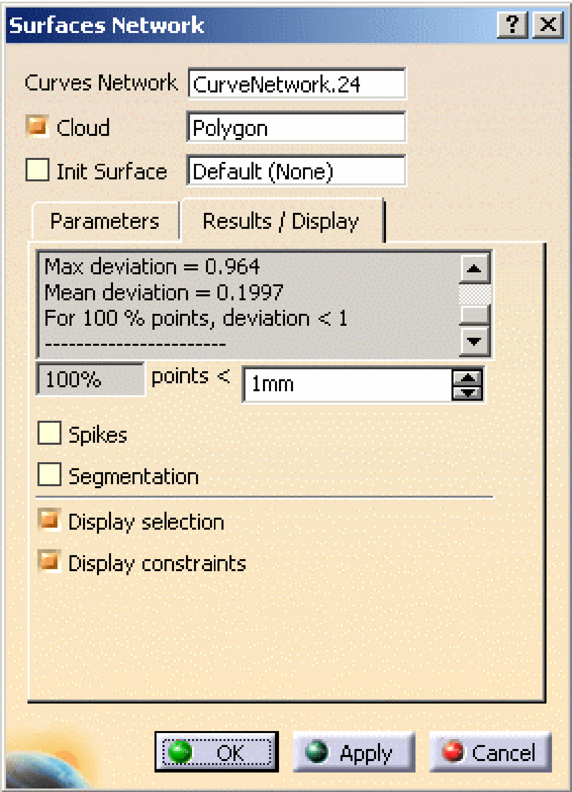


Use the **Reset parameters** button to reset the parameters

3. Push the **Apply** button. The filling surface is computed. A progress bar is displayed.



4. Go to the **Results / Display** tab.



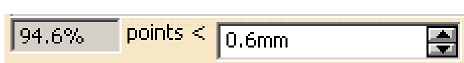
Information on the points for the parameters taken into account by the computation are available in the box at the top of the tab (no dynamic display):

- the maximum deviation found between the points of the cloud and the surface,
- the mean deviation found between the points of the cloud and the surface. This deviation should be as small as possible.
- the percentage of points of the cloud that are below the mean deviation.

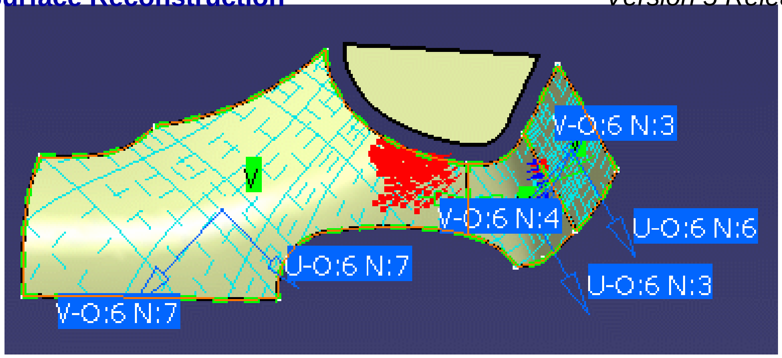
- Check the **Spikes** option to display the deviations.

- Check the **Segmentation** option to display

  - the segmentation on the surfaces computed,
  - the number of segments and order in U,
  - the number of segments and order in V.

- Use the **Deviation** field 

to enter the value above which the deviation spikes will be displayed. When you first enter the action, the **Deviation** value is the same as the **Tolerance.** Once a surface has been computed, the **Deviation** value is the computed one.



By default, the options **Display selection** and **Display constraints** are checked. You can uncheck them according to your needs.

5. Once you are satisfied, click OK to create the result: the surfaces are assemble (Tolerance = 0.1 mm)

   If the assembly does not respect the tolerance an error message is issued.

   If the assembly failed, an error message is issued and the surfaces are created separately.

   A Surface.xx element is created in the specification tree.

Transformations

This chapter deals with transformations in Quick Surface Reconstruction.

Performing a Symmetry on Geometry

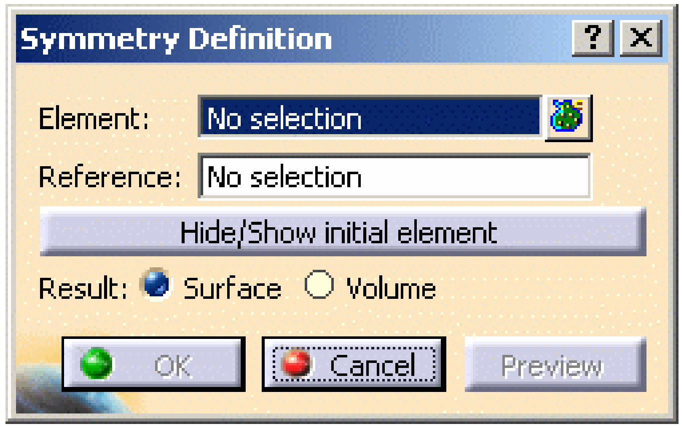
This functionality is P2 for FreeStyle Shaper, Optimizer, and Profiler.

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

Open the Transform1.CATPart document.

1. Click the **Symmetry** icon .

   The Symmetry Definition dialog box appears as well as the Tools Palette.

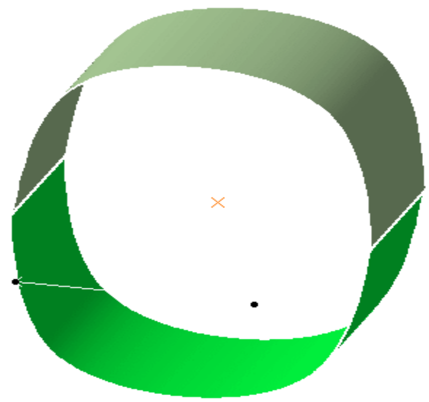


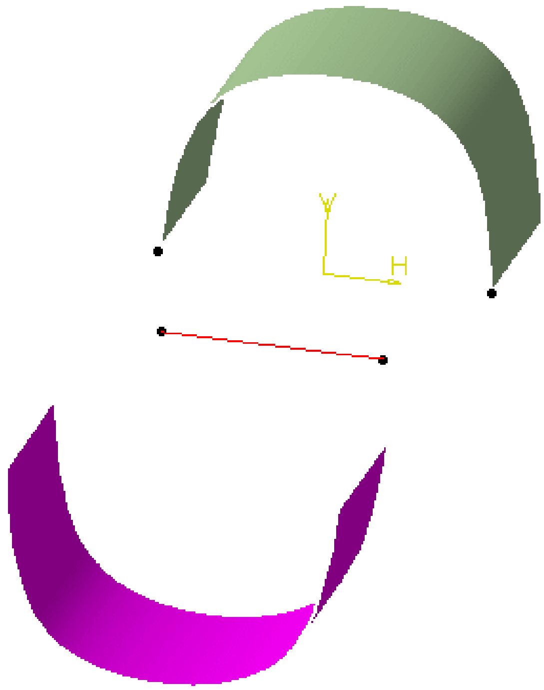
2. Select the **Element** to be transformed by symmetry.

3. Select a point, line or plane as **Reference** element.

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

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





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

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

- Use the **Hide/Show initial element** button to hide or show the original element for the translation.

- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
To have further information about volumes, please refer to the corresponding chapter.

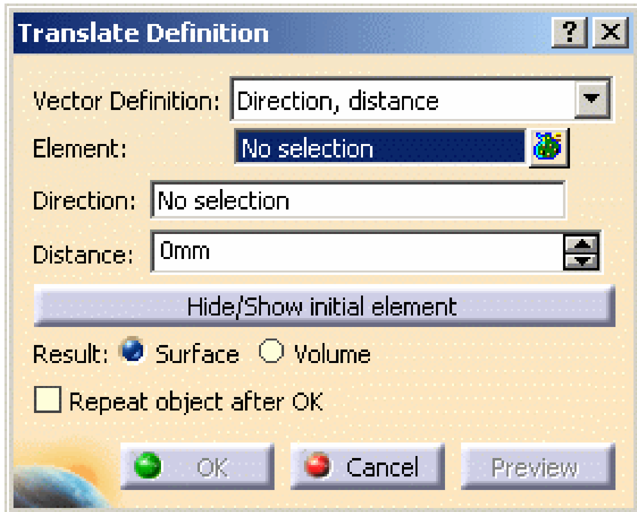
The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.

Translating Geometry

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

Open the Translate1.CATPart document.

1. Click the **Translate** icon.



The Translate Definition dialog box appears as well as the Tools Palette.

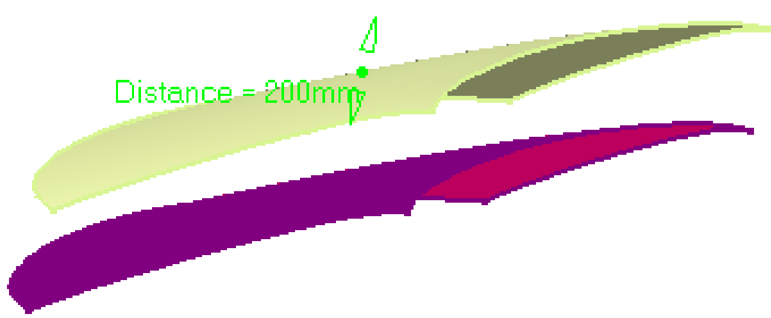
2. Select the **Element** to be translated.

3. Select the **Vector Definition**.

Direction, distance

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

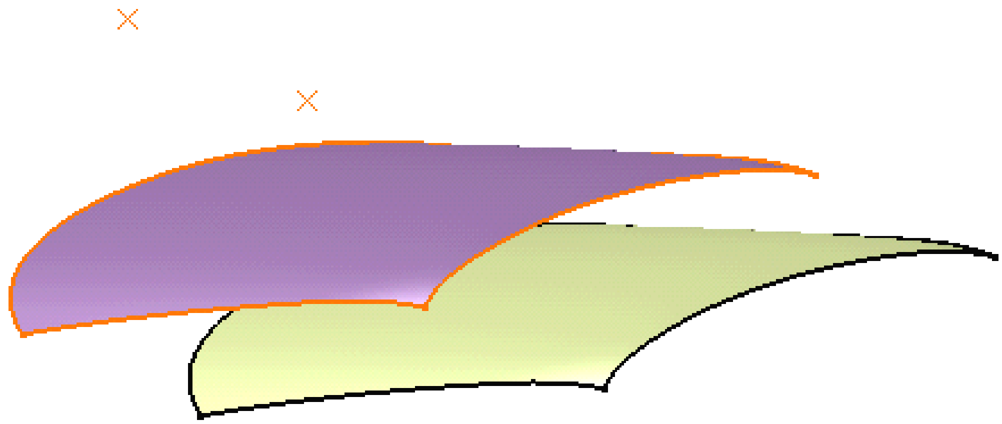
   You can also specify the direction by means of X, Y, Z vector components by using the contextual menu on the **Direction** field.



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

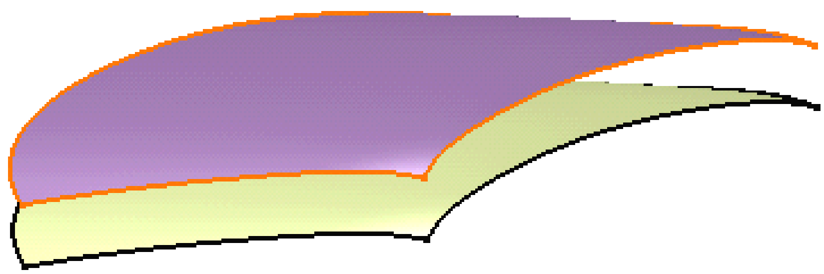
Point to Point

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



Coordinates

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



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

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

- Use the **Hide/Show initial element** button to hide or show the original element for the translation.

- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  To have further information about volumes, please refer to the corresponding chapter.

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

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

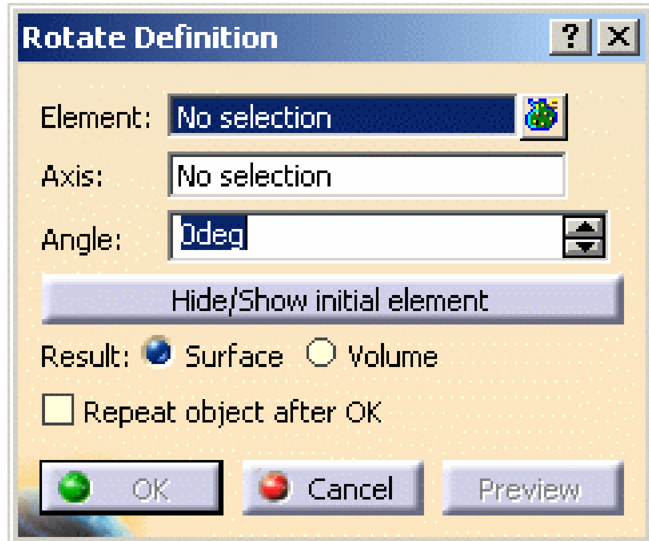
Parameters can be edited in the 3D geometry. To have further information, please refer to the Editing Parameters chapter.
The following capabilities are available: Stacking Commands, Selecting Using Multi-Output, Measure Between and Measure Item.

Rotating Geometry

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

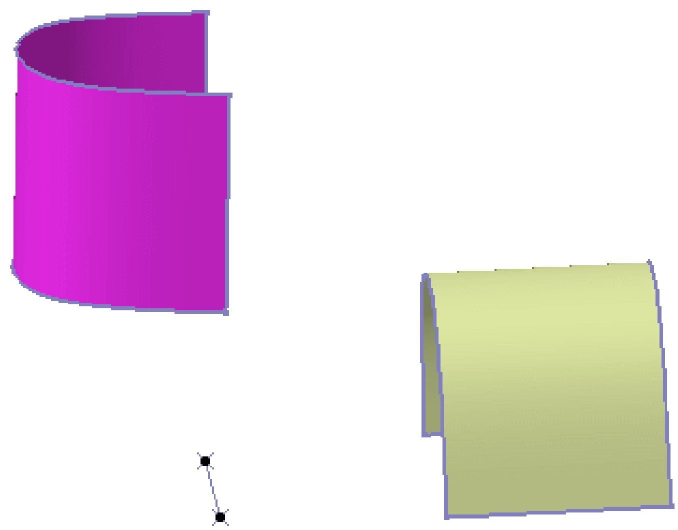
Open the Transform1.CATPart document.

1. Click the **Rotate** icon .



The Rotate Definition dialog box appears as well as the Tools Palette.

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

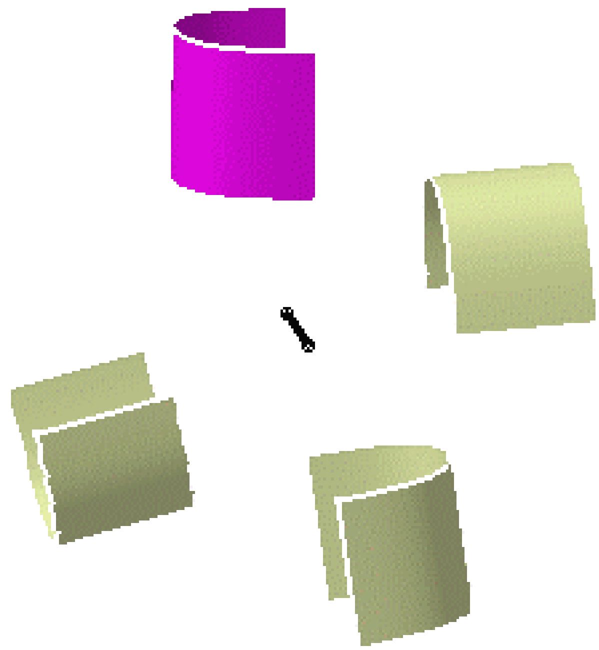


5. Click **OK** to create the rotated element.

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

- Use the **Hide/Show initial element** button to hide or show the original element for the translation.
- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  To have further information about volumes, please refer to the corresponding chapter.

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



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

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

- You can edit the rotated element's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

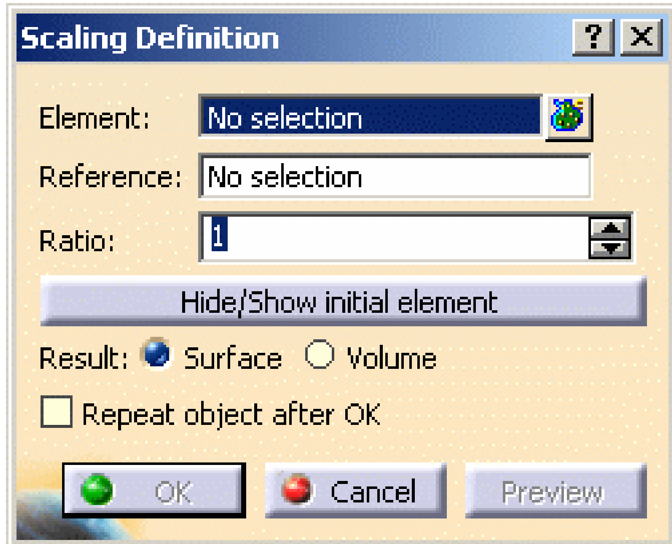
The following capabilities are available: Stacking Commands, Selecting Using Multi-Output, Measure Between and Measure Item.

Transforming Geometry by Scaling

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

Open the Transform1.CATPart document.

1. Click the **Scaling** icon .



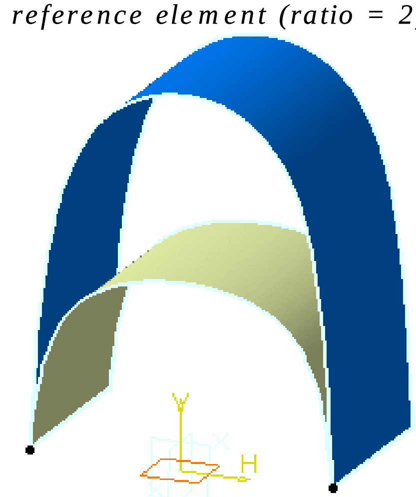
The Scaling Definition dialog box appears as well as the Tools Palette.

2. Select the **Element** to be transformed by scaling.

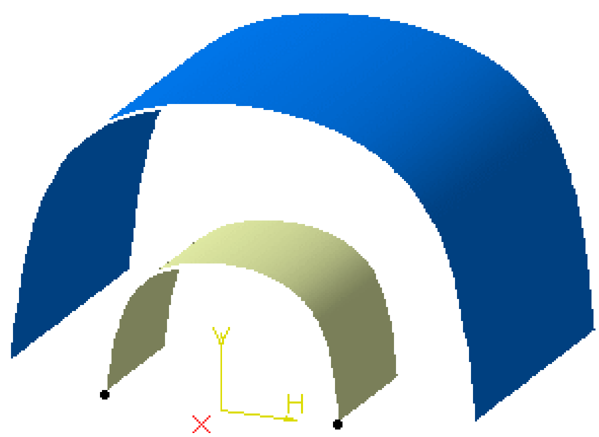
3. Select the scaling **Reference** point, plane or planar surface.

4. Specify the scaling **Ratio** by entering a value or using the drag manipulator.

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



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



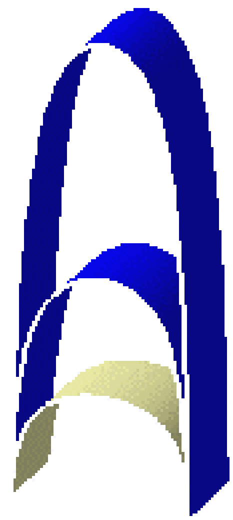
5. Click OK to create the scaled element.

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

- Use the **Hide / Show initial element** button to hide or show the original element for the translation.

- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  To have further information about volumes, please refer to the corresponding chapter.

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



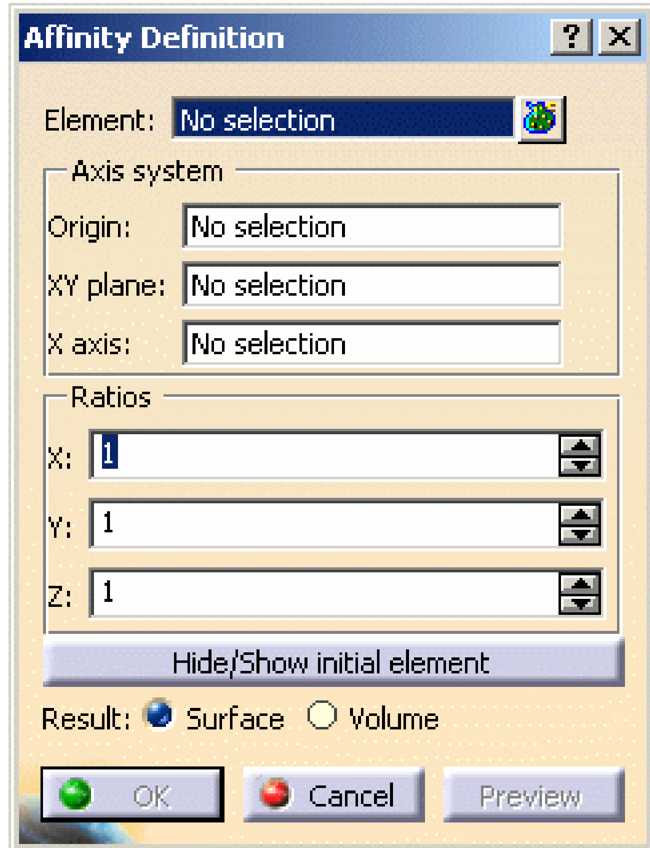
The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.

Transforming Geometry by Affinity

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

Open the Transform1.CATPart document.

1. Click the **Affinity** icon .



The Affinity Definition dialog box appears as well as the Tools Palette.

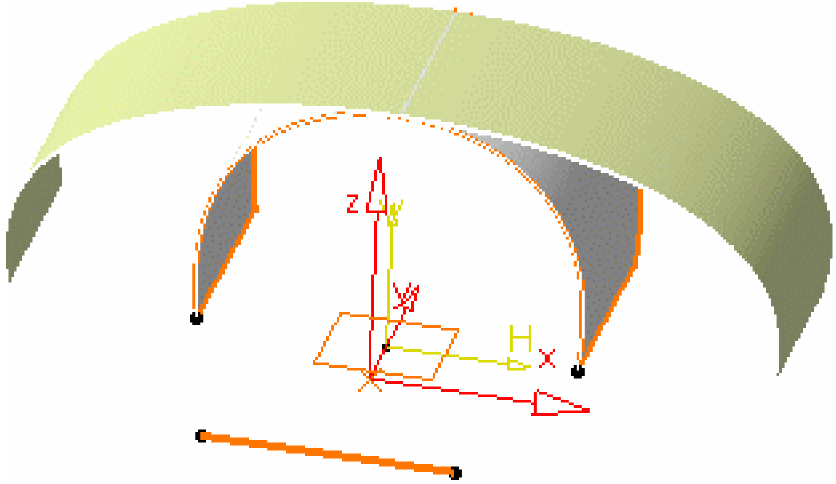
2. Select the **Element** to be transformed by affinity.

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

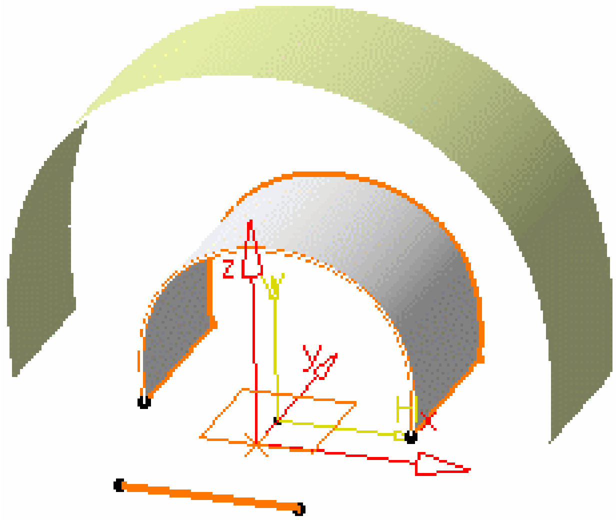
- the **Origin** (Point.1 in the figures below)
- the **XY plane** (the XY plane in the figures below)
- the **X axis** (Line.1 in the figures below).

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

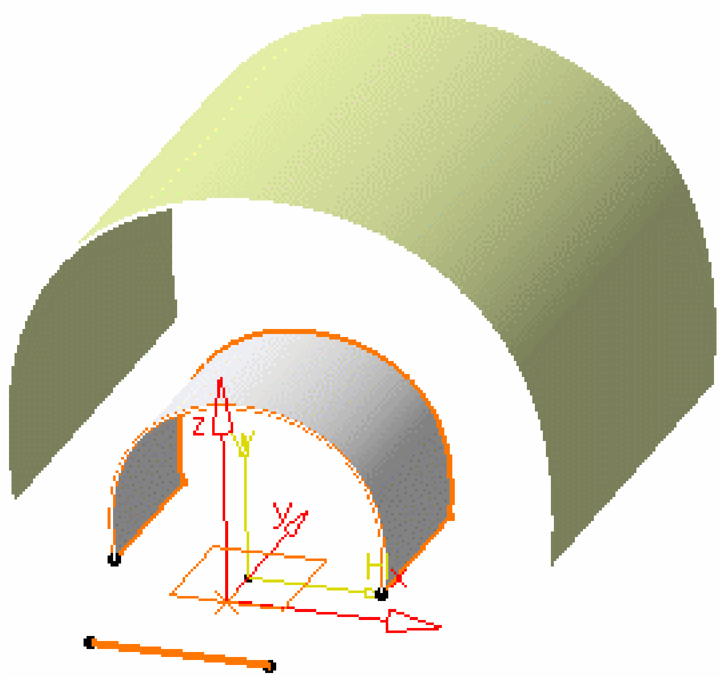
The figure below illustrates the resulting affinity with ratios X = 2, Y =1 and Z=1.



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



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



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

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

- Use the **Hide/Show initial element** button to hide or show the original element for the translation.

- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
To have further information about volumes, please refer to the corresponding chapter.

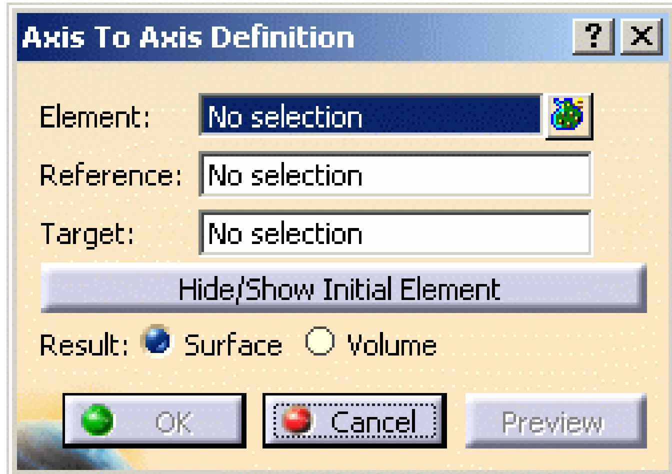
The following capabilities are available: Stacking Commands and Selecting Using Multi-Output.

Transforming Elements From an Axis to Another

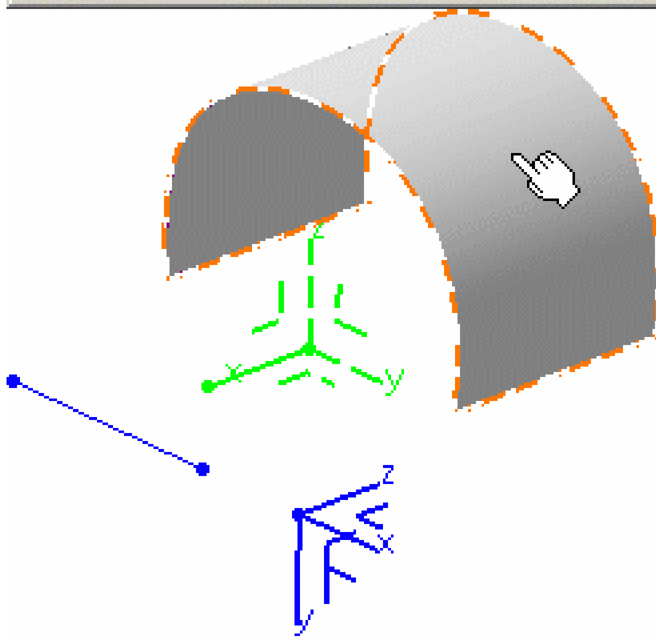
This task shows you how to transform geometry positioned according to a given axis system into a new axis system. The geometry is duplicated and positioned according to the new axis system. One or more elements can be transformed at a time, using the standard multi-selection capabilities.
See also Defining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

Open the Transform2.CATPart document.

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

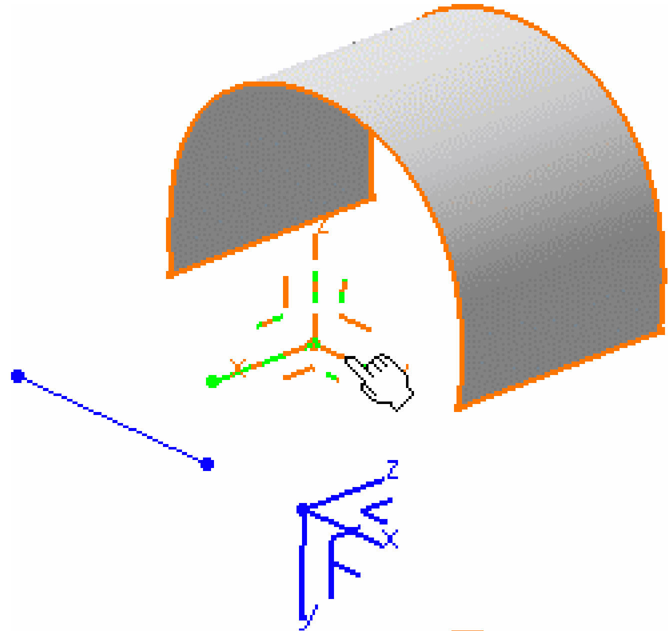


The Axis to Axis Definition dialog box appears as well as the Tools Palette.

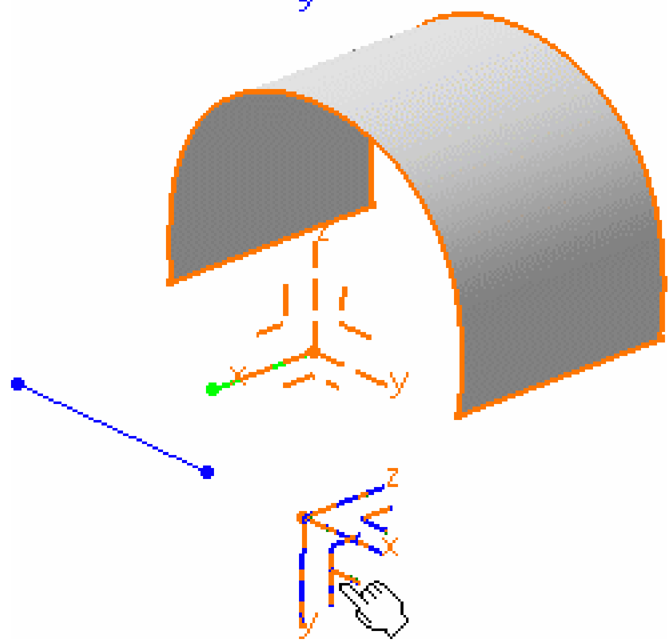


2. Select the **Element** to be transformed into a new axis system.

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



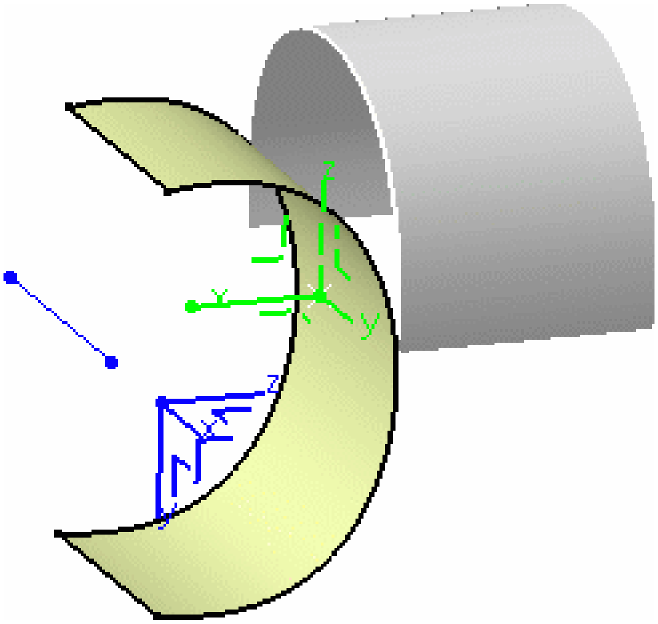
4. Select the **Target** axis system, that is the one into the element should be positioned.



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

   New geometry is now positioned into the new axis system.

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



- Use the **Hide/Show initial element** button to hide or show the original element for the translation.

- Choose whether you want the result of the transformation to be a surface or a volume by switching to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  To have further information about volumes, please refer to the corresponding chapter.

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

Segmentation

This chapter deals with segmentation in Quick Surface Reconstruction.

Segmentation by Curvature Criterion

This task will show you how to define areas on a polygon along the curvatures or curvature radii.

There are five curvatures:

- Maximum,
- Minimum,
- Mean,
- Gaussian,
- Absolute.

The geometric construction of the **maximum** and **minimum** curvatures is the following: let be a plane containing the normal to the surface in a given point. This plane cuts the surface along a curve that has a given curvature in this point. If this plane rotates around the normal, the curvatures of the curves intersecting the surface will vary between two utmost values. These two values are the maximum (KM) and the minimum (Km) curvatures.

The **mean** curvature is equal to (KM+Km)/2. The utmost values appear where the surface is the most warped. The mean curvature is largely used to detect irregularities on the surface. A minimal surface is characterized by a null mean curvature.

The **gaussian** curvature is equal to KM.Km. It describes the local shape of a surface in one point:

- if it is positive, the point is elliptic, i.e. the surface has locally the shape of an ellipsoid around that point,
- if it is negative, the surface is hyperbolic in this points, i.e. the local shape is a horse saddle,
- it it is null, the surface is parabolic in this point, i.e. one of the maximum or minimum curvatures is null in this point. The cone and the cylinder are two surfaces where all points are parabolic.

The **absolute** curvature is equal to |KM|+|Km|. It is used to detect the surface areas where the surface is locally almost flat (the absolute curvature is almost null).

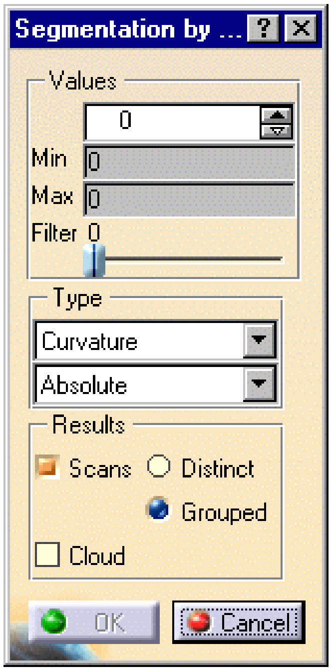
The **curvature radii** are the inverse of the corresponding curvatures. Only the **maximum** and the **minimum** radii are relevant.

It is carried out on polygons only.

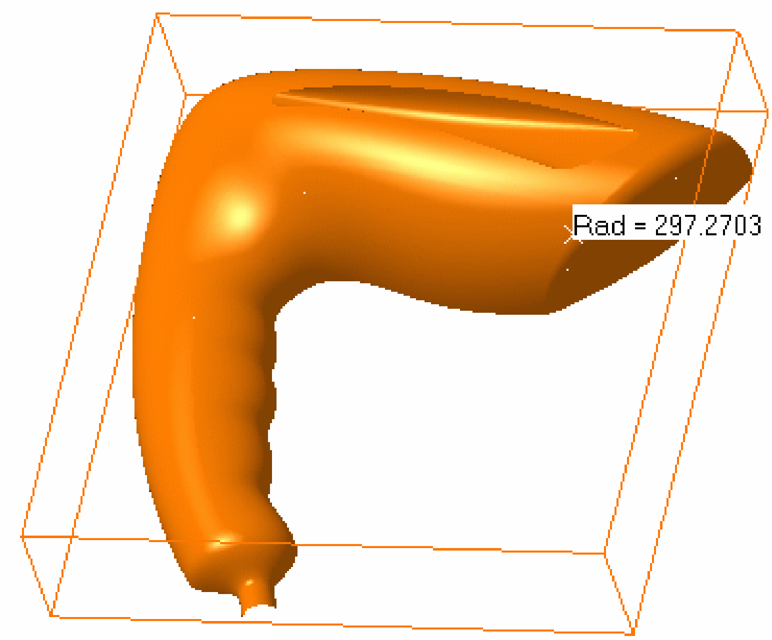
Open the Curvature01.CATPart from the samples directory.

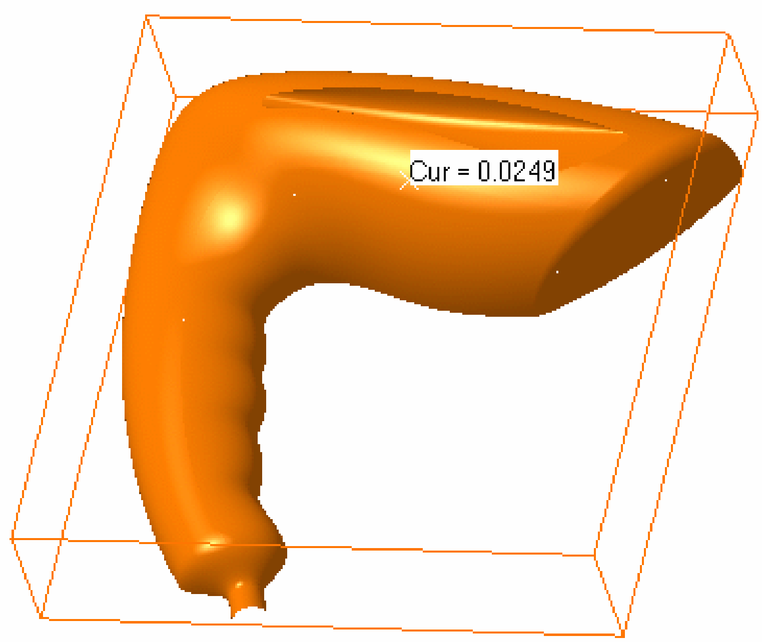
1. Click the **Segmentation by Curvature Criterion** icon. The **Curvature Analysis** dialog box is displayed.

2. Select the polygon to analyze.



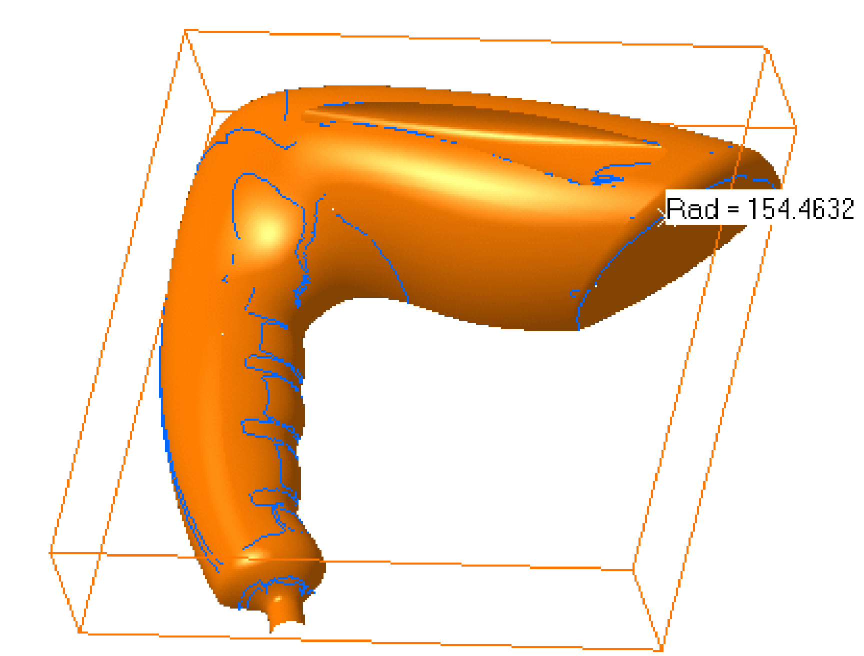
3. Select the type of analysis from the combo box: **Curvature** or **Radius**. Sweep the polygon with the cursor: The value of the curvature or the radius is displayed dynamically as you move the cursor:



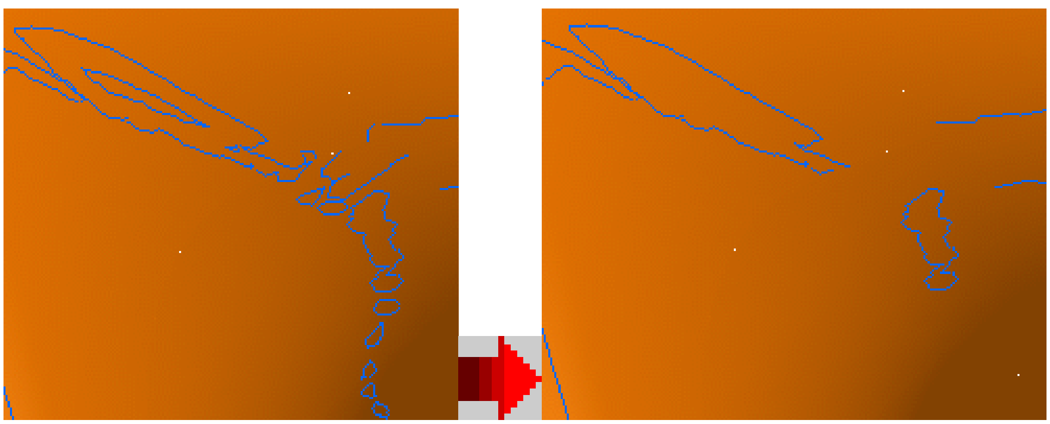


This is especially helpful to retrieve the value of fillets.

4. Click the polygon: scans are displayed: combine the types of analysis and the value in the top spinner box to display scans according to your needs:



5. Use the **Filter** cursor to remove unwanted points.

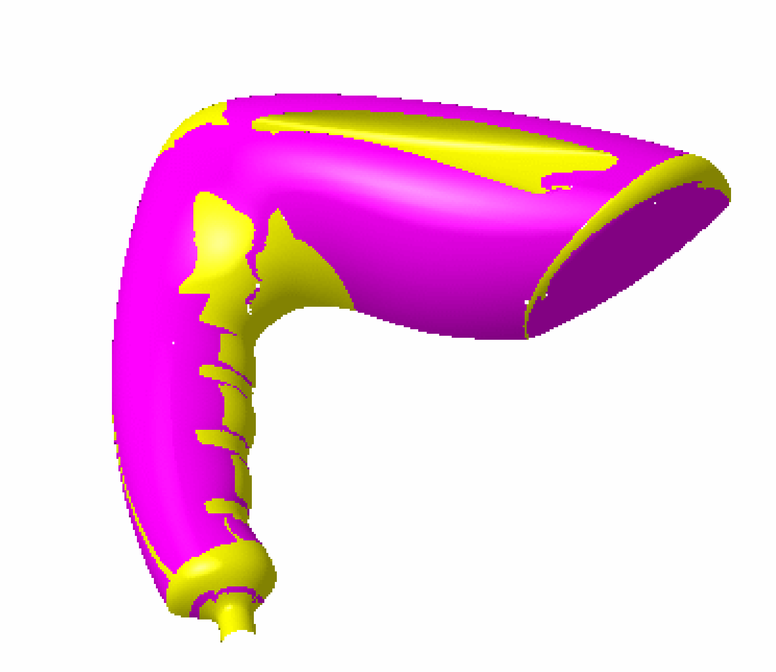


6. Click OK to create the result. You can choose to create:

- Scans, either distinct or grouped:

Scans.x elements are created in the specification tree.

- or Clouds, that is sub-polygons:



Sub Polygon.x elements are created in the specification tree.

These polygons can then be processed with the Basic Surface Recognition action, for example.

The input polygon is sent to the NoShow.

Segmentation by Slope Criterion

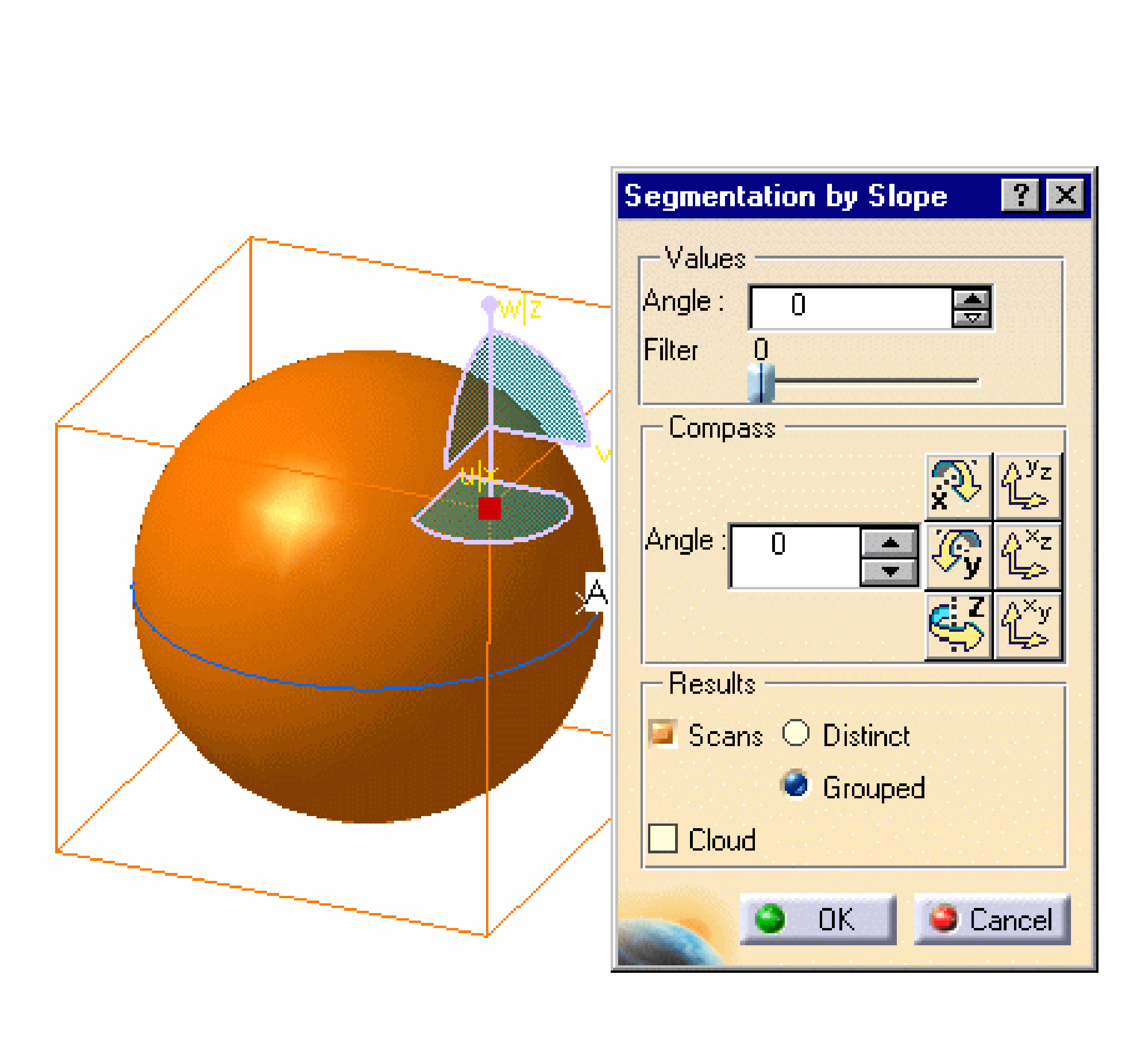
This task shows you how to perform a slope analysis.

This type of analysis identifies lines on the analyzed element where the deviation from the slope direction at any points corresponds to a specified value.

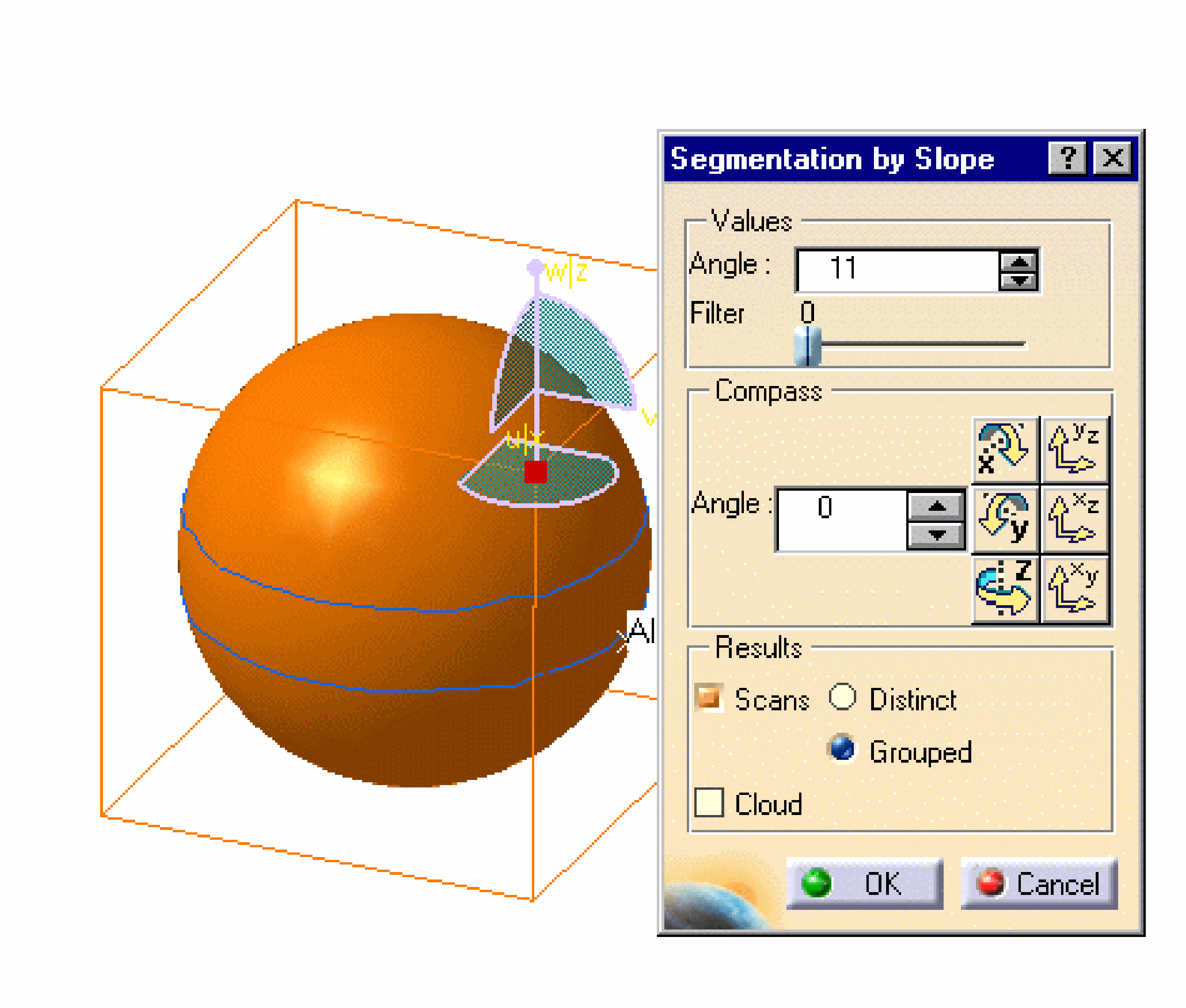
The Z axis gives the view direction. If the deviation angle=0, the lines are the zones of the analyzed element where the normal is orthogonal to the view direction (apparent contour). If the deviation angle is different from 0, the lines are the zones where the normal is orthogonal to the view direction increased by the angle.

Open the Slope1.CATPart from the samples directory.

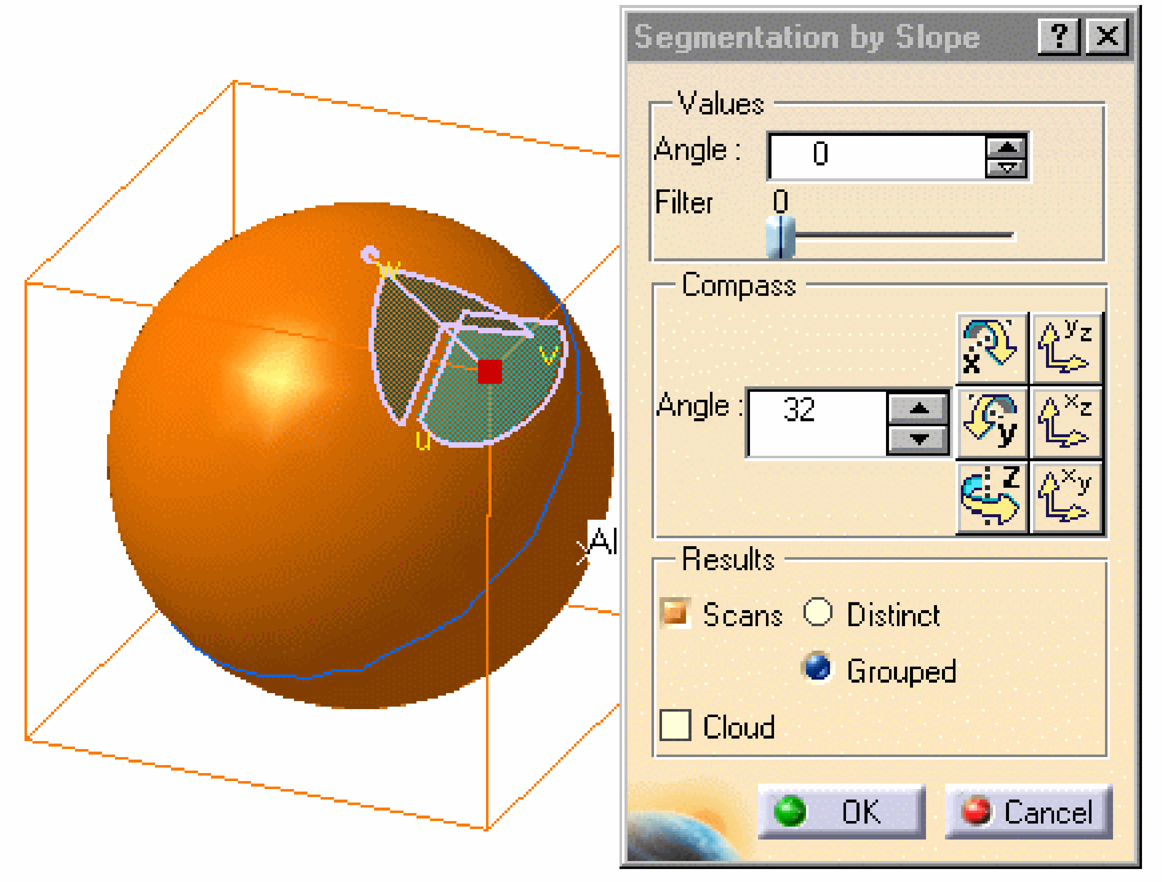
1. Click the **Segmentation by Slope Criterion** icon and select the polygon.
2. The **Segmentation by Slope Criterion** dialog box is displayed. The compass is put on the polygon. By default, the angle is set to 0.



3. Use the **Values/Angle** field to set the angle of the deviation with the view direction.



4. Use the **Compass/Angle** field and the Compass push buttons to set the view direction. Or manipulate the compass as you wish.

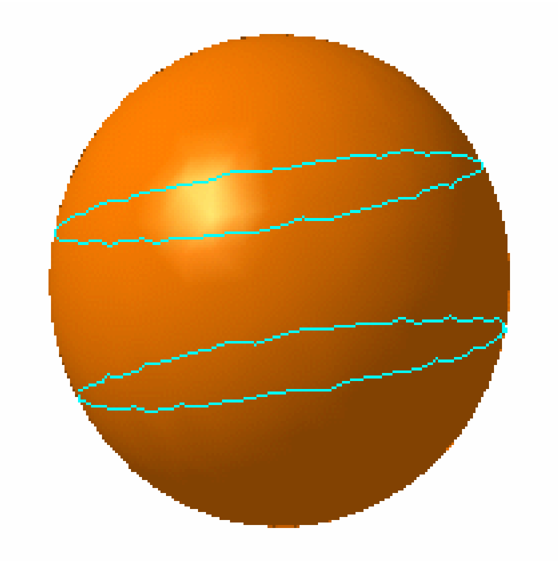


5. Use the **Filter** spinner to reduce the number of points of the lines.

6. If you sweep the cursor on the polygon, the deviation angle is displayed.

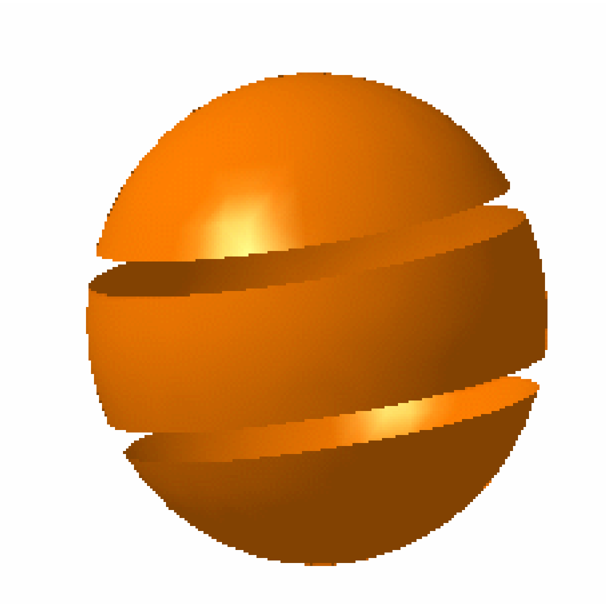
7. Click OK to create the result. You can choose to create:

- Scans, either distinct or grouped:



Scans.x elements are created in the specification tree.

- or Clouds, that is sub-polygons:



Sub Polygon.x elements are created in the specification tree.

These polygons can then be processed with the Basic Surface Recognition action, for example.

The input polygon is sent to the NoShow.

Analysis

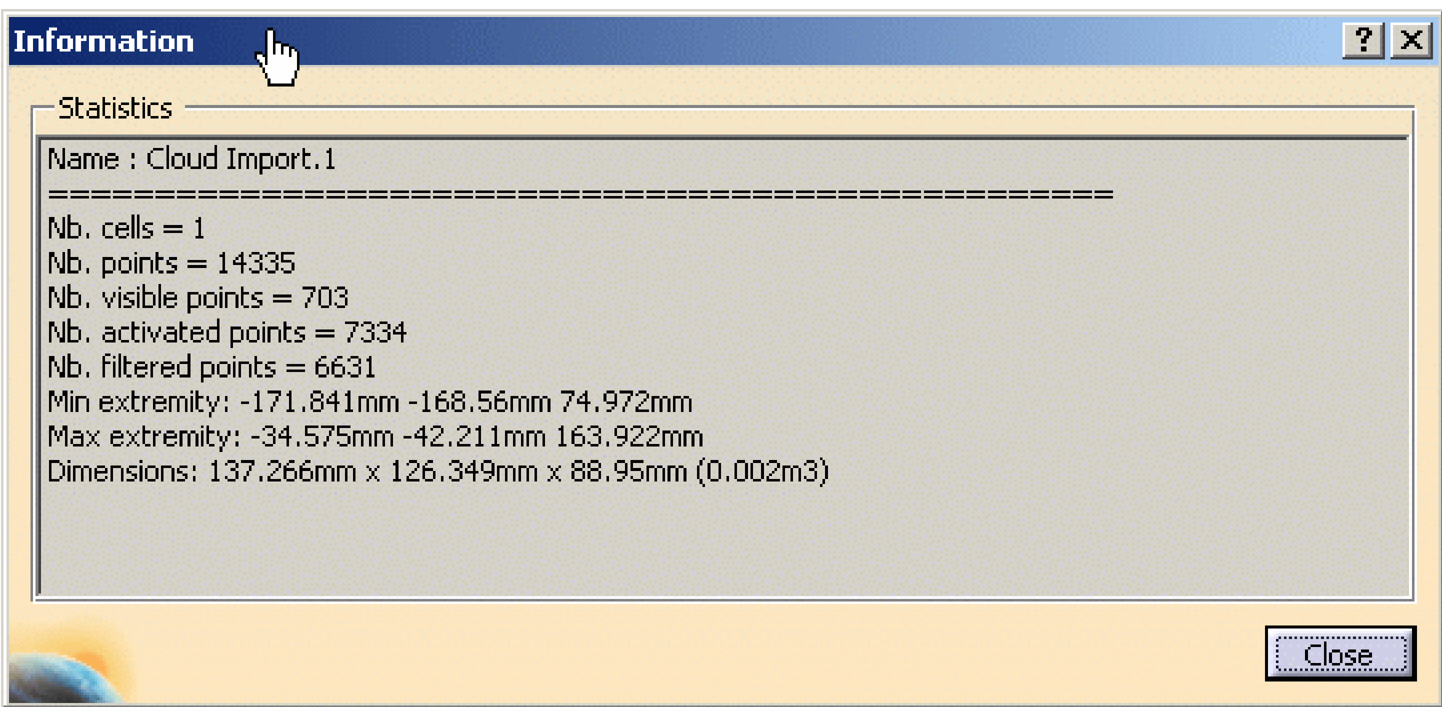
This chapter deals with analyses in Quick Surface Reconstruction.

Information

This task shows you how to get information on a cloud of points.

Open Info1.CATPart from the samples directory.

1. Click the **Information** icon and select a cloud of points.
2. An information box is displayed, with the statistics about the selected cloud:
   - Bounding box ,and active bounding box,
   - Number of points, of active points, of selected points, of filtered points,
   - Total number of triangles and of active triangles.
3. If you select another cloud, the information box is updated with the statistics of that cloud.
4. Push the **Close** button when you are finished to exit the action.



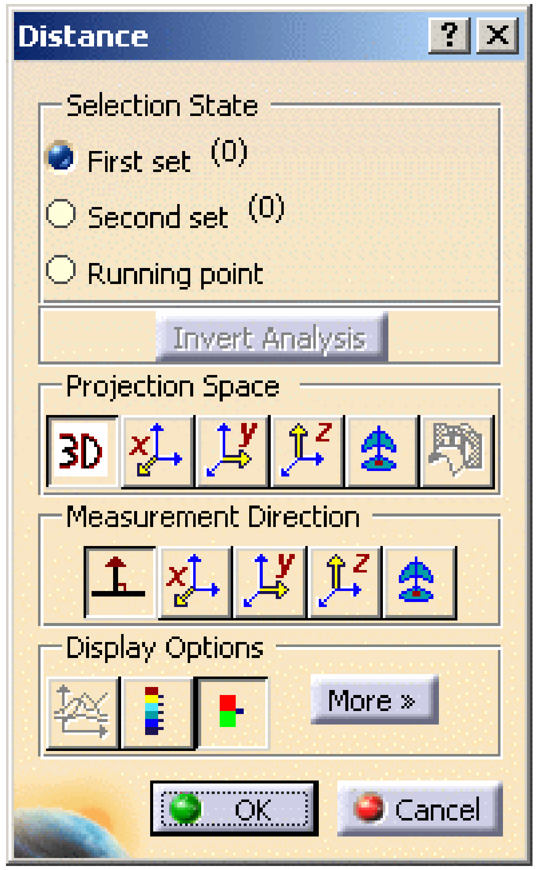
Analyzing Distances Between Two Sets of Elements

This task shows how to analyze the distance between any two geometric elements, or between two sets of elements.

Open the DistanceAnalysis1.CATPart document.

1. Select an element, or set of elements.
2. Click the **Distance Analysis** icon in the Shape Analysis toolbar.

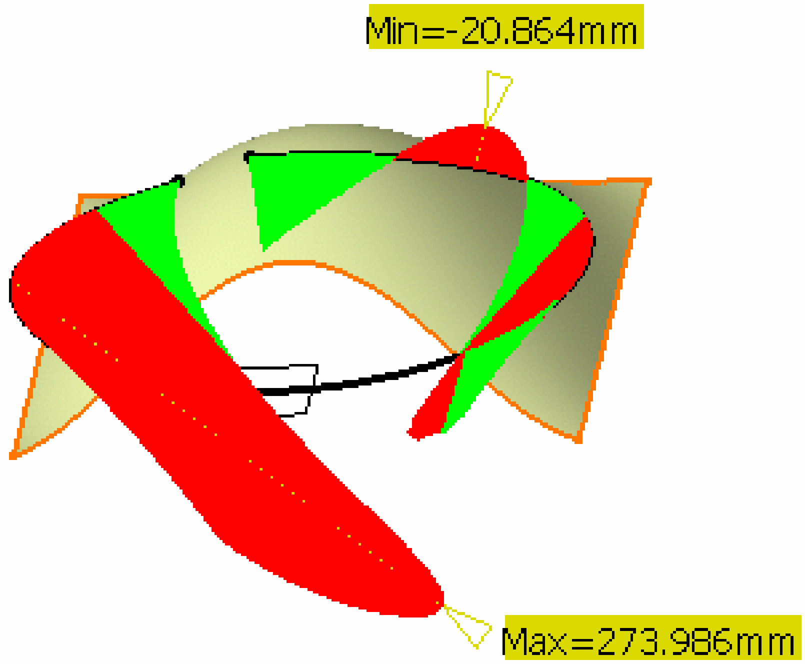
   The Distance dialog box is displayed.



3. Click **Second set** and select a second element, or set.

The distance analysis is computed.

Each color identifies all discretization points located at a distance between two values, as defined in the Color Scale dialog box.

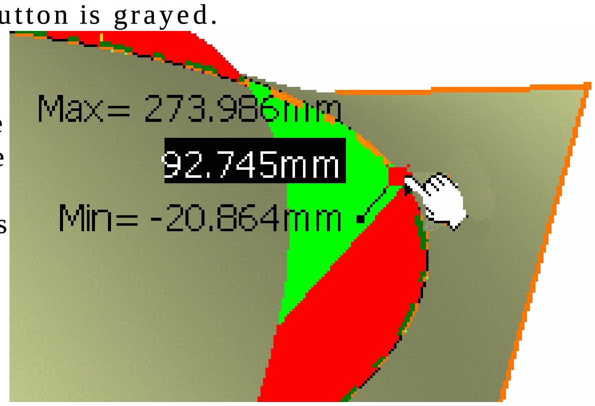


- When computing the distance between two curves, there is no negative values possible as opposed to when analyzing the distance between a surface and another element. Indeed, surfaces present an orientation in all three space directions whereas, in the case of planar curves for example, only two directions are defined. Therefore the distance is always expressed with a positive value when analyzing the distance between two curves.
- The element which dimension is the smallest (0 for points,1 for curves, 2 for surfaces for example) is automatically discretized, if needed.
  When selecting a set of element, the system compares the greatest dimension of all elements in each set, and discretizes the one with the smallest dimension.

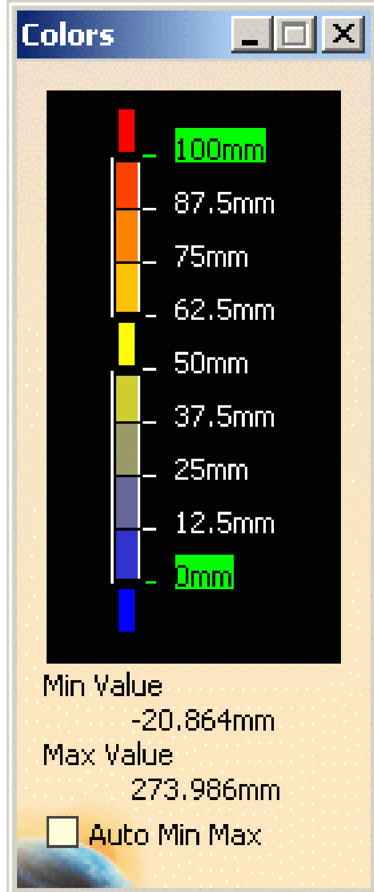
- Use the **Invert Analysis** button to invert the computation direction.

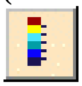
In some cases, when inverting the computation direction does not make sense, when one of the elements is a plane for example, the **Invert Analysis** button is grayed.

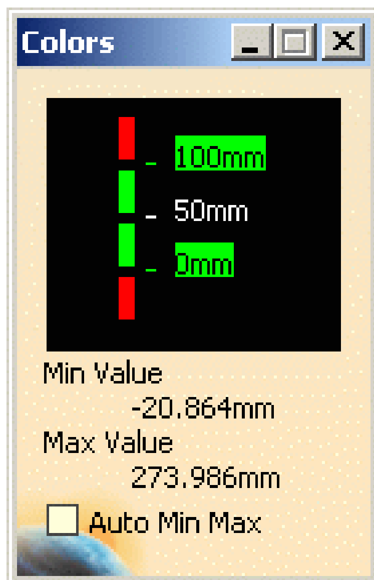
- If you check the **Running point** option, you need to move the pointer over the discretized element to display more precise distance value between the point below the pointer and the other set of elements.
  The projection is visualized and the value is displayed in the geometry area.

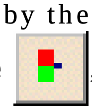


Note that the analyzed point is not necessarily a discretized point in this case. This is obvious when a low discretization value is set, as shown here.
Two analysis modes are available, with corresponding color ramps, provided the **Color scale** checkbox is checked.



a. Full (P2 only): activated by the **Full color range**  icon, it provides a complete analysis based on the chosen color range. This allows you to see exactly how the evolution of the distance is performed on the selected element.



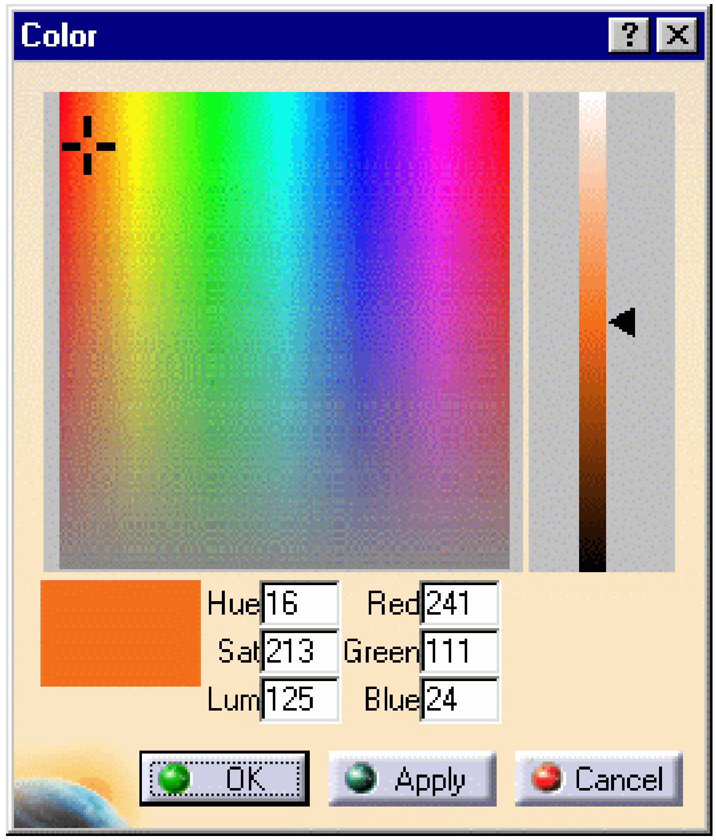
b. Limited: activated by the **Limited color range** , it provides a simplified analysis, with only three values and four colors.

Whichever mode you choose the use of the color scale is identical: it lets you define colors in relation to distance values.

You can define each of the values and color blocks, therefore attributing a color to all elements which distance falls into to given values.

- The **Auto Min Max** button enables to automatically update the minimum and maximum values (and consequently all values between) each time they are modified.

- You can right-click on a color in the color scale to display the contextual menu:

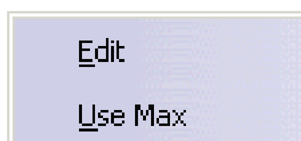


- Edit: it allows you to modify the values in the color range to highlight specific areas of the selected surface. The Color dialog box is displayed allowing the user to modify the color range.

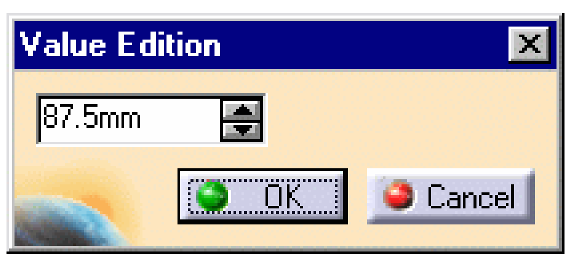
- **Unfreeze**: it allows you to perform a linear interpolation between non defined colors. The unfreezed values are no longer highlighted in green.

- **No Color**: it can be used to simplify the analysis, because it limits the number of displayed colors in the color scale. In this case, the selected color is hidden, and the section of the analysis on which that color was applied takes on the neighboring color.

- You can also right-click on the value to display the contextual menu:



- **Edit**: it allows you to modify the edition values. The Value Edition dialog box is displayed: enter a new value (negative values are allowed) to redefine the color scale, or use the slider to position the distance value within the allowed range, and click OK.
The value is then frozen, and displayed in a green rectangle.



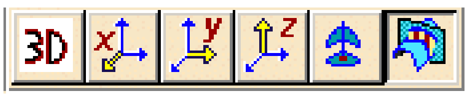
- **Use Max/Use Min** : it allows you to evenly distribute the color/value interpolation between the current limit values, on the top/bottom values respectively, rather than keeping it within default values that may not correspond to the scale of the geometry being analyzed. Therefore, these limit values are set at a given time, and when the geometry is modified after setting them, these limit values are not dynamically updated.

For now only the linear interpolation is allowed, meaning that between two set (or frozen) colors/values, the distribution is done progressively and evenly.

The color scale settings (colors and values) are saved when exiting the command, meaning the same values will be set next time you edit a given distance analysis capability.
However, new settings are available with each new distance analysis.

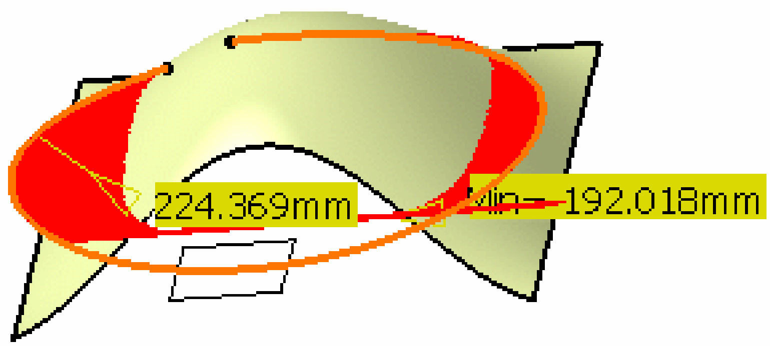
5. Set the distance analysis type (we checked the Auto Scale button and unchecked the Min/Max values button):

Projection Space

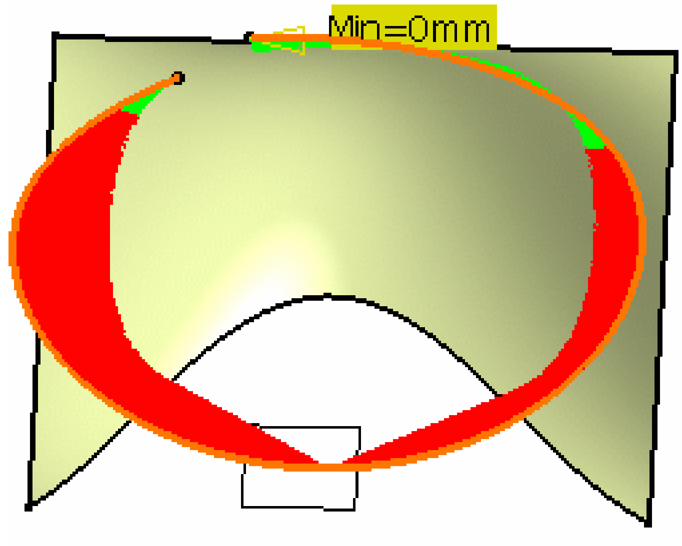


The **Projection Space** area helps you define the preprocessing of the input elements used for the computation.

This frame is only available when analyzing distances between curves.

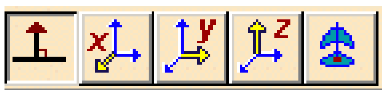


- **3D** : elements are not modified and the computation is done between the initial elements.
- Projection according to the **X** , **Y** , or **Z** axis: the computation is done between the projection of selected elements.
- Projection according to the **compass** current orientation : the computation is done between the projection of selected elements.

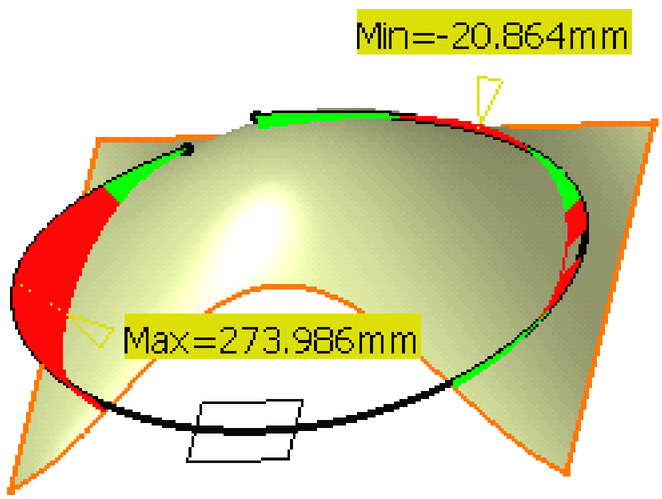


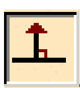
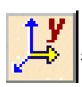
- **Planar distance**  : the distance is computed between a curve and the intersection of the plane containing that curve.

Measurement Direction

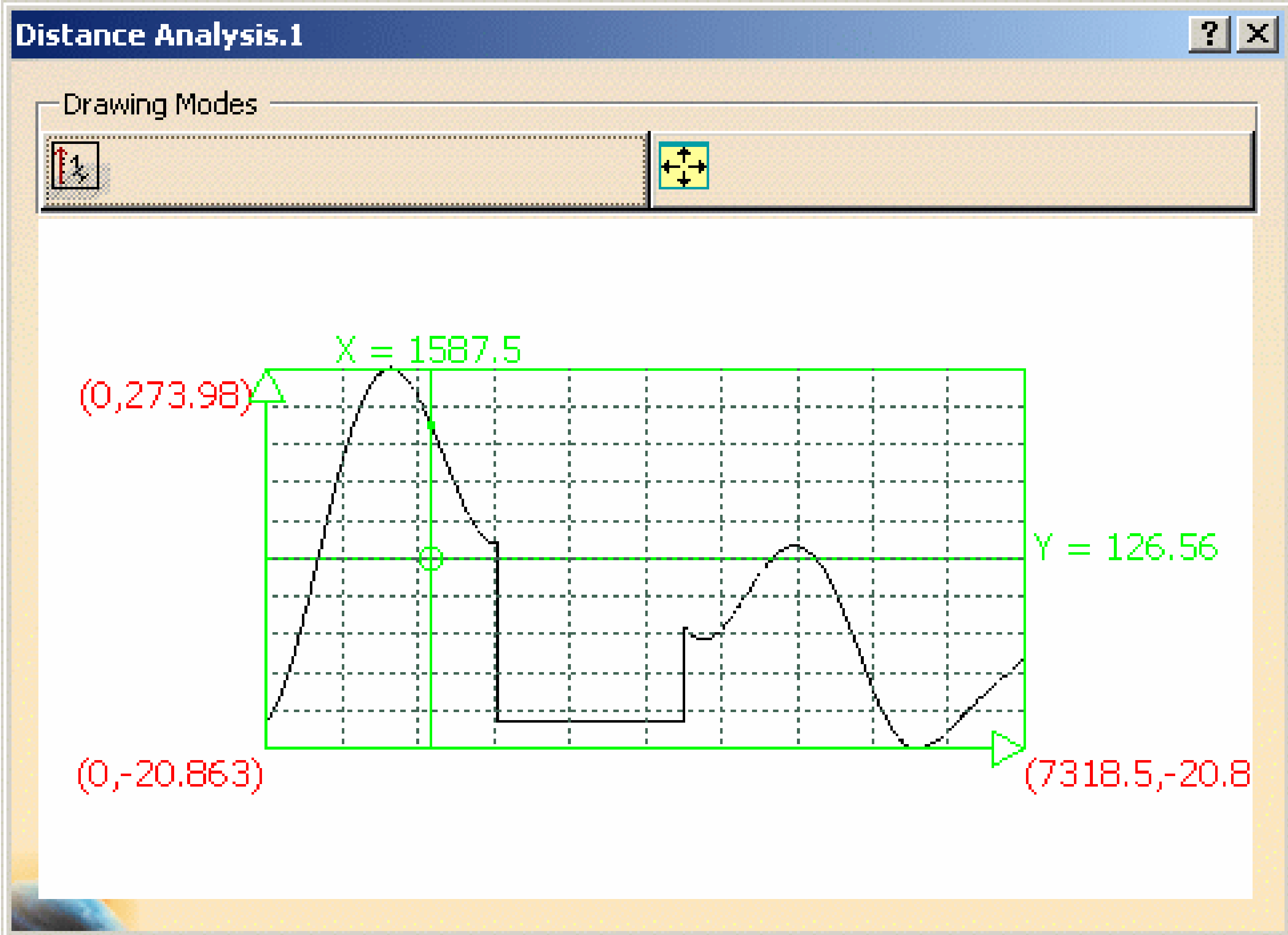


The Measurement Direction area provides options to define how set the direction used for the distance computation.

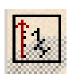
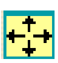


- **Normal distance**  : the distance is computed according to the normal to the other set of elements.
- Direction according to the **X** , **Y** , or **Z**  axis.
- Direction according to the **compass** current orientation .

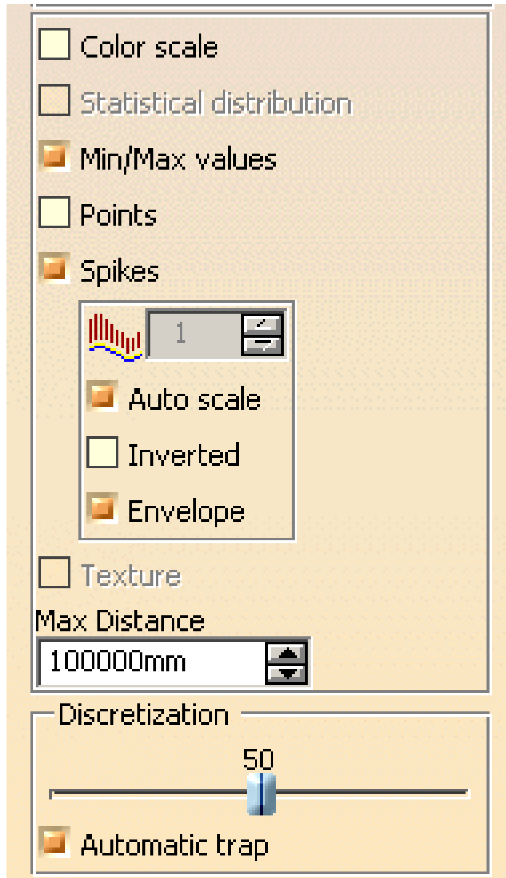
6. Click the  icon to display the 2D diagram distance analysis window. The latter allows to visualize the distance evolution.



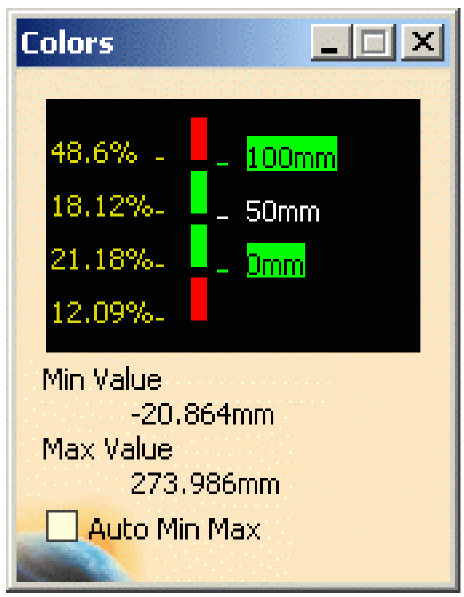
Drawing modes:

- **Vertical Inverse Scale**  : to draw the curves in a linear horizontal scale and and inverse vertical scale.
- **Reframe**  : to reframe the frame

7. Click **More>>** in the Distance dialog box to see, and choose further display and discretization options:



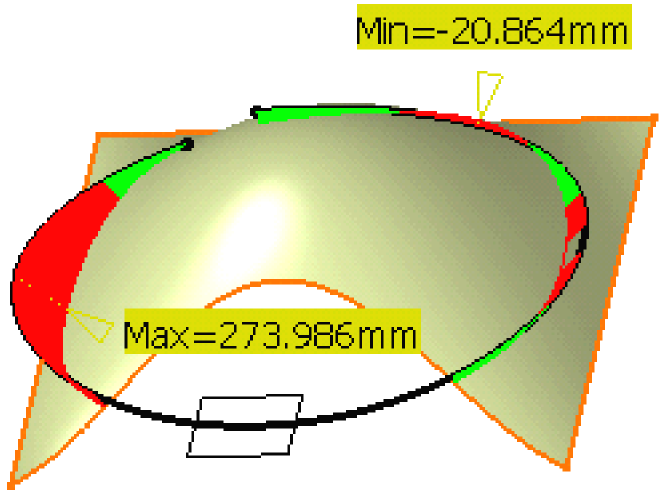
- **Color scale**: to display the Color Scale dialog box whether the full or the limited color range.



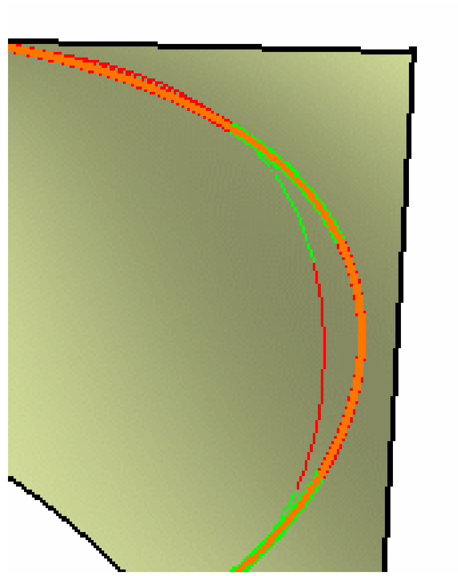
- **Statistical distribution**: to display the percentage of points between two values.

This option is only available if the **Color Scale** checkbox is checked.

- **Min/Max values**: to display the minimum and maximum distance values and locations on the geometry.



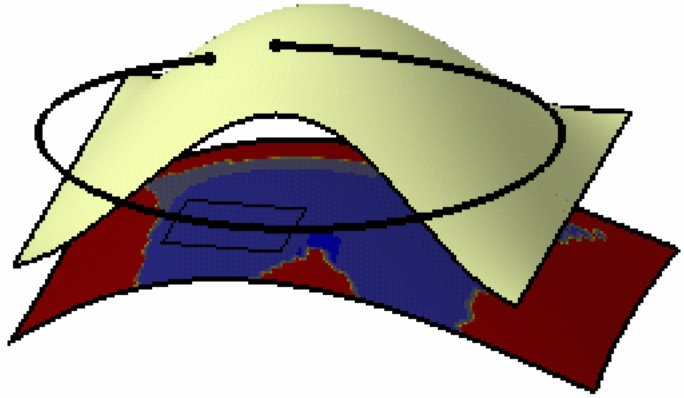
- **Points**: to see the distance analysis in the shape of points only on the geometry (The **Spikes** button is unchecked)



- **Spikes**: to see the distance analysis in the shape of spikes on the geometry. You can further choose to:
  - set a ratio for the spike size
  - choose an automatic optimized spike size (**Auto scale**)
  - invert the spike visualization on the geometry
  - display the envelope, that is the curve connecting all spikes together

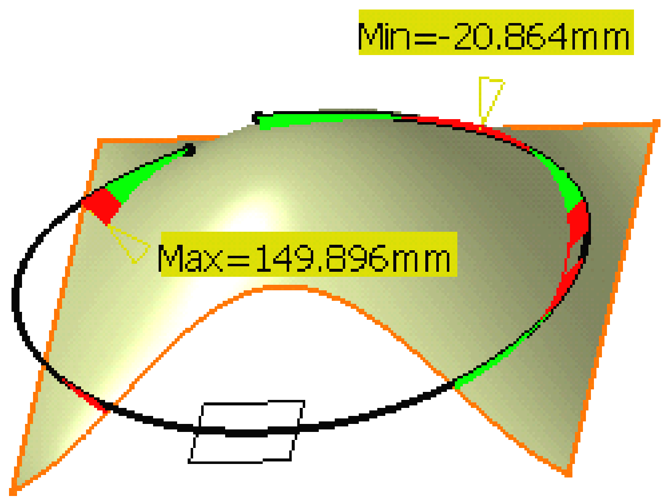
- Use the **Texture** option to check the analysis using color distribution.

- This option is only available with surfacic elements in at least one set, providing this set is discretized.
  The distance is computed from this discretized set to the other set. The texture mapping is computed on the discretized surface.
  It is not advised to use it with planar surfaces or ruled surfaces.



- **Statistical distribution**, **Min/Max values**, and **Points** cannot be visualized when using the Texture option.
- The visualization mode should be set to **Shading with Texture and Edges**, and the discretization option should be set to a maximum (in *CATIA Infrastructure User Guide*, see Improving Performances, the 3D **Accuracy -> Fixed** option should be set to 0.01).
  Check the **Materials** option in the **View -> Render Style -> Customize View** command to be able to see the analysis results on the selected element. Otherwise a warning is issued.

- Use the **Max Distance** option to relimit the distance: for example, set the value to 150mm. The maximum value is displayed accordingly on the geometry.



- Use the **Discretization** option to reduce or increase the number of points of the second set of elements taken into account when computing the distance.
- **Automatic trap**: to delimit the second set of points to be taken into account for the computation, in the case of a large cloud of points, thus improving the performances.
  Be careful when using the **Automatic trap** option with certain cloud configurations, such as spiralling clouds of points for example, as the automatic trap may remove too many points to generate consistent results.
  In this case, it is best to deactivate the check button.

8. Click **OK** to exit the analysis while retaining it.

The analysis (identified as Distance Analysis.x) is added to the specification tree.

- Even though you exit the analysis, the color scale is retained till you explicitly close is. This is like a shortcut allowing you to modify one of the analyzed elements, which leads to a dynamic update of the distance analysis, while viewing the set values/colors at all times and without having to edit the distance analysis.
- When analyzing clouds of points, in normal projection type, the distances are computed as the normal projection of each point of the first cloud onto the triangle made by the three points closest to that projection onto the second cloud.
As it is a projection, using the Invert Analysis button does not necessarily gives symmetrical results.
- When you select the geometrical set as an input in the specification tree, all the elements included in this geometrical set are automatically selected too.

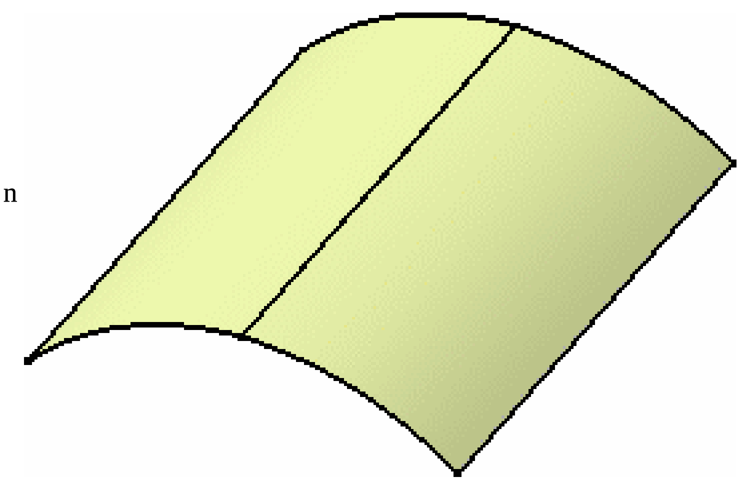
The auto detection capability is available from the FreeStyle Dashboard.

Checking Connections Between Surfaces

This task shows how to analyze how two surfaces are connected, following a blend, match, or fill operation for example.
Three types of analyses are available.

- **Distance**: the values are expressed in millimeters
- **Tangency**: the values are expressed in degrees
- **Curvature**: the values are expressed in percentage.

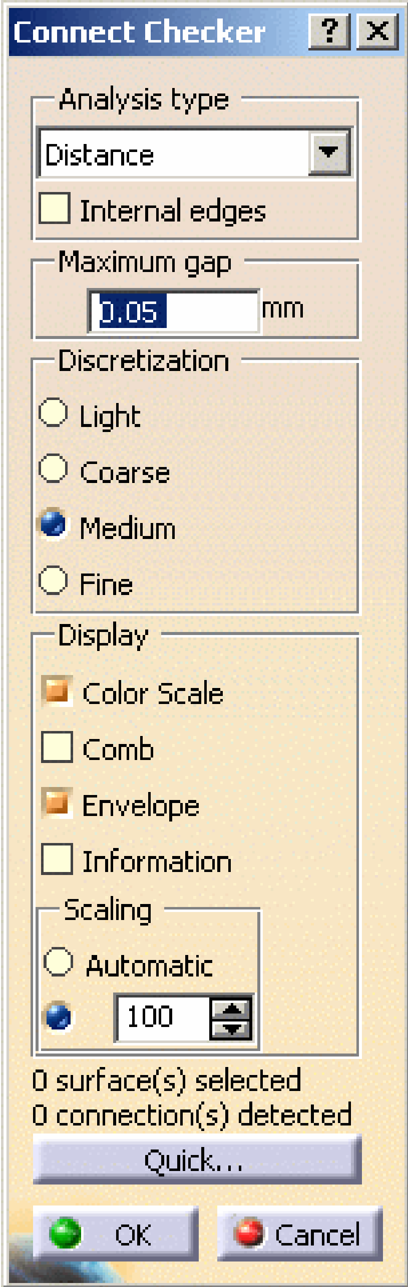
Open the ConnectChecker1.CATPart document.

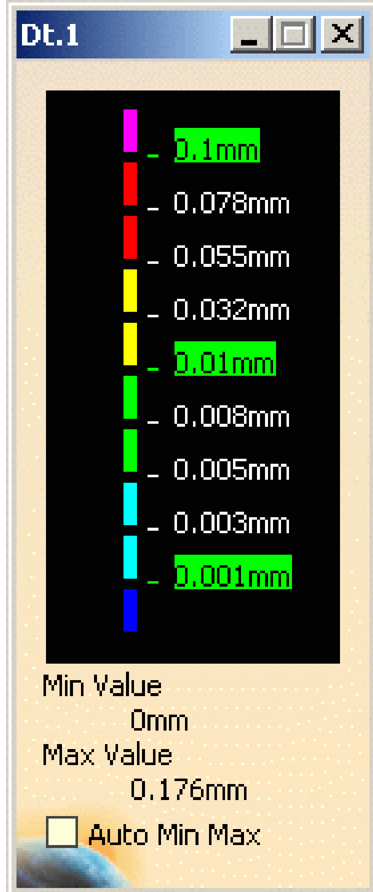


1. Select both surfaces to be analyzed.
2. Click the **Connect Checker** icon  in the Shape Analysis toolbar.

   The Connect Checker dialog box is displayed as well as another dialog box showing the color scale and identifying the maximum and minimum values for the analysis type.

   The **Auto Min Max** button enables to automatically update the minimum and maximum values (and consequently all values between) each time they are modified.





Check the Internal edges option if you want to analyze the internal connections.
By default, the check box is unchecked.

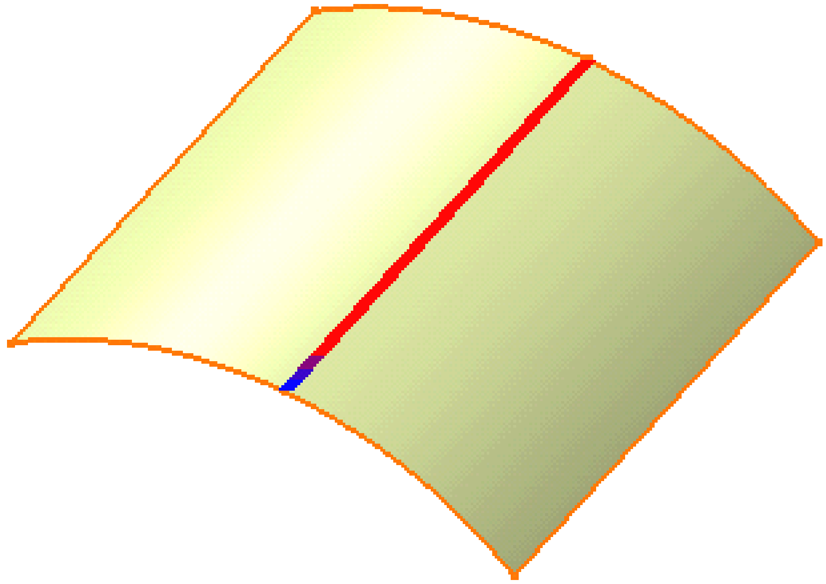
3. Choose the analysis type to be performed: **Distance**, **Tangency** or **Curvature**.

4. Set the **Maximum gap** above which no analysis will be performed. All elements apart from a greater value than specified in this field are considered as not being connected, therefore do not need to be analyzed.

   Be careful not to set a **Maximum gap** greater than the size of the smallest surface present in the document.

5. Check the analysis results on the geometry.

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



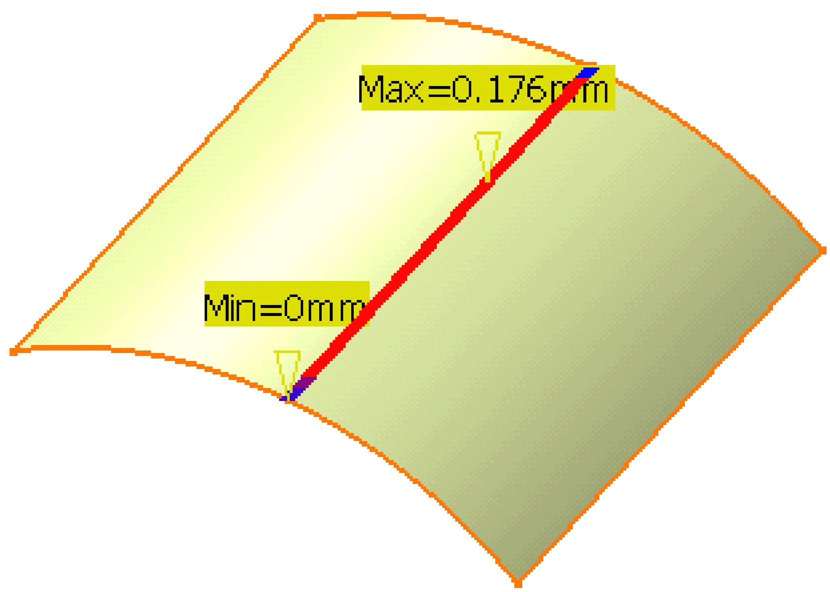
From the Connect Checker dialog box, you can choose a number of visualization and computation options:

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

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

6. Check the **Information** button:

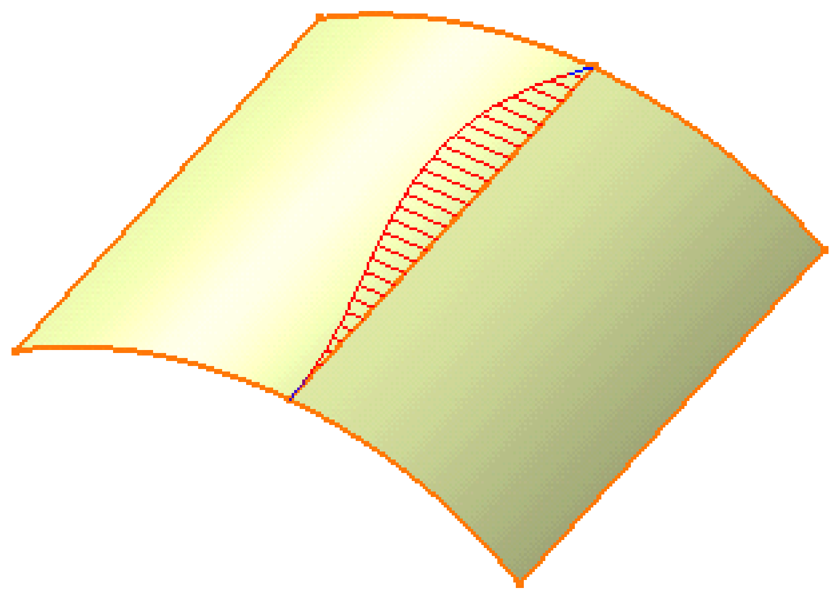
Two texts are displayed on the geometry localizing the minimum and maximum values of the analysis as given in the Connect Checker dialog box.



You can also choose the discretization, that is the numbers of spikes in the comb (check the **Comb** option to see the difference).
The number of spikes corresponds to the number of points used for the computation:

- **Light**: 5 spikes are displayed.

This mode enables to obtain consistent results with the visualization of sharp edges.
An edge is considered as sharp if its tangency deviation is higher than 0.5 degree. To only detect tangency deviations on sharp edges, specify a deviation of 0.5 degree minimum.



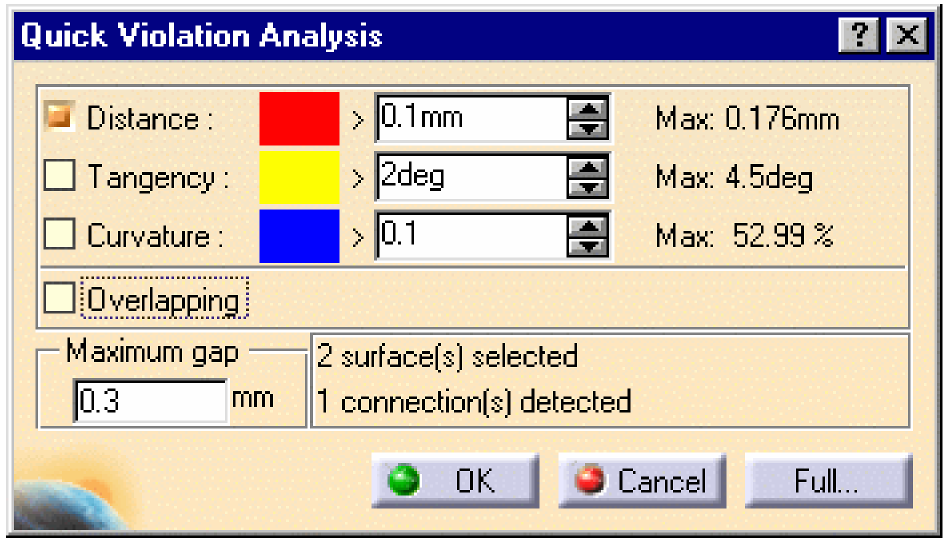
To visualize sharp edges, make sure the **View -> Render Style -> Shading with Edges and Hidden Edges** option is checked.

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

The Full result is only available with the Generative Shape Design 2 product.

The number of selected elements and the number of detected connections are displayed below the color range.

7. Click the **Quick...** button to obtain a simplified analysis taking into account tolerances (either distance, tangency, or curvature).



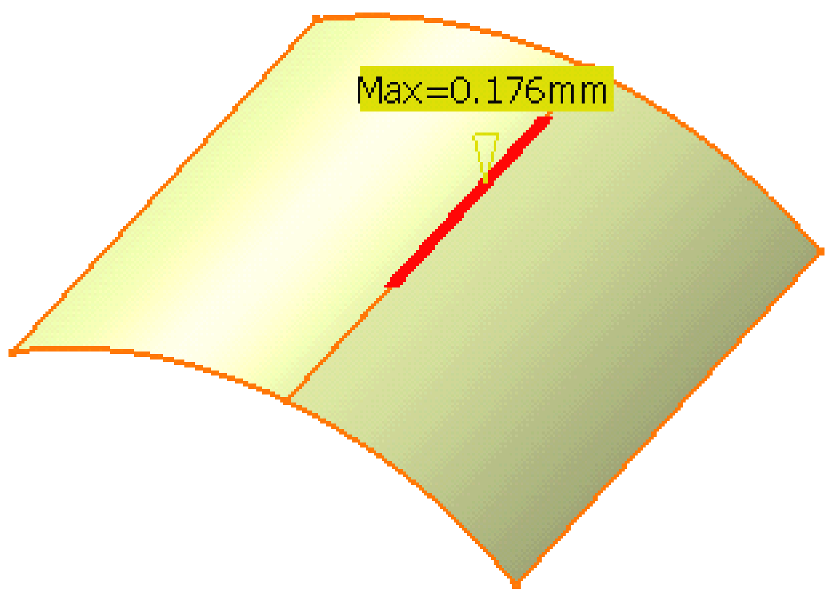
The comb is no longer displayed.
The Connect Checker dialog box changes to this dialog box.

You can use the check button to switch from one analysis type to another.
The **Maximum gap** and information are retained from the full analysis.
The maximum deviation value is also displayed on the geometry.

You can check the **Overlapping** button to highlight where, on the common boundary, the two surfaces overlap. In this case the other analysis types are deactivated.

In P1 mode, only the quick analysis is available.

8. Use the spinners to define the deviation tolerances.



For example, the red area indicates all points that are distant of more than 0.1 mm.

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

9. Click **OK** to create the analysis.

The analysis (identified as Surface Connection Analysis.x) is added to the specification tree (P2 only).

This allows the automatic update of the analysis when you modify any of the surfaces, using the control points for example.

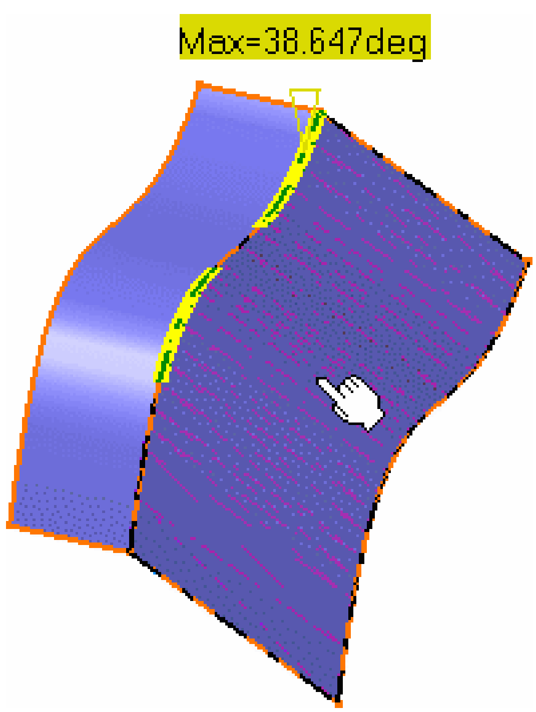
If you do not wish to create the analysis, simply click **Cancel**.

- You can edit the color range in both dialog boxes by double-clicking the color range manipulators (Connect Checker) or color areas (Quick Violation Analysis) to display the **Color** chooser.
- If you wish to edit the Connection Analysis, simply double-click it from the specification tree.
- If you no longer need the Connection Analysis, right-click **Connection Analysis** in the specification tree, and choose **Delete**.
- The curvature difference is calculated with the following formula:

$$(|C2 - C1|) / ((|C1 + C2|) / 2)$$

The result of this formula is between 0% et 200%.

- You can analyze internal edges of a surfacic element, such as a Join for example, by selecting only one of the initial elements:



- You can create an analysis on an entire geometrical set simply by selecting it in the specification tree.

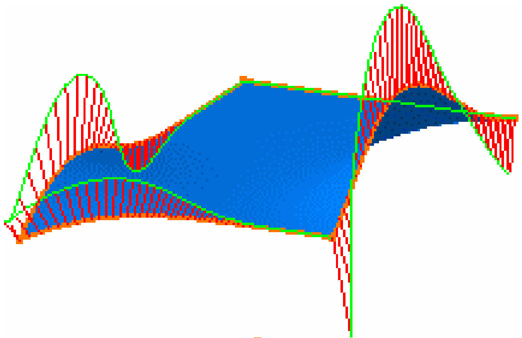
Performing a Curvature Analysis

This task shows how to analyze the curvature of curves, or surface boundaries.

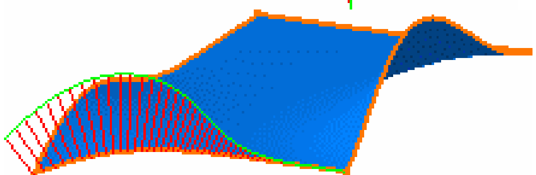
Open the Analysis1.CATPart document.

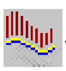
When analyzing surface boundaries:

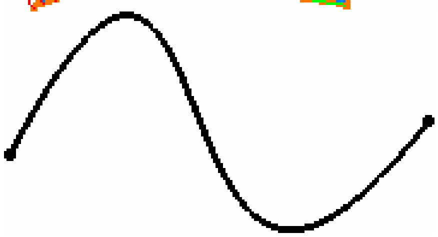
- if you select the surface, the analysis is performed on all its boundaries



- if you select a specific boundary, the analysis is performed only on this boundary.

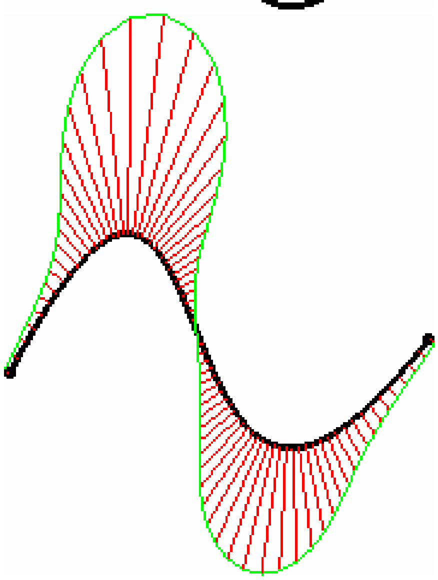


1. Click the **Porcupine Curvature Analysis** icon .

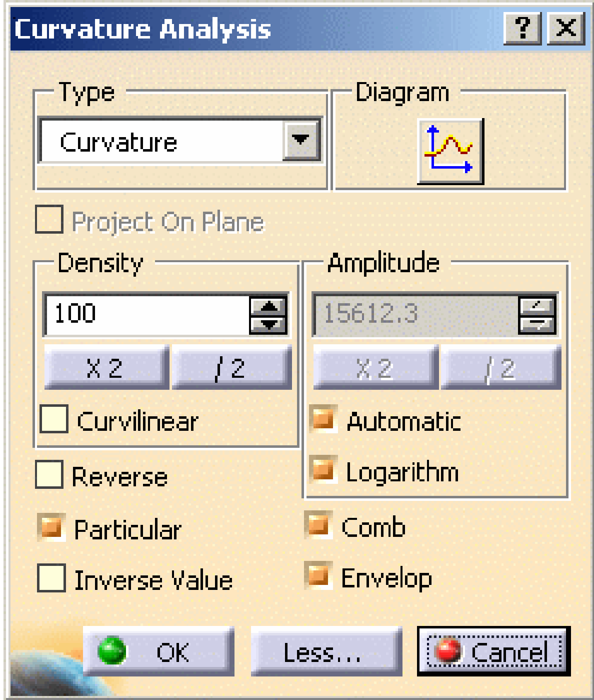


2. Select the curve.

Automatically the curvature comb is displayed on the selected curve:



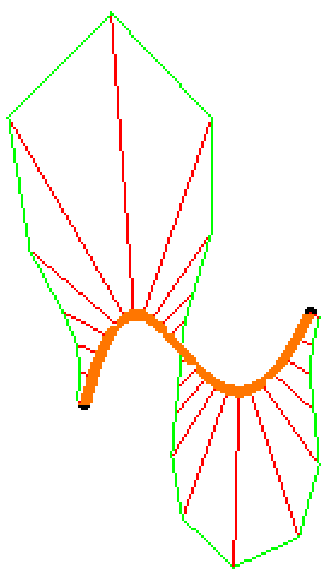
3. Define the analysis parameters in the Curvature Analysis dialog box.



- Use the **Project on Plane** checkbox to analyze the projected curve in the selected plane referenced by the compass.
- If you uncheck the **Project On Plane** option, the analysis is performed according to the curve orientation. This is the default option.

4. Use the spinners to adjust the number of strikes and modify the density.

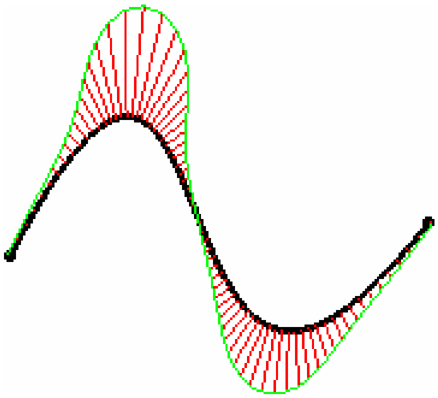
5. You can also decide to halve the number of spikes in the comb clicking as many times as wished the **/ 2** button.



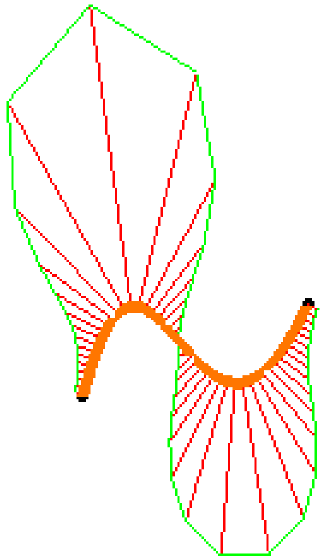
This option is particularly useful when the geometry is too dense to be read but the resulting curve may not be smooth enough for your analysis needs.

You could just as well double the number of spikes using the **X2** button.

6. Similarly, click the **/ 2** button to fine-tune the amplitude (size) of the spikes, and re-compute the analysis curve accordingly.

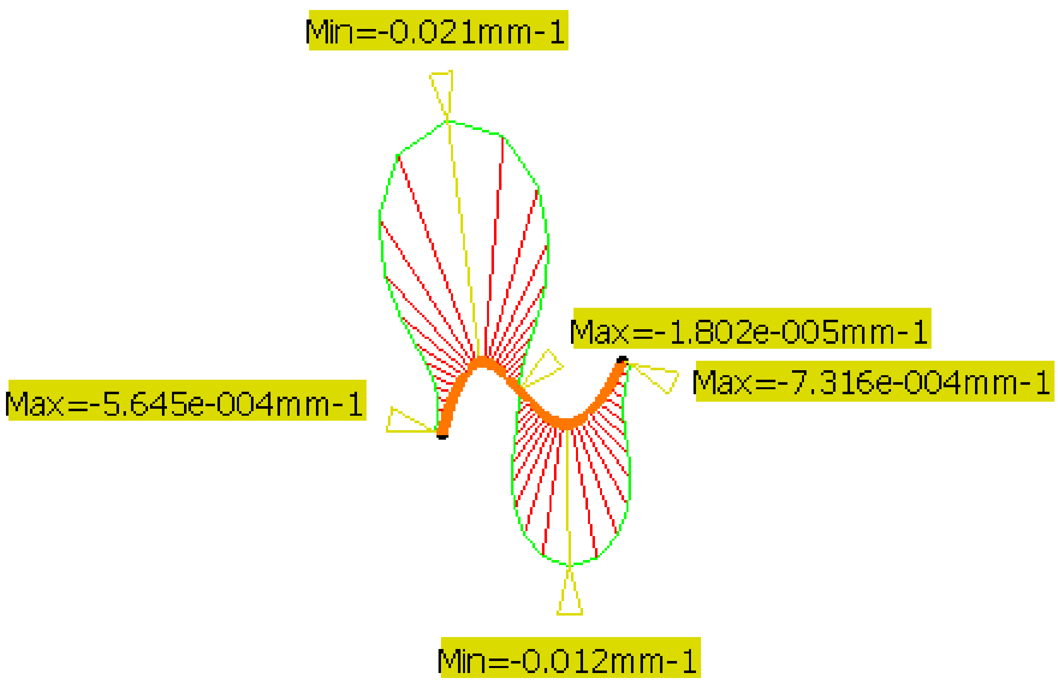


7. Click **Curvilinear** to switch from the Parametric discretization mode to the Curvilinear analysis. You will get something like this:



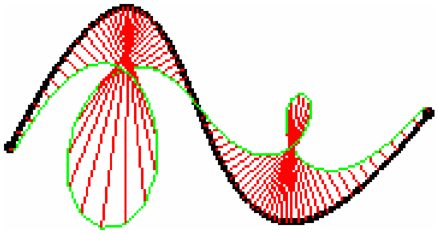
8. Check the **Automatic** option optimizes the spikes length so that even when zooming in or out, the spikes are always visible.

9. Check the **Logarithm** option to display the logarithmic values in the 3D geometry.



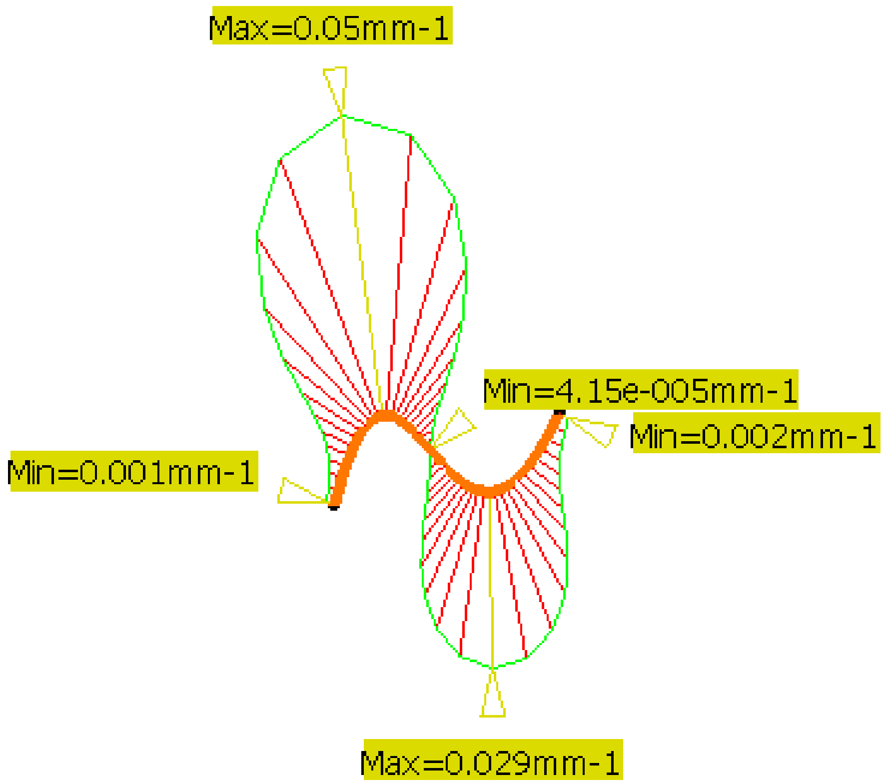
Displaying these values does not modify the analysis.

10. Click **Reverse**, you will get something like this:



That is the analysis opposite to what was initially displayed. This is useful when from the current viewpoint, you do not know how the curve is oriented.

11. Use the **Particular** checkbox to display at anytime the minimum and the maximum points.



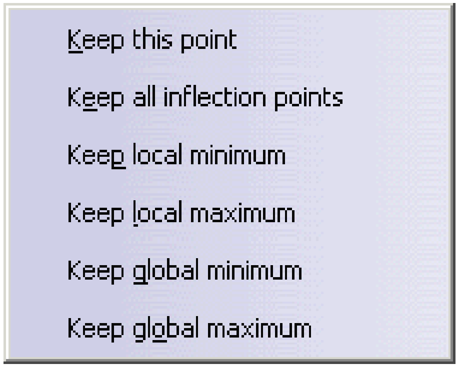
Inflection points are displayed only if the **Project on Plane** and Particular checkboxes are checked.

12. The **Inverse Value** checkbox displays the inverse value in **Radius**, if **Curvature** option is selected, or in **Curvature**, if **Radius** option is selected.

- You can right-click on any of the spikes and select **Keep this Point** to keep the current point at this location.
  A Point.xxx appears in the specification tree.
  If you check the **Particular** option, you have more options:
  - **Keep all inflection points**
  - **Keep local minimum** (corresponds to the absolute minimum under

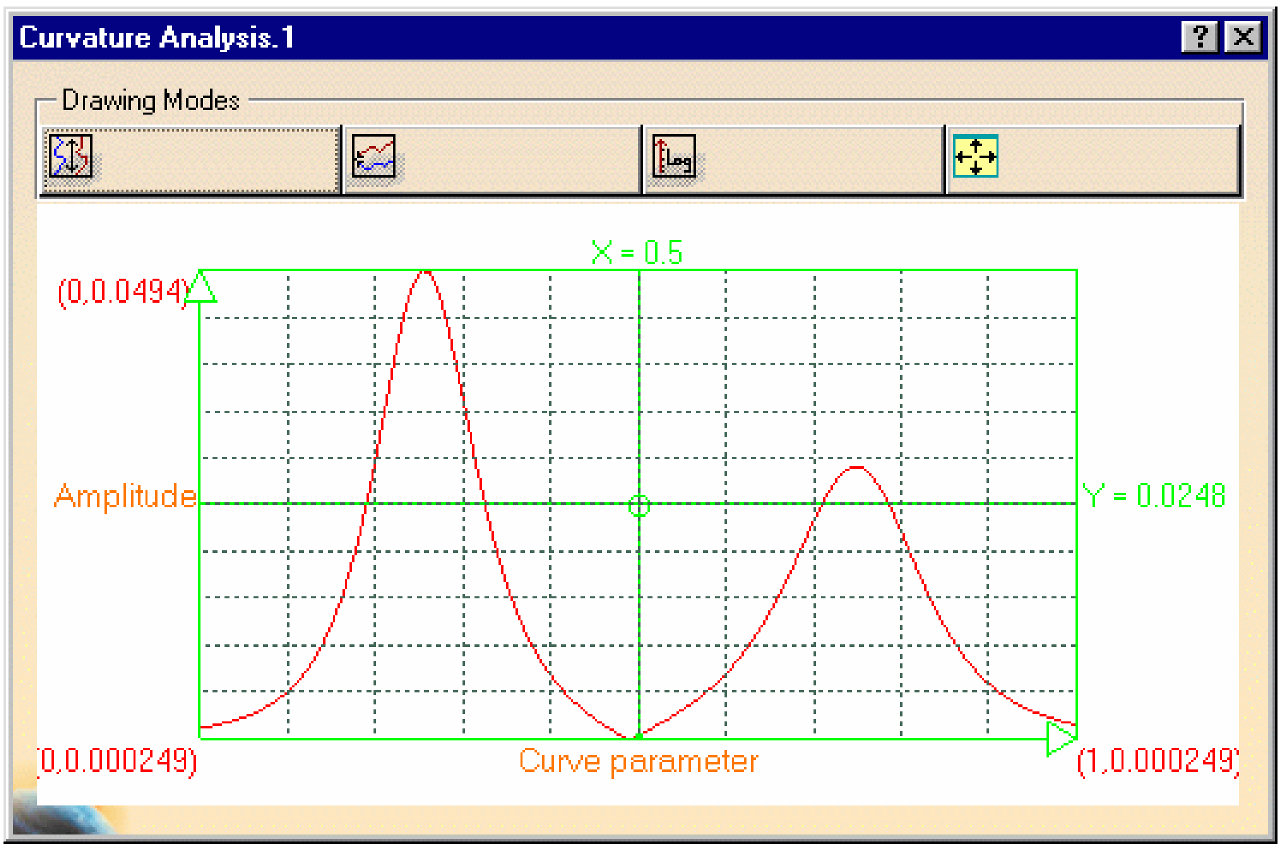
the running point)

- **Keep local maximum** (corresponds to the absolute maximum under running point)
- **Keep global minimum** (in case there are two curves, the point will be found on one or other of the curves)
- **Keep global maximum** (in case there are two curves, the point will be found on one or other of the curves)



P2 This option is only available in P2 mode in FreeStyle Shaper, Optimizer, and Profiler.

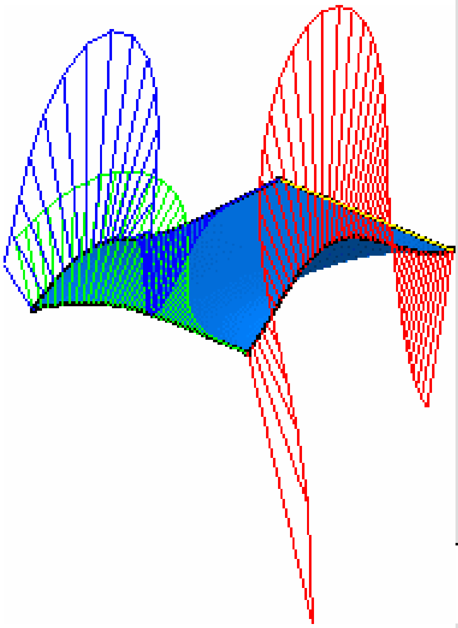
13. Finally, click the  icon to display the curvature graph:

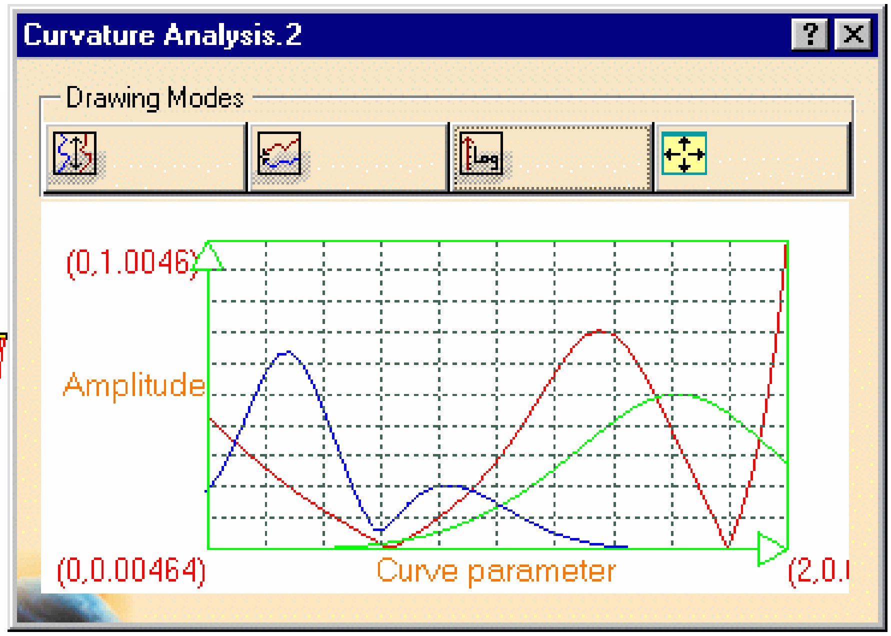


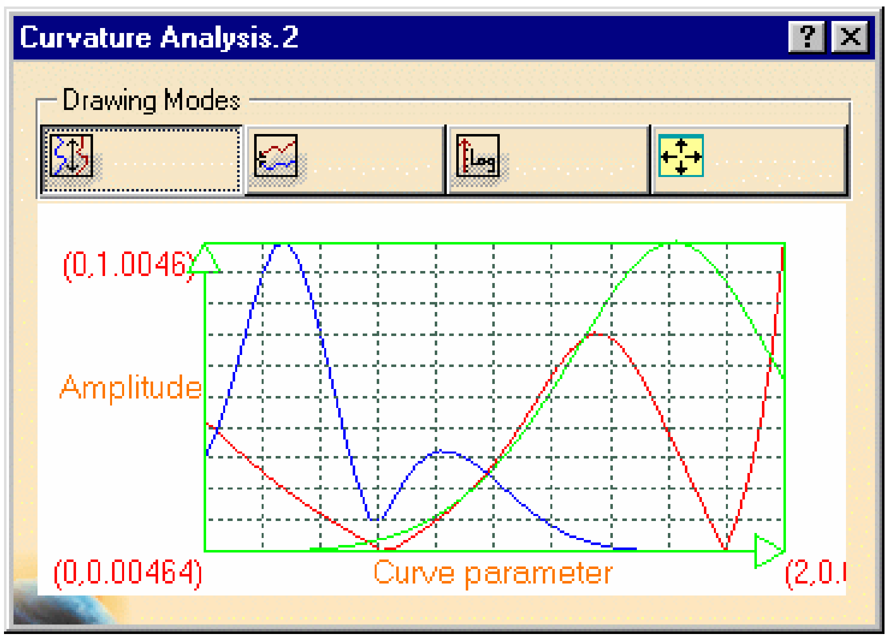
The curvature profile and amplitude of the analyzed curve is represented in this diagram.

When analyzing a surface or several curves, i.e. when there are several curvature analyses on elements that are not necessarily of the same size for example, you can use different options to view the analyses.

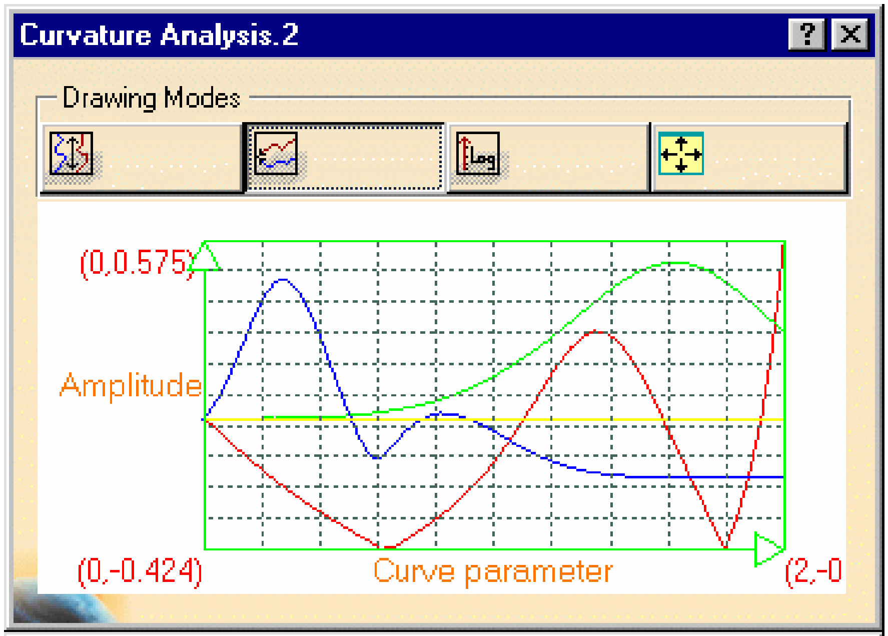
For example, when analyzing a surface, by default you obtain this diagram, where the curves color match the ones on the geometry.

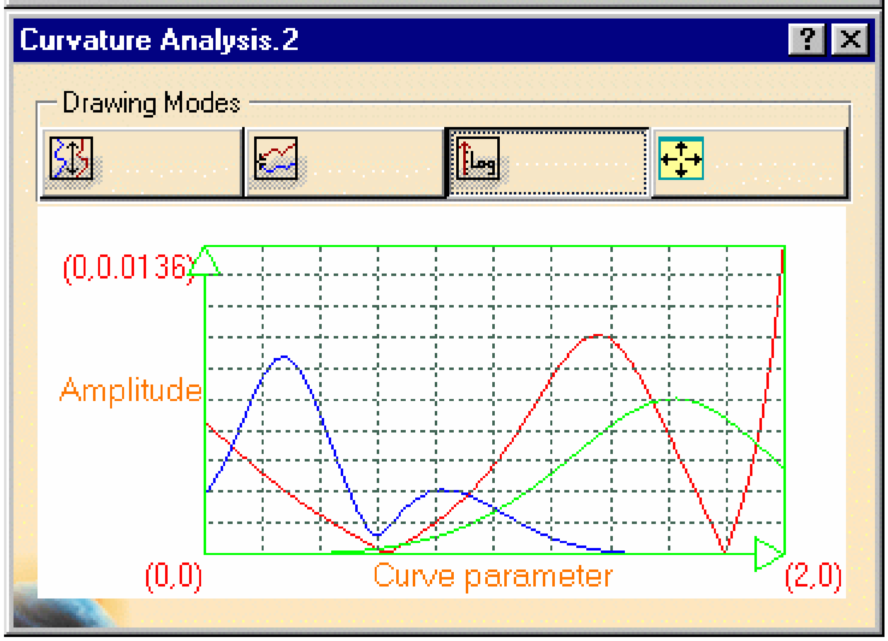






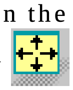
- **Same vertical length** : all curves are displayed according to the same vertical length, regardless of the scale
- **Same origin** : all curves are displayed according to a common origin point on the **Amplitude** scale



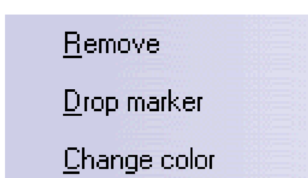


- **Vertical logarithm scale** : all curves are displayed according to a logarithm scale for the **Amplitude**, and a linear scale for the **Curve parameter**.

Depending on the chosen option, values displayed in the diagram are updated.

The last icon  is used to reframe the diagram within the window, as you may move and zoom it within the window.

14. Right-click a curve and choose one of the following options from the contextual menu:



- **Remove**: removes the curve
- **Drop marker**: adds Points.xxx in the specification tree
- **Change color**: displays the Color selector dialog box that enables you to change the

color of the curve.

15. Slide the pointer over the diagram to display the amplitude at a given point of the curve.

    You can slide the pointer over the diagram and the 3D analysis.

    Click the x in the top right corner to close the diagram.

16. Click OK in the Curvature Analysis dialog box once you are satisfied with the performed analysis.

    The analysis (identified as Curvature Analysis.x) is added to the specification tree.

In case of clipping, you may want to temporarily modify the Depth Effects' **Far** and **Near Limits**. See Setting Depth Effects in *CATIA Infrastructure User Guide*.

WireFrame

Creating Points

This task shows the various methods for creating points:

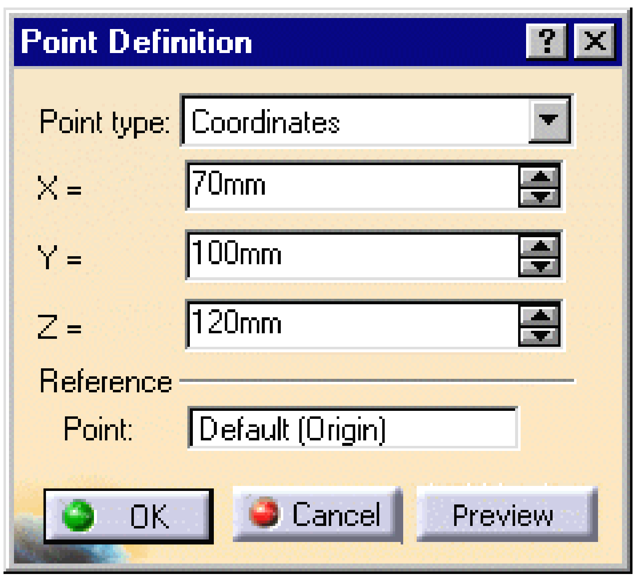
- by coordinates
- on a curve
- on a plane
- on a surface
- at a circle center
- tangent point on a curve
- between

Open the Points3D-1.CATPart document.

1. Click the **Point** icon .

   The Point Definition dialog box appears.

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



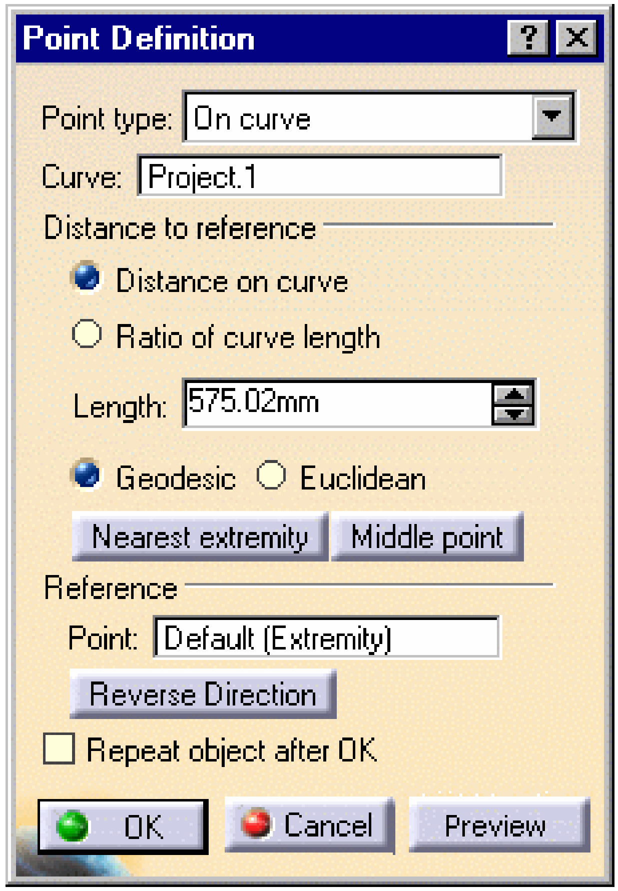
Coordinates

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

The corresponding point is displayed.

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

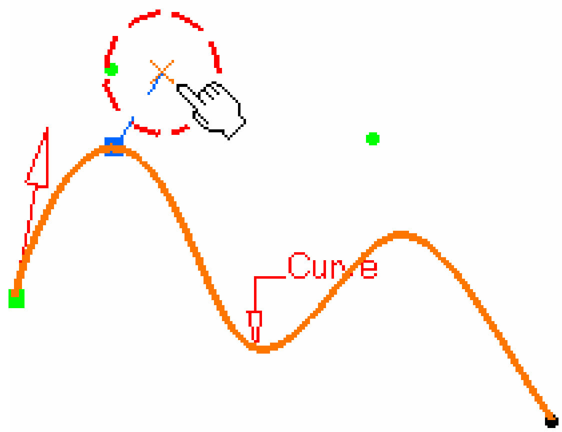
The axis system must be different from the absolute axis.



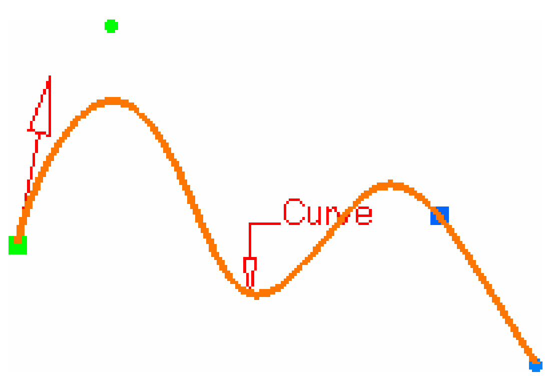
On curve

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

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



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



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

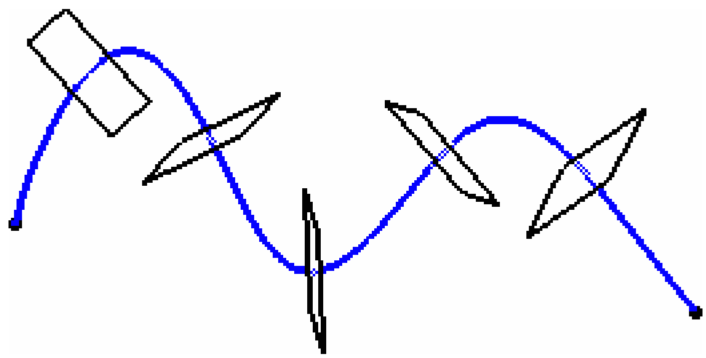
  The corresponding point is displayed.

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

You can also:

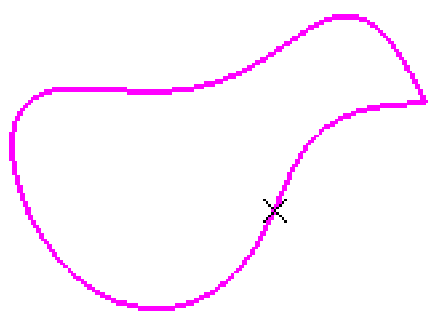
- click the **Nearest extremity** button to display the point at the nearest extremity of the curve.
- click the **Middle Point** button to display the mid-point of the curve.
- use the **Reverse Direction** button to display:
  - the point on the other side of the reference point (if a point was selected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

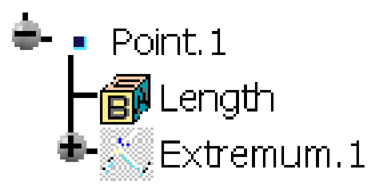
  You will also be able to create planes normal to the curve at these points, by checking the **Create normal planes also** button, and to create all instances in a new geometrical set by checking the **Create in a new geometrical set** button.
  If the button is not checked the instances are created in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



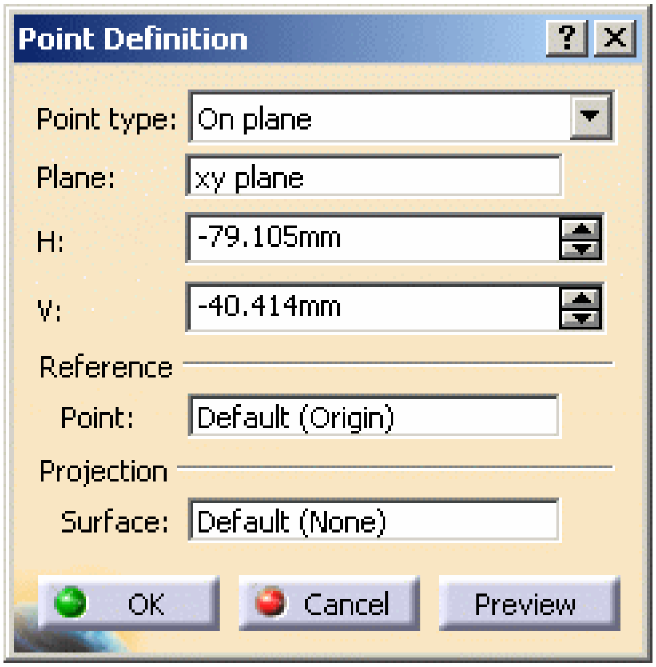
- If the curve is infinite and no reference point is explicitly given, by default, the reference point is the projection of the model's origin
- If the curve is a closed curve, either the system detects a vertex on the curve that can be used as a reference point, or it creates an extremum point, and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

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





On plane



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

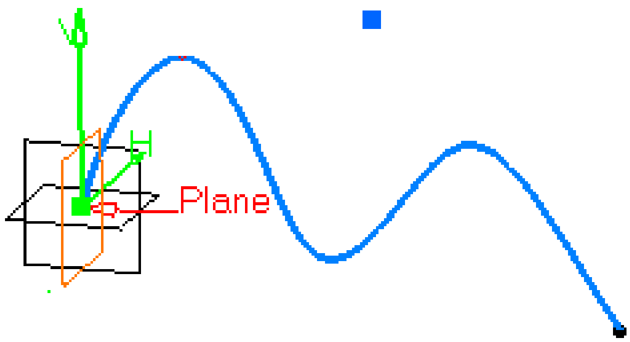
  If no point is selected, the projection of the model's origin on the plane is taken as reference.

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

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

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

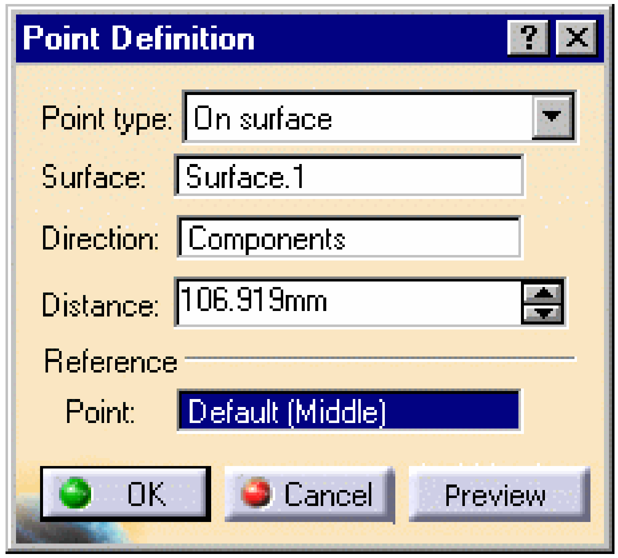
  Would the plane move, during an update for example, the reference direction would then be projected on the plane.



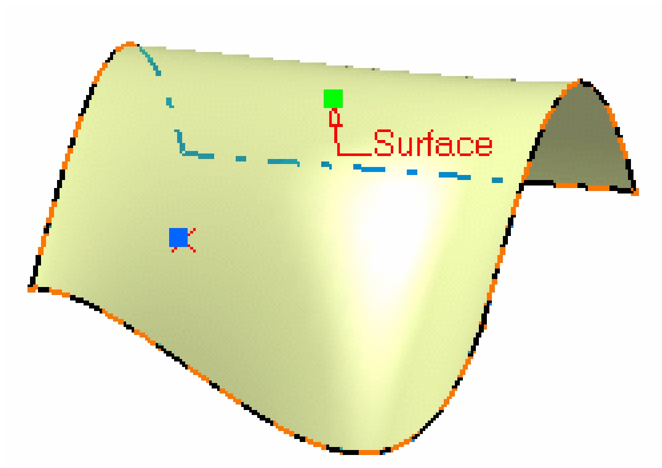
- Click in the plane to display a point.

On surface

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

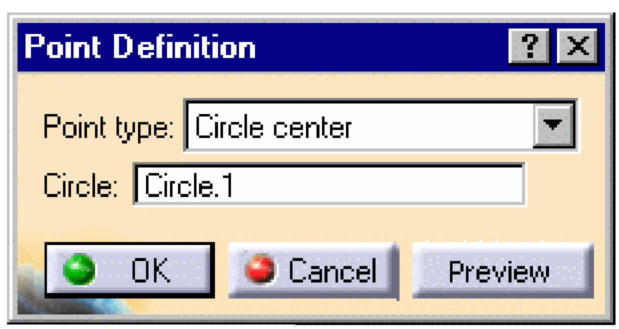


- Optionally, select a reference point. By default, the surface's middle point is taken as reference.
- You can select an element to take its orientation as reference direction or a plane to take its normal as reference direction.
  You can also use the contextual menu to specify the X, Y, Z components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

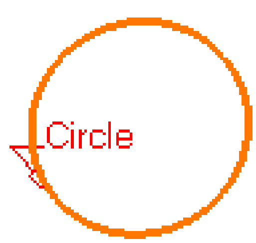


Circle center

- Select a circle, circular arc, or ellipse.

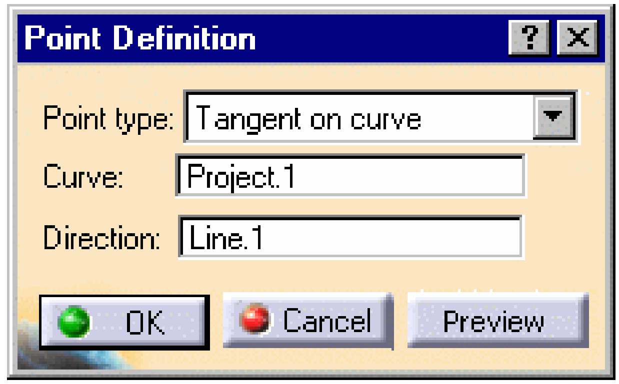


A point is displayed at the center of the selected element.



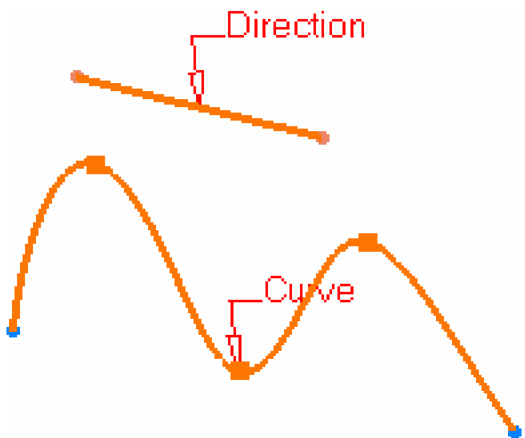
Tangent on curve

- Select a planar curve and a direction line.

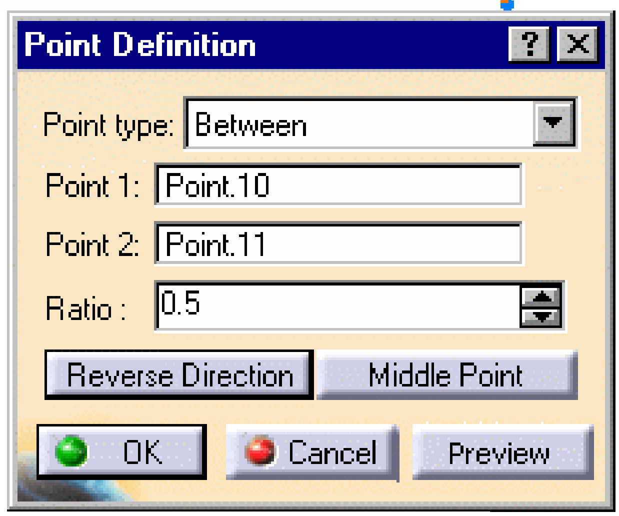


A point is displayed at each tangent.

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



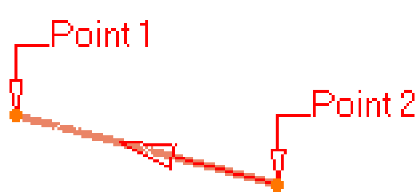
- Click **YES**: you can then select a reference element, to which only the closest point is created.
- Click **NO**: all the points are created.

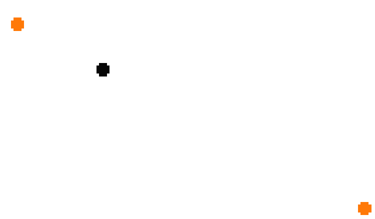


Between

- Select any two points.

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





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

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

3. Click OK to create the point.

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

Creating Lines

This task shows the various methods for creating lines:

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

It also shows you how to automatically reselect the second point.

Open the Lines1.CATPart document.

1. Click the **Line** icon .

   The Line Definition dialog box appears.

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

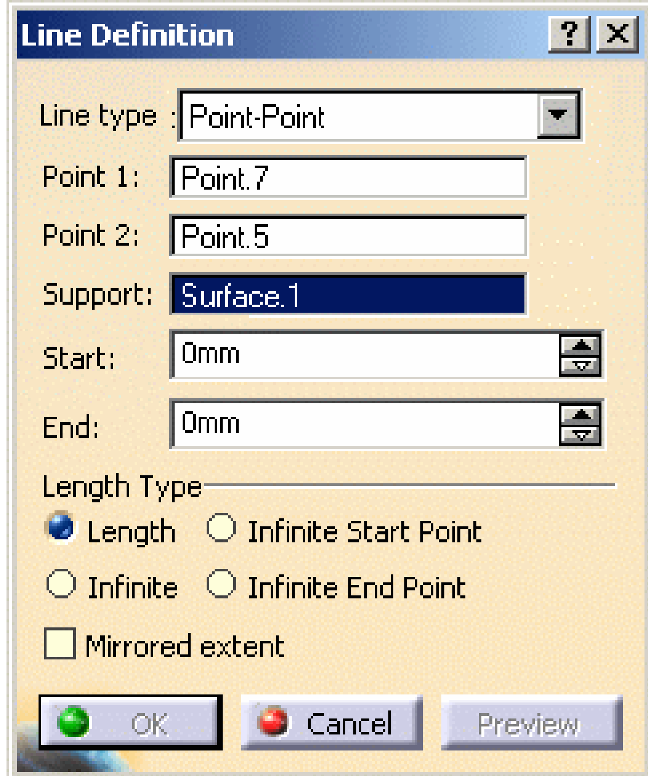
A line type will be proposed automatically in some cases depending on your first element selection.

Point - Point

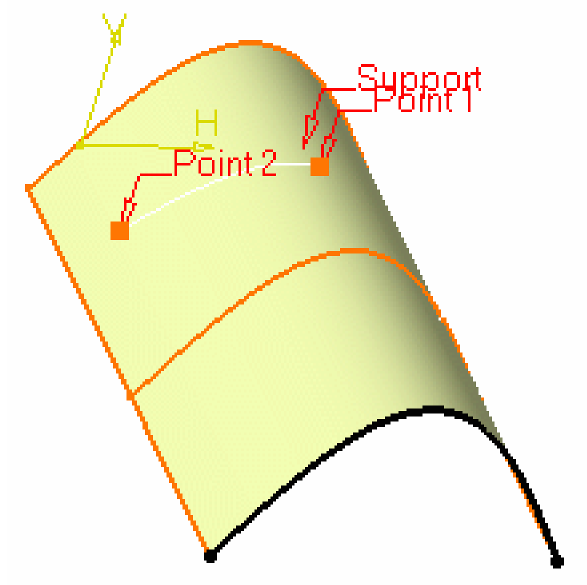
This command is only available with the Generative Shape Design 2 product.

- Select two points.

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

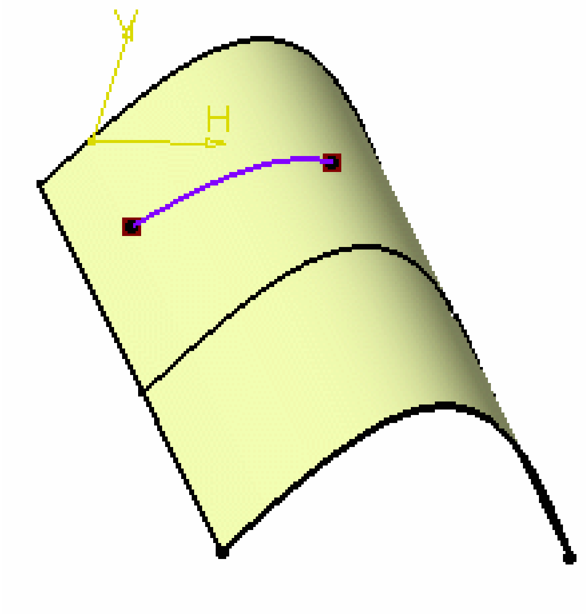


- If needed, select a support surface.
  In this case a geodesic line is created, i.e. going from one point to the other according to the shortest distance along the surface geometry (blue line in the illustration below).
  If no surface is selected, the line is created between the two points based on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



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

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

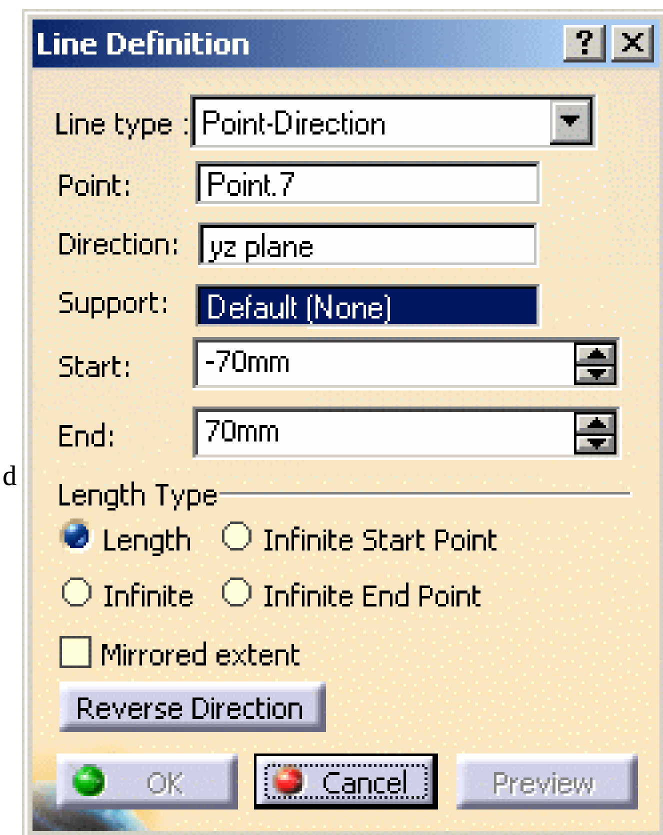


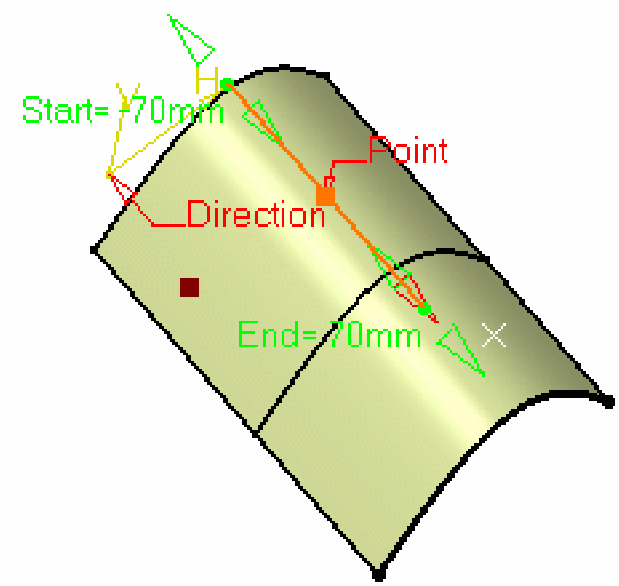
- Specify the **Start** and **End** points of the new line, that is the line endpoint location in relation to the points initially selected. These **Start** and **End** points are necessarily beyond the selected points, meaning the line cannot be shorter than the distance between the initial points.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

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

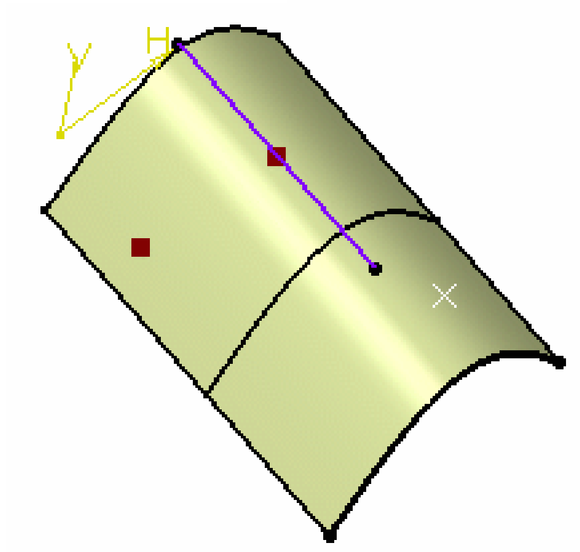
Point - Direction

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





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



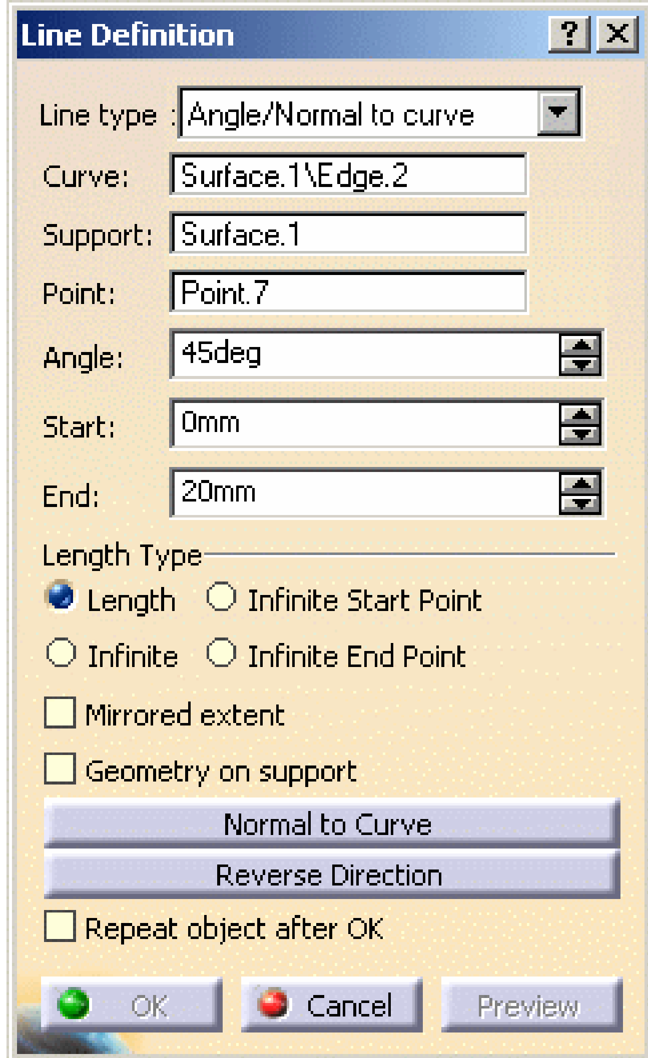
The projections of the 3D point(s) must already exist on the selected support.

Angle or Normal to curve

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

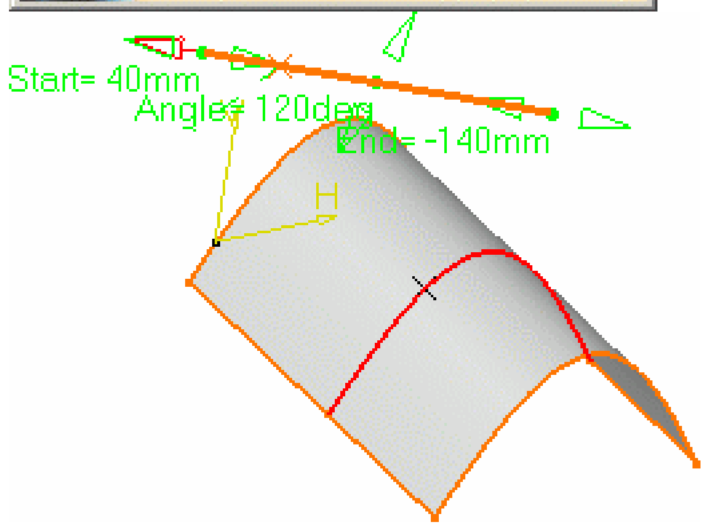
- If the selected curve is planar, then the **Support** is set to Default (Plane).

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



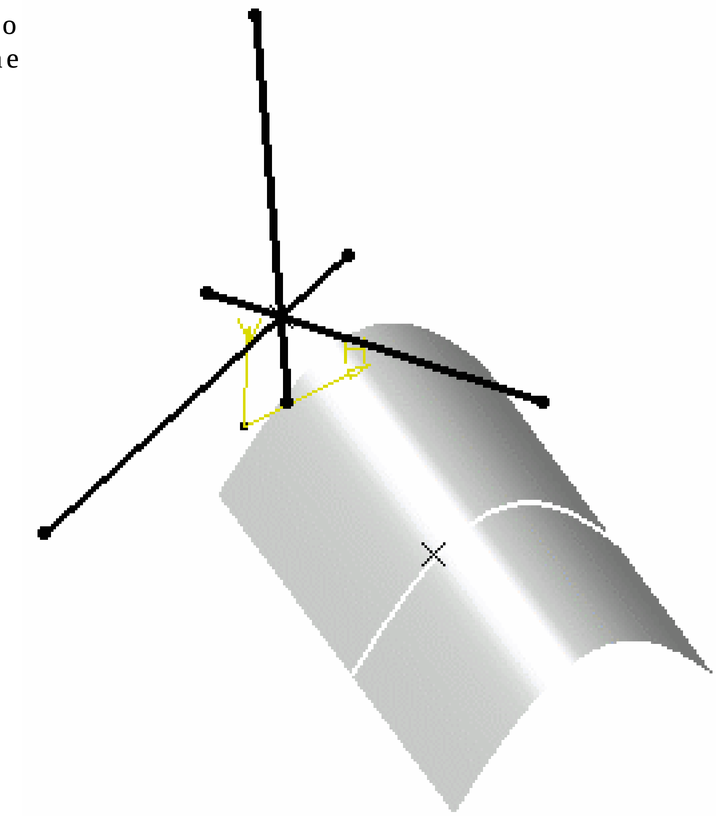
- Select a **Point** on the curve.
- Enter an **Angle** value.

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



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

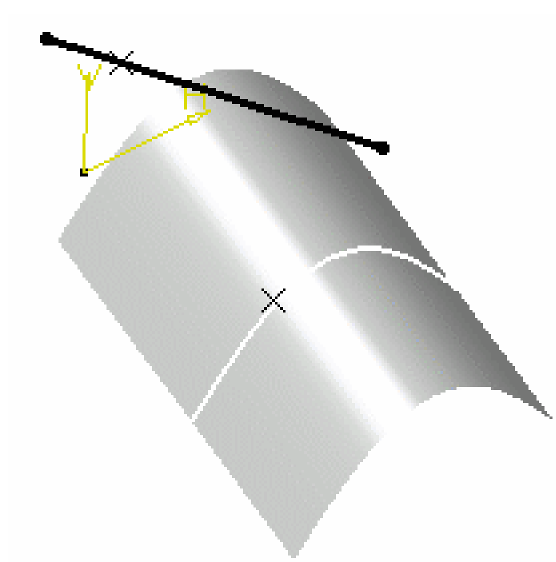
- Click the **Repeat object after OK** if you wish to create more lines with the same definition as the currently created line.
  In this case, the Object Repetition dialog box is displayed, and you key in the number of instances to be created before pressing OK.



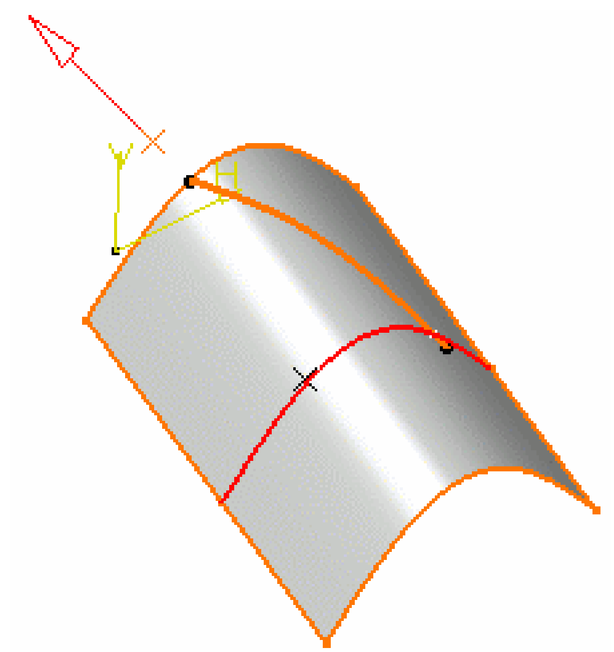
As many lines as indicated in the dialog box are created, each separated from the initial line by a multiple of the **angle** value.

You can select the **Geometry on Support** check box if you want to create a geodesic line onto a support surface.

The figure below illustrates this case.



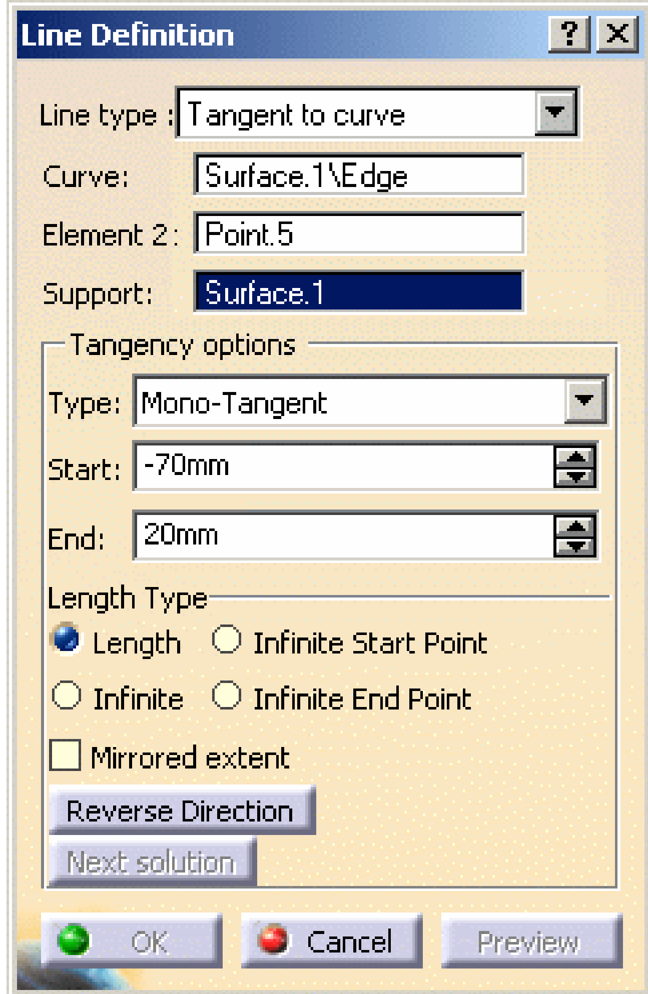
Geometry on support option not checked



Geometry on support option checked

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

Tangent to curve

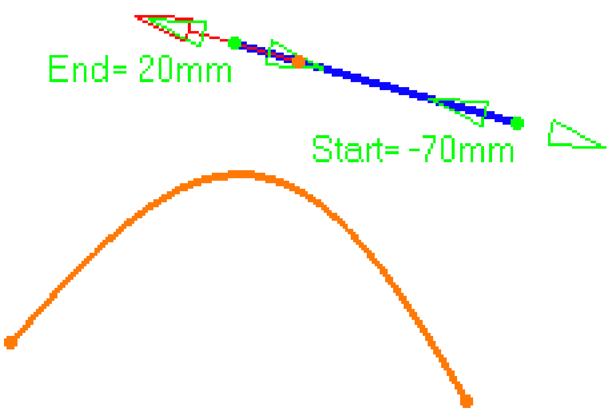


- Select a reference **Curve** and a **point** or another **Curve** to define the tangency.
  - if a point is selected (mono-tangent mode): a vector tangent to the curve is displayed at the selected point.
  - If a second curve is selected (or a point in bi-tangent mode), you need to select a support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

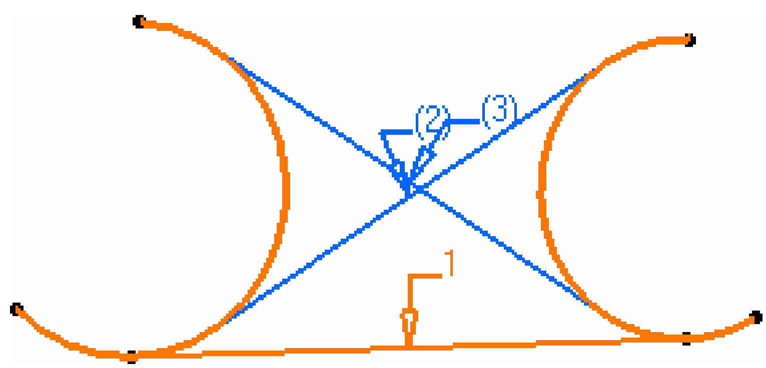
  - If the selected curve is a line, then the **Support** is set to Default (Plane).

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

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

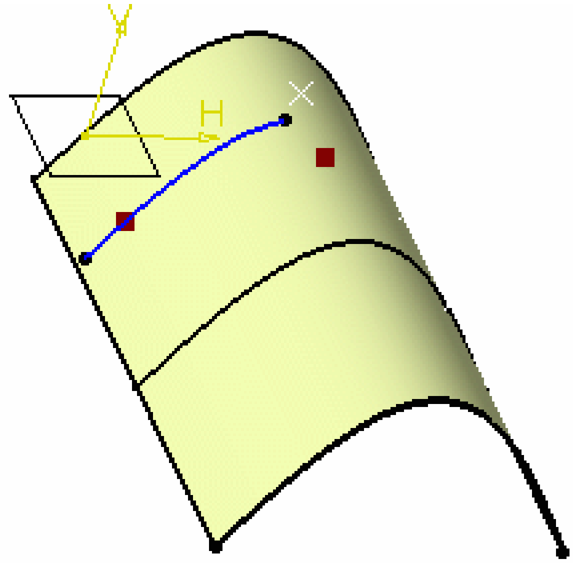


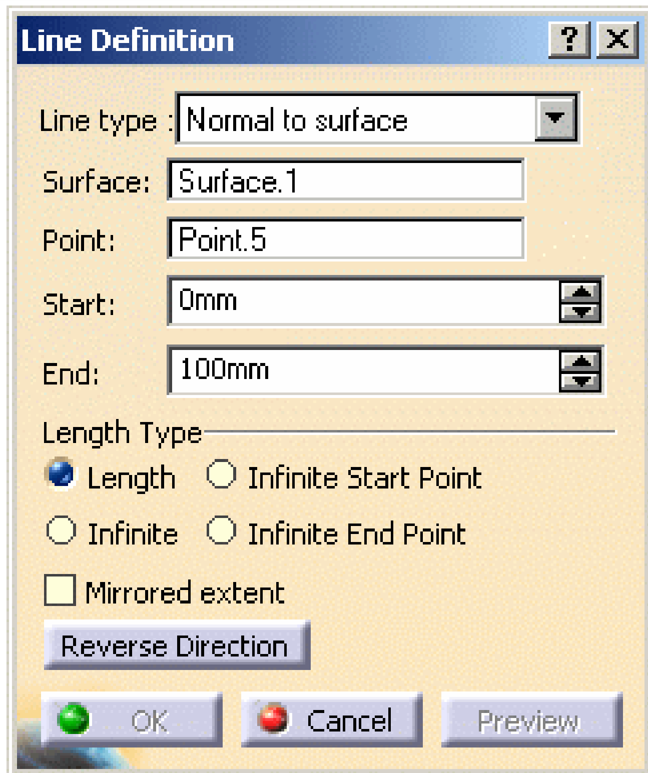
Line tangent to curve at a given point



Line tangent to two curves

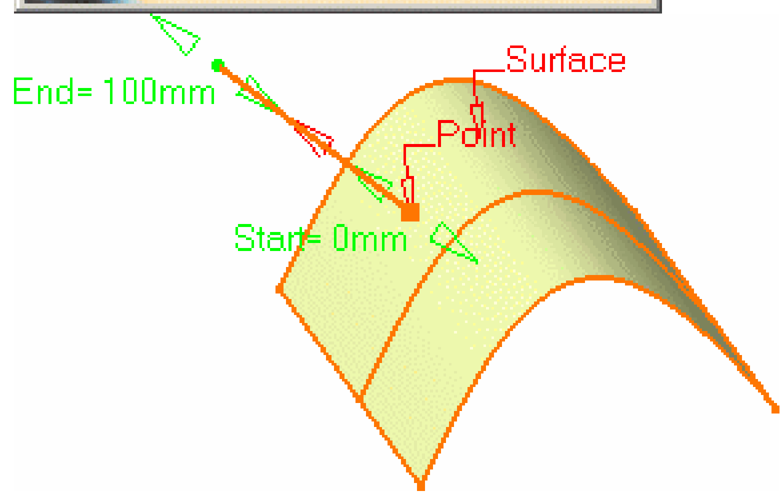
- Specify **Start** and **End** points to define the new line. The corresponding line is displayed.



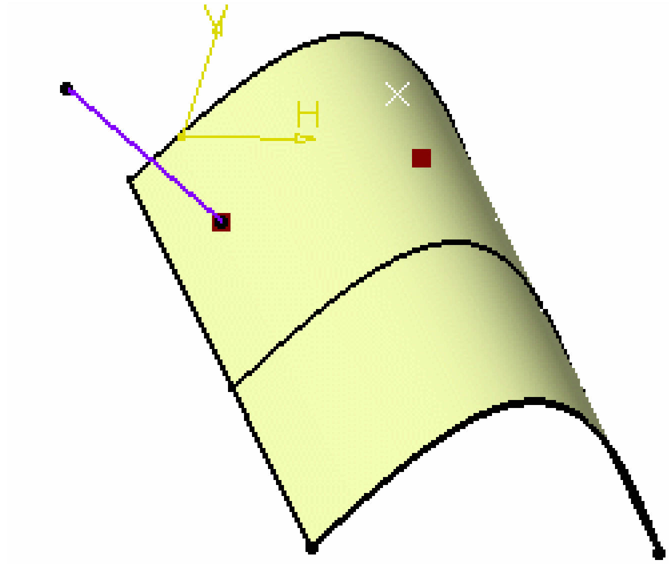


Normal to surface

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

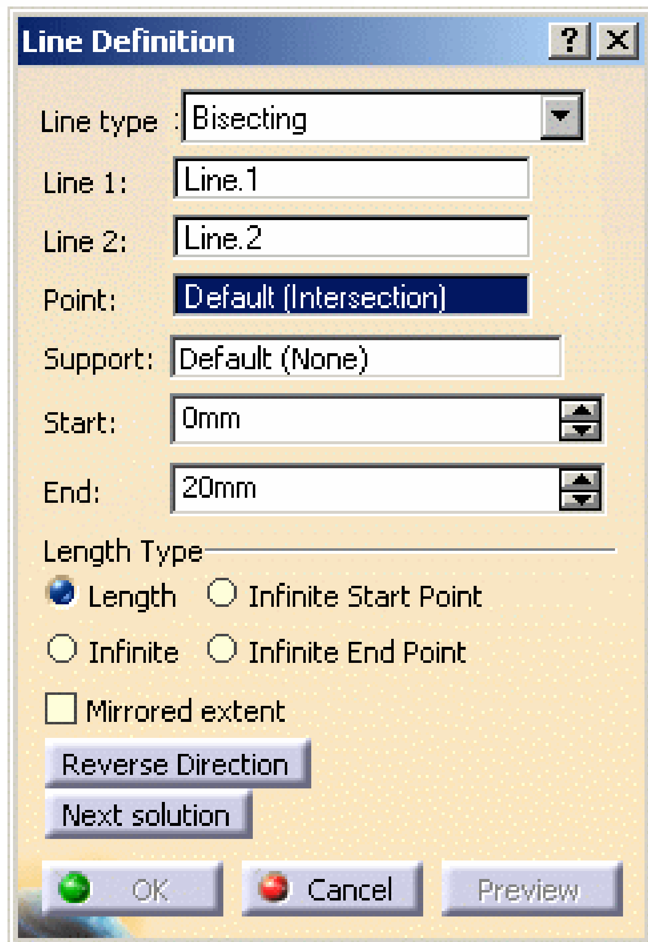


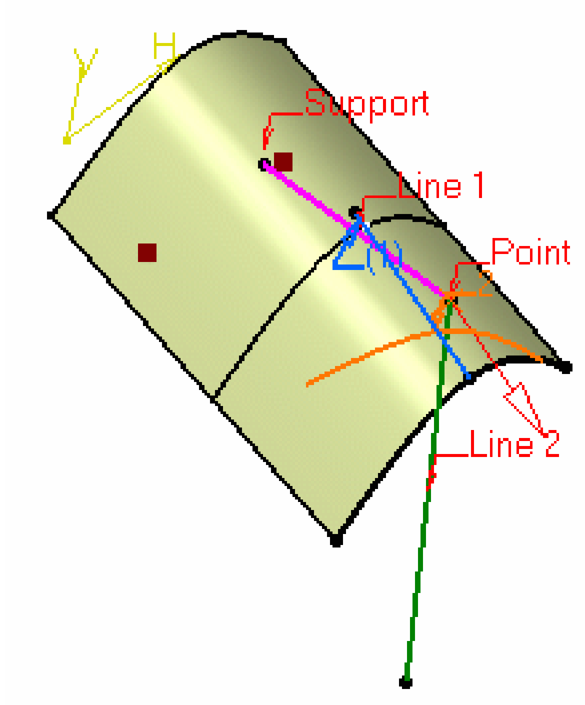
- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.



Bisecting

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





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

3. Click OK to create the line.

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

- Regardless of the line type, **Start** and **End** values are specified by entering distance values or by using the graphic manipulators.
- Start and End values should not be the same.

- Select the Length Type:
  - **Length**: the line will be defined according to the **Start** and **End** points values
  - **Infinite**: the line will be infinite
  - **Infinite Start Point**: the line will be infinite from the **Start** point
  - **Infinite End Point**: the line will be infinite from the **End** point

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

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

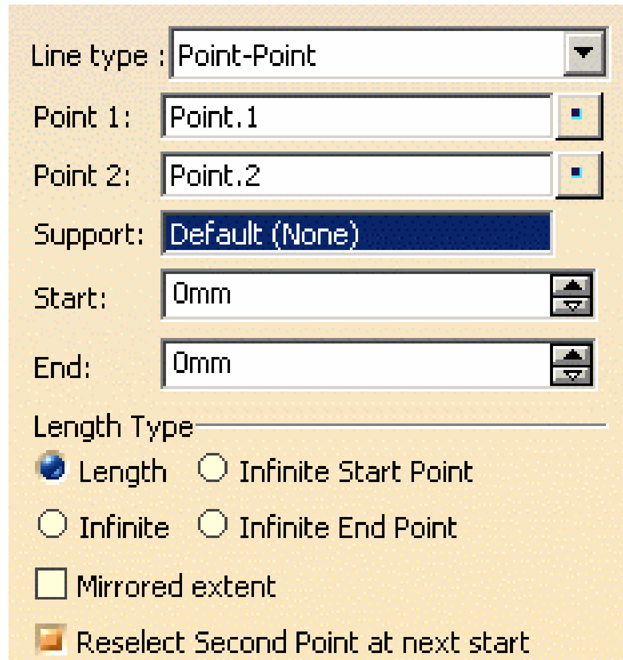
Automatic Reselection

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

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

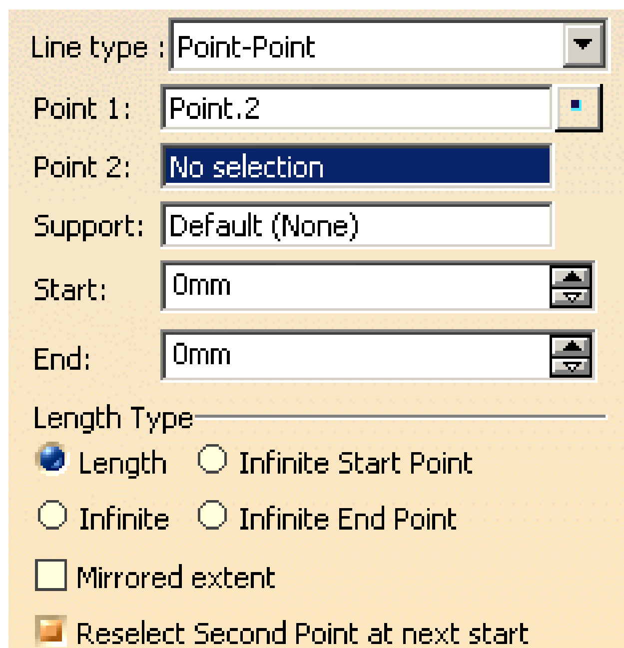
   The Line dialog box is displayed.

2. Create the first point.



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

3. Check it to be able to later reuse the second point.
4. Create the second point.
5. Click OK to create the first line.



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

6. Click OK to create the second point.
7. Click OK to create the second line, and so on.

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

Creating Planes

This task shows the various methods for creating planes:

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

Open the Planes1.CATPart document.

1. Click the **Plane** icon .

   The Plane Definition dialog box appears.

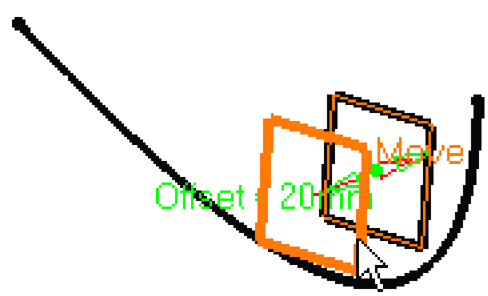
2. Use the combo to choose the desired **Plane type**.

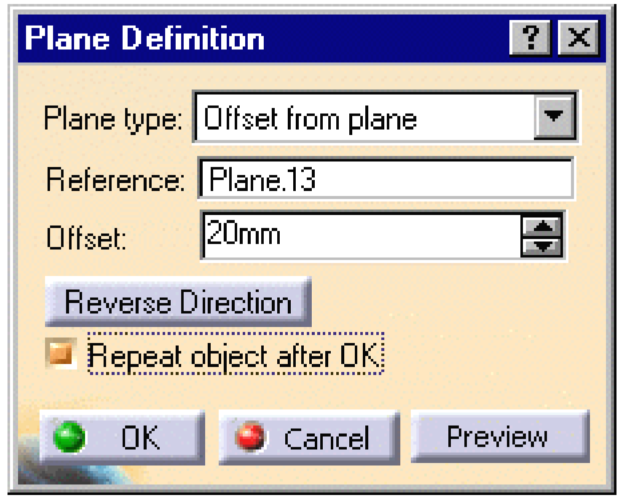
Once you have defined the plane, it is represented by a red square symbol, which you can move using the graphic manipulator.

Offset from plane

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

A plane is displayed offset from the reference plane.

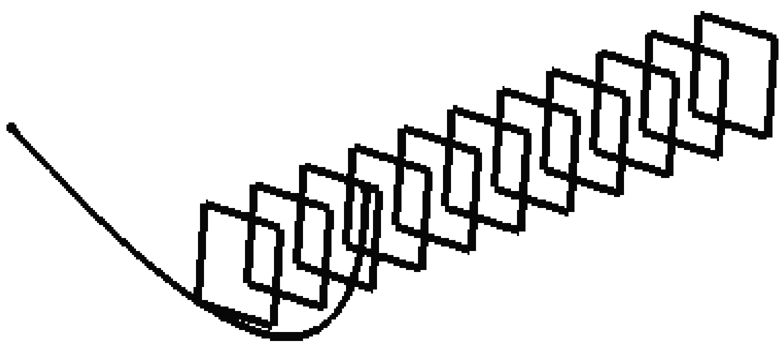




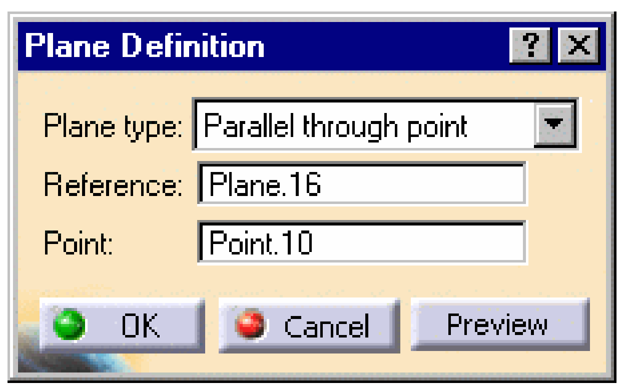
Use the **Reverse Direction** button to reverse the change the offset direction, or simply click on the arrow in the geometry.

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

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

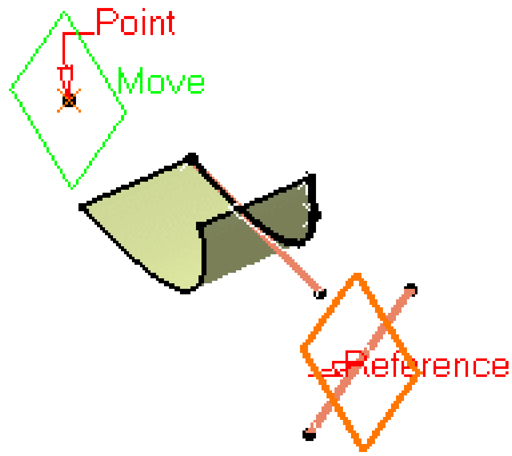


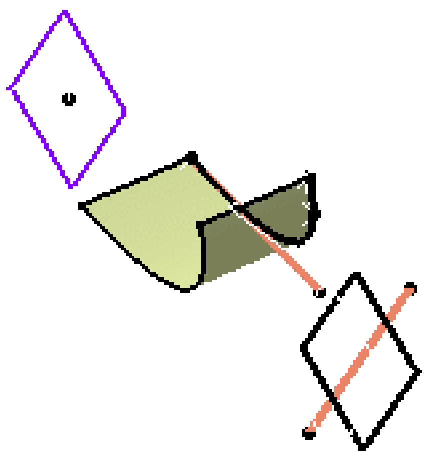
Parallel through point



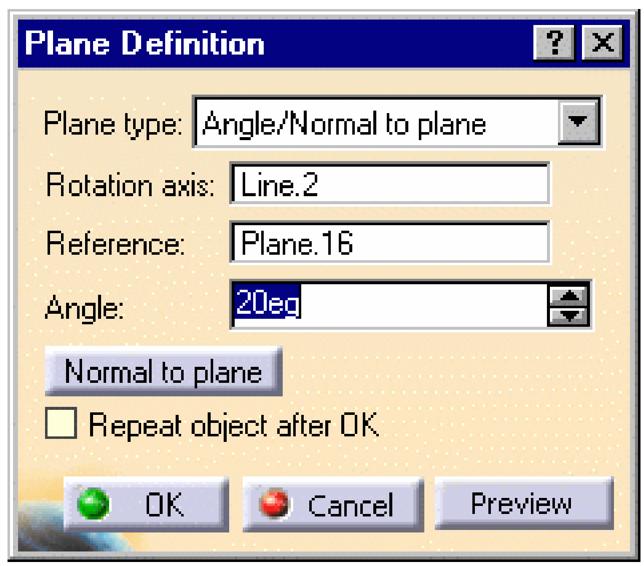
- Select a reference **Plane** and a **Point**.

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



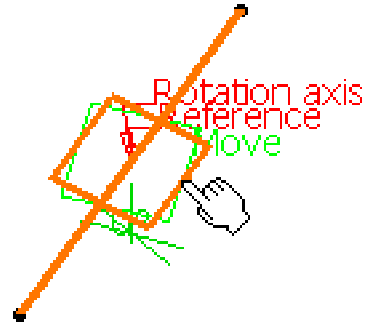


Angle or normal to plane



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

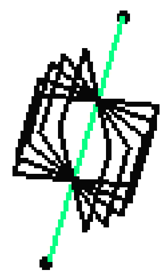
A plane is displayed passing through the rotation axis. It is oriented at the specified angle to the reference plane.



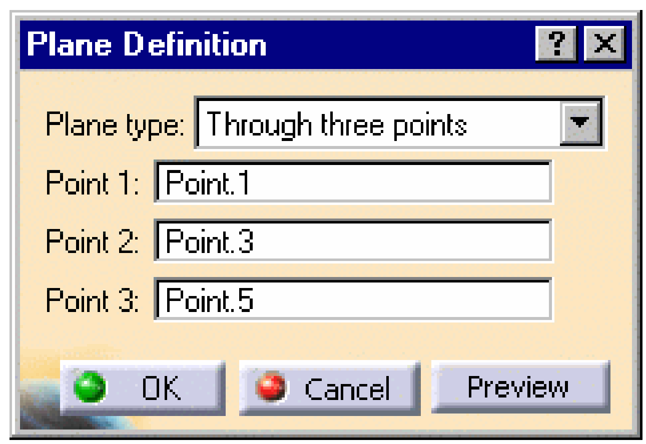
- Click the **Repeat object after OK** if you wish to create more planes at an angle from the initial plane.
  In this case, the **Object Repetition** dialog box is displayed, and you key in the number of instances to be created before pressing OK.

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

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

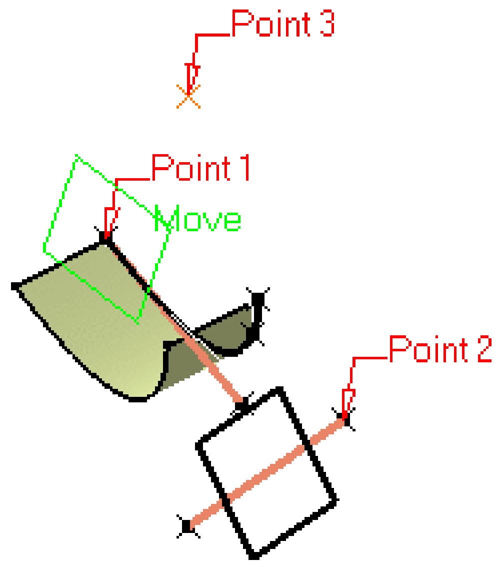


This plane type enables to edit the plane's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

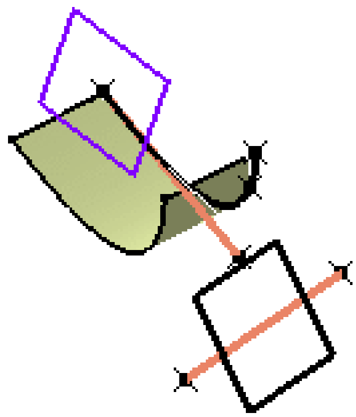


Through three points

- Select three points.

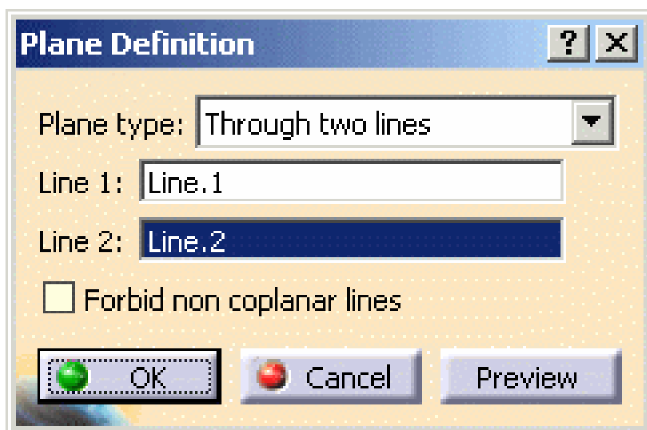


The plane passing through the three points is displayed. You can move it simply by dragging it to the desired location.

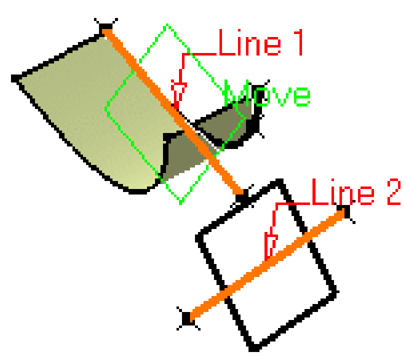


Through two lines

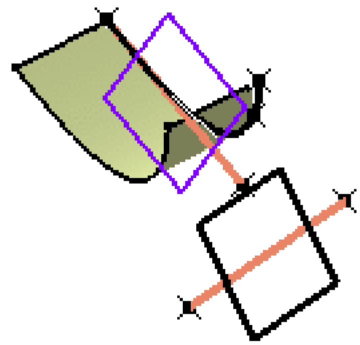
- Select two lines.



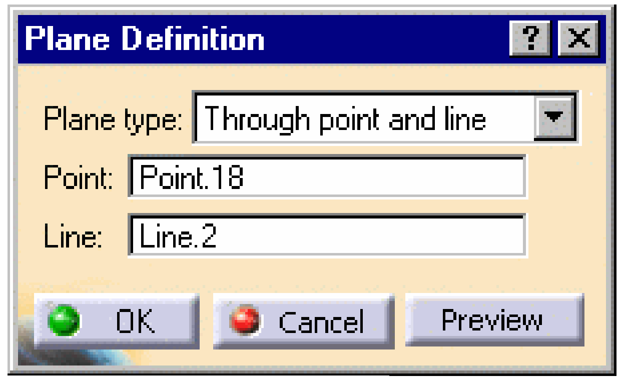
The plane passing through the two line directions is displayed.
When these two lines are not coplanar, the vector of the second line is moved to the first line location to define the plane's second direction.



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

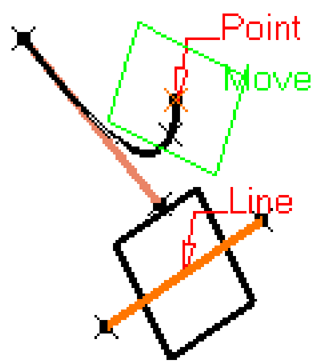


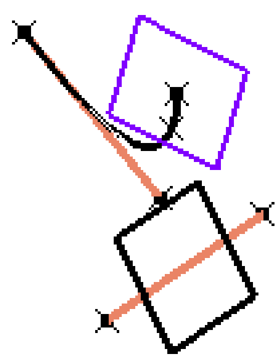
Through point and line



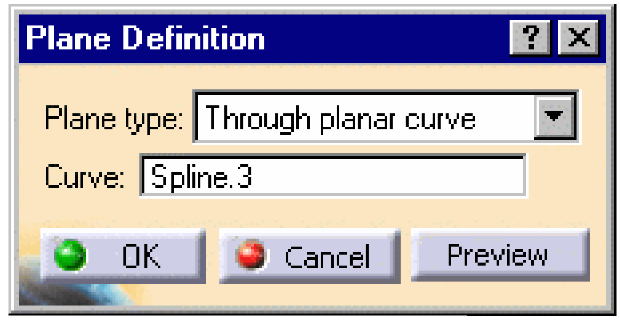
- Select a **Point** and a **Line**.

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

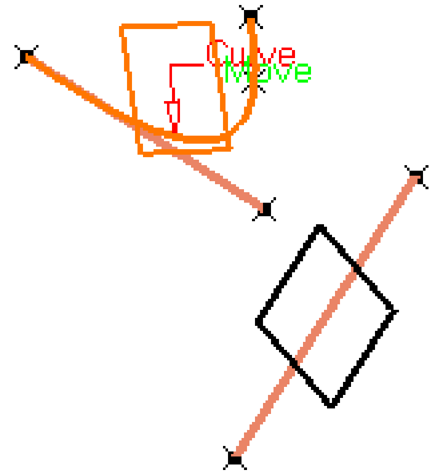




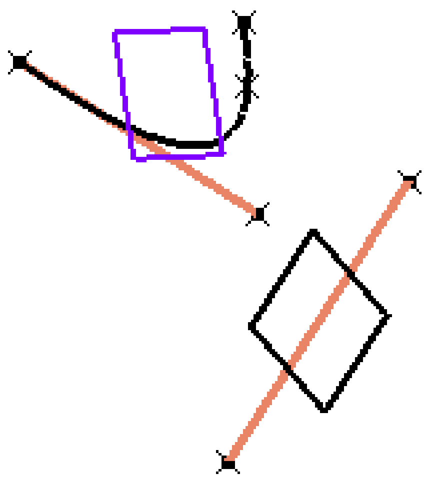
Through planar curve



- Select a planar **Curve**.

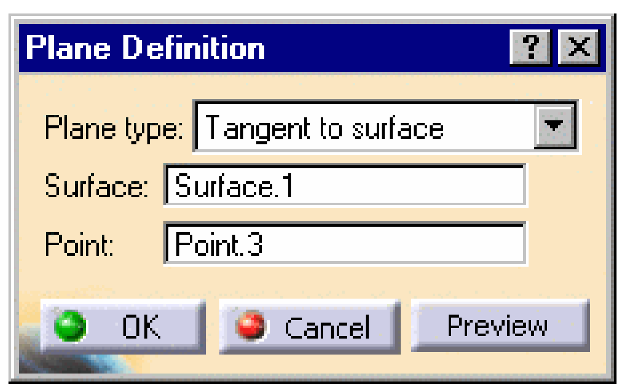


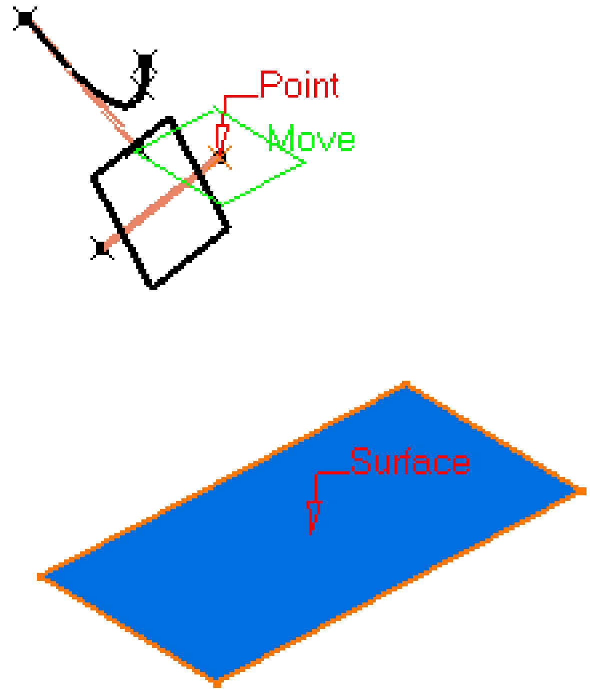
The plane containing the curve is displayed.



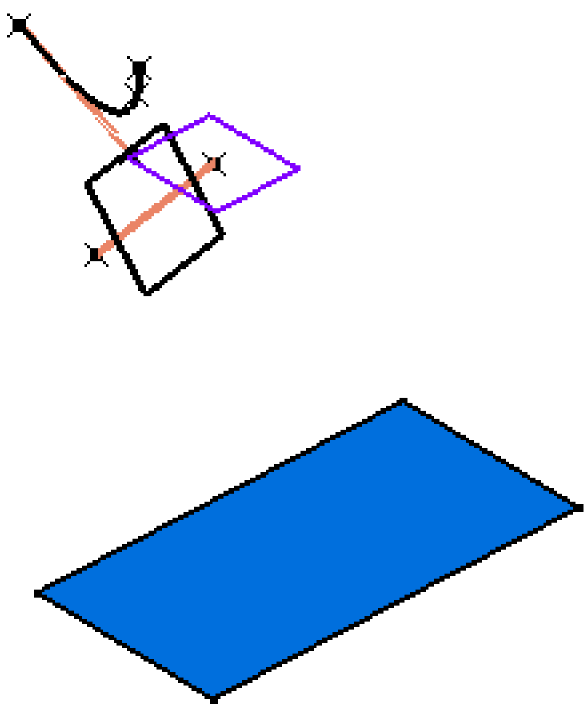
Tangent to surface

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



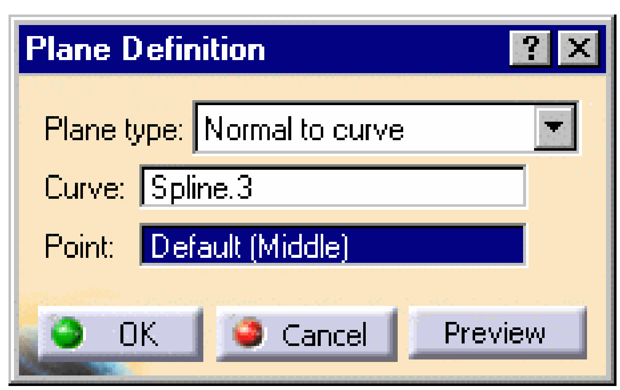


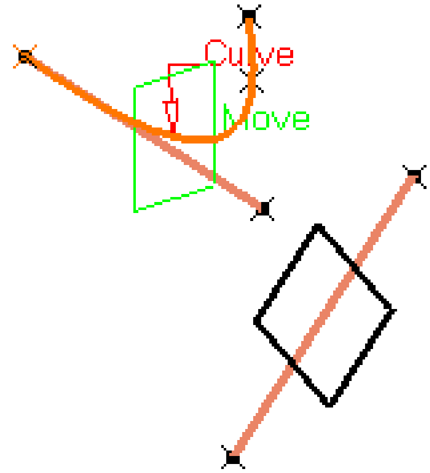
A plane is displayed tangent to the surface at the specified point.



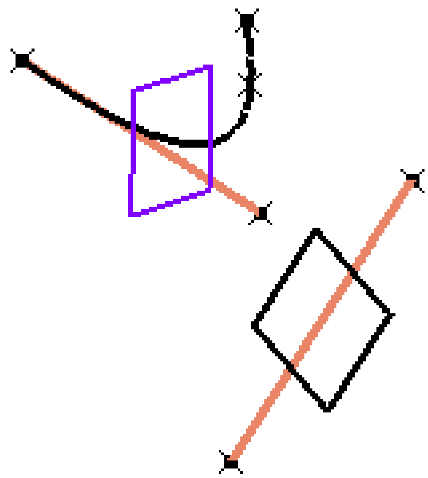
Normal to curve

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

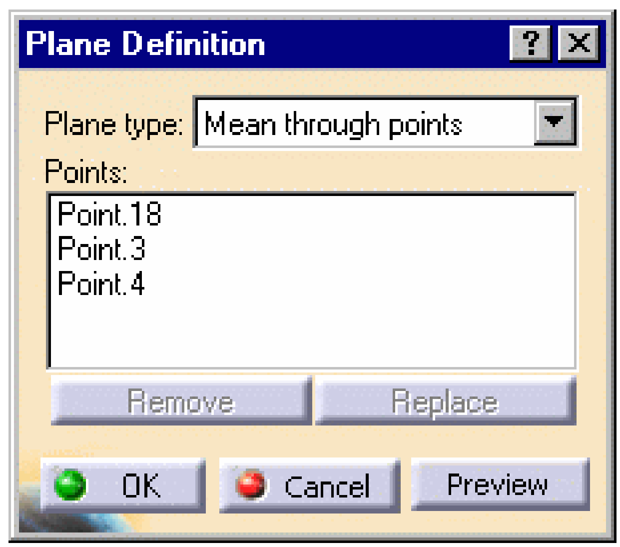




A plane is displayed normal to the curve at the specified point.

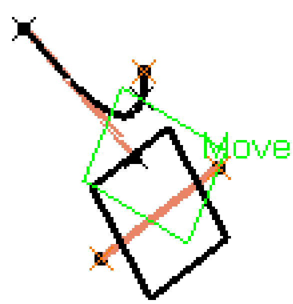


Mean through points

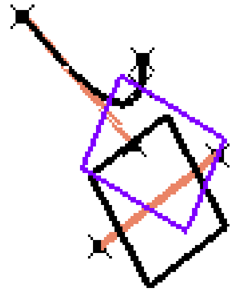


- Select three or more points to display the mean plane through these points.

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

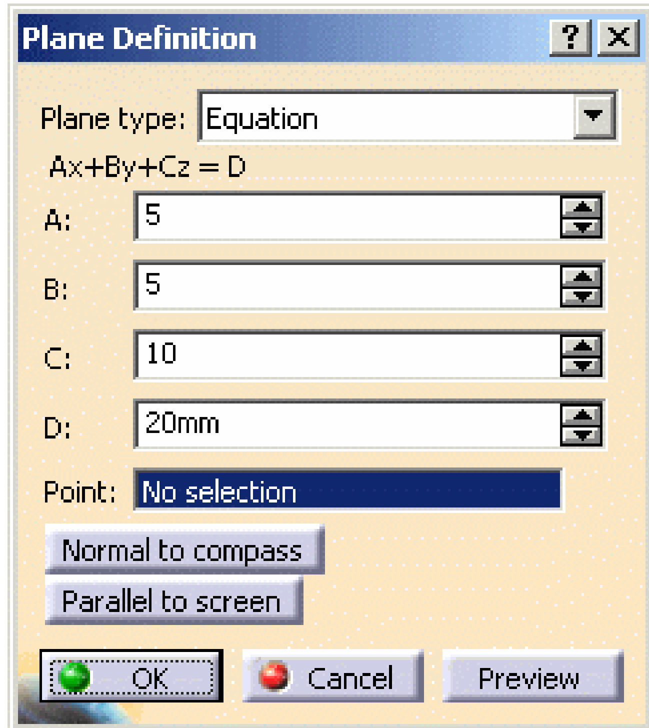


- **Remove** the selected point
- **Replace** the selected point by another point.

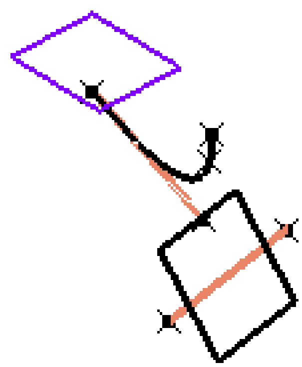


Equation

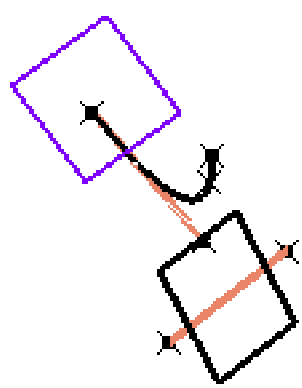
- Enter the **A**, **B**, **C**, **D** components of the **Ax + By + Cz = D** plane equation.



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



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



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

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

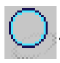
The plane (identified as Plane.xxx) is added to the specification tree.

Creating Circles

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

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

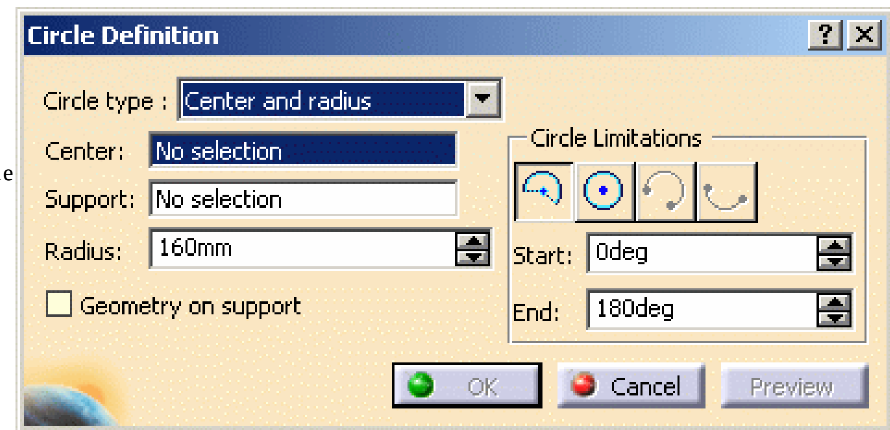
Open the Circles1.CATPart document.
Please note that you need to put the desired geometrical set in show to be able to perform the corresponding scenario.

1. Click the **Circle** icon .

   The Circle Definition dialog box appears.

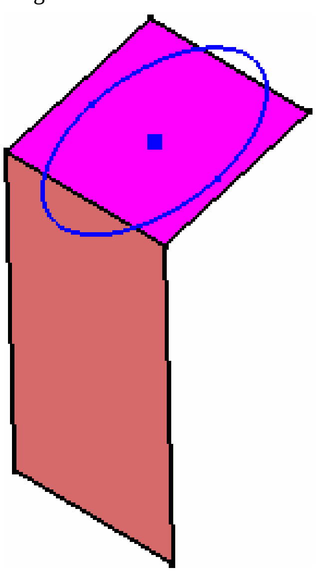
2. Use the combo to choose the desired circle type.

Center and radius



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

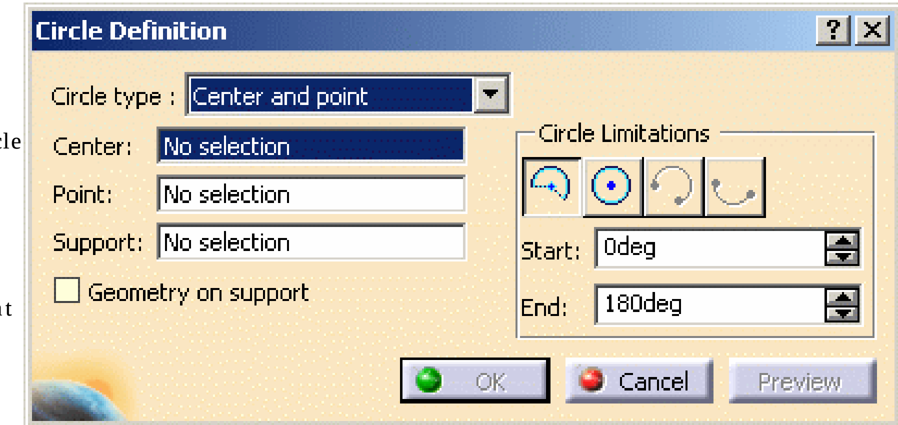
Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the **Start** and **End** angles of the arc.



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

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

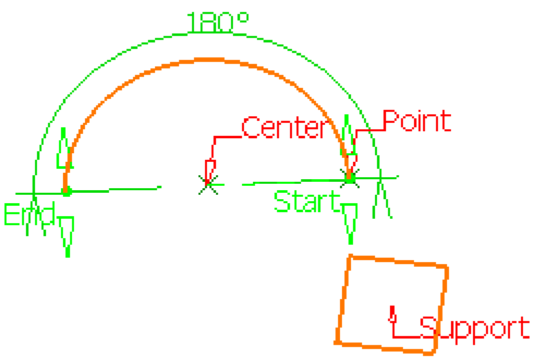
Center and point



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

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

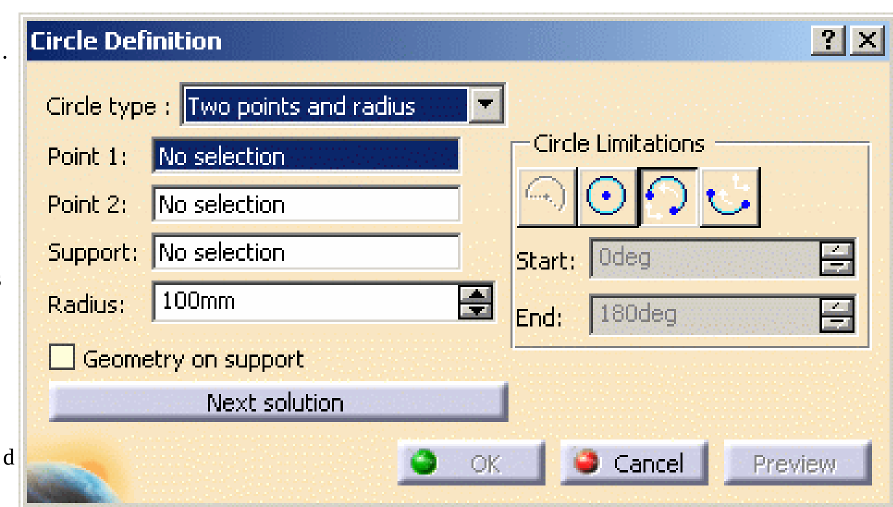
Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the **Start** and **End** angles of the arc.



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.

Two points and radius

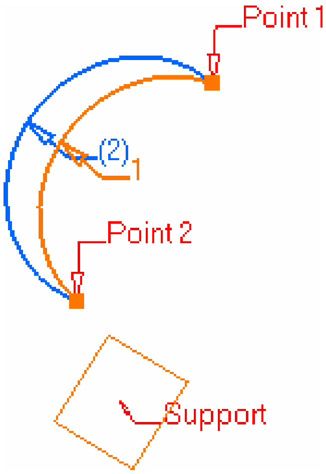


- Select two points on a surface or in the same plane.
- Select the **Support** plane or surface.
- Enter a **Radius** value.

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

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

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

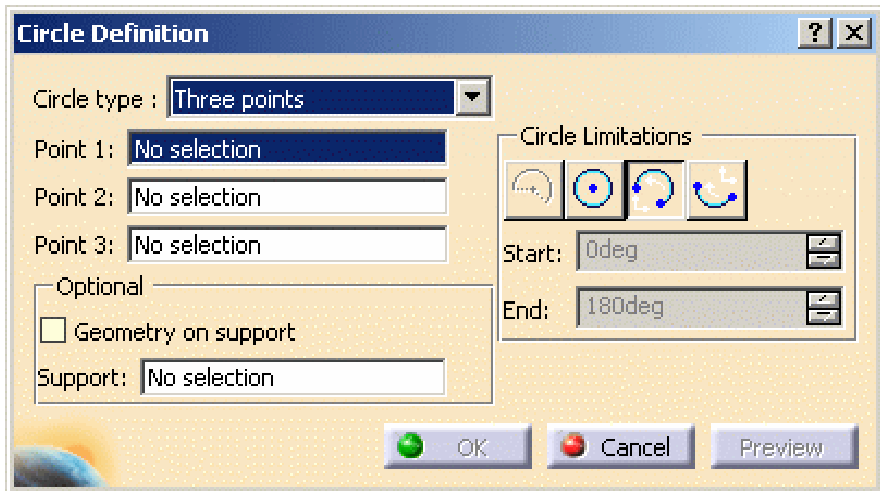


You can select the **Geometry on Support** check box if you want the circle to be projected onto a support surface.

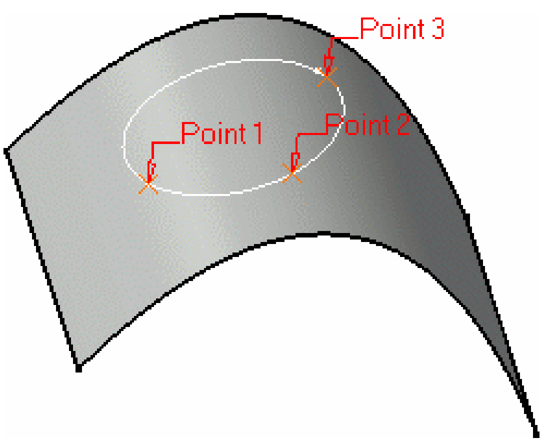
In this case just select a support surface.

Three points

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



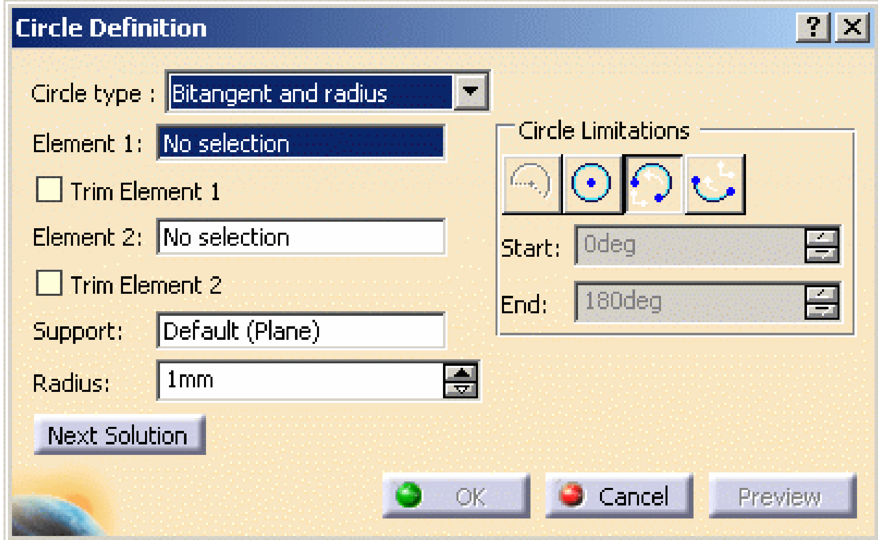
Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the trimmed or complementary arc using the two of the selected points as end points.



You can select the **Geometry on Support** check box if you want the circle to be projected onto a support surface.

In this case just select a support surface.

Bi-tangent and radius



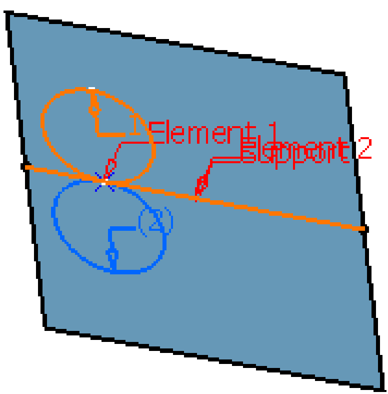
- Select two **Elements** (point or curve) to which the circle is to be tangent.
- Select a **Support** surface.

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

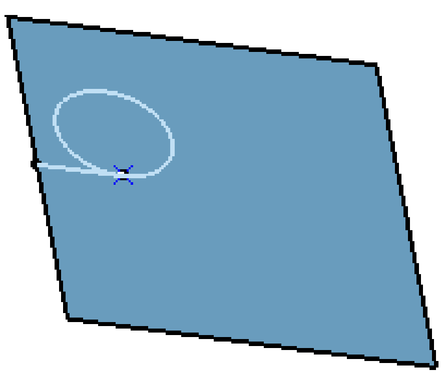
This automatic support definition saves you from performing useless selections.

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

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

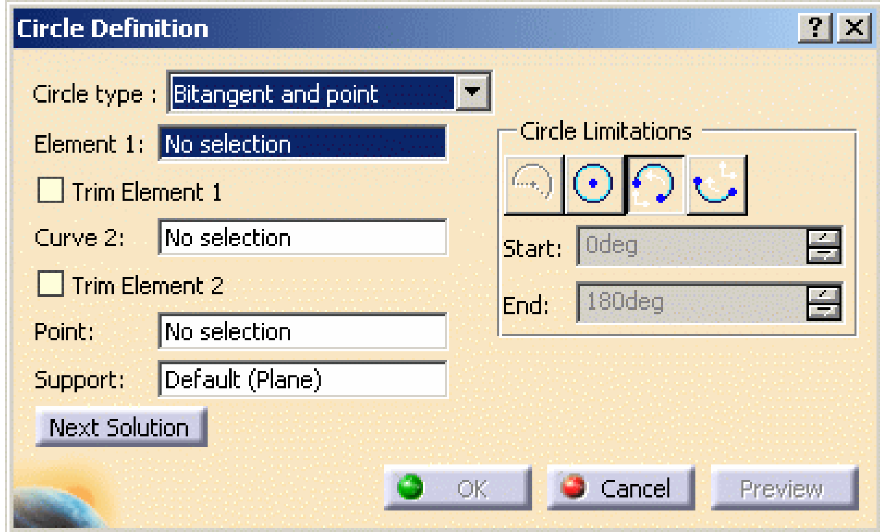


You can select the **Trim Element 1** and **Trim Element 2** check boxes to trim the first element or the second element, or both elements.
Here is an example with Element 1 trimmed.



These options are only available with the Trimmed Circle limitation.

Bi-tangent and point



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

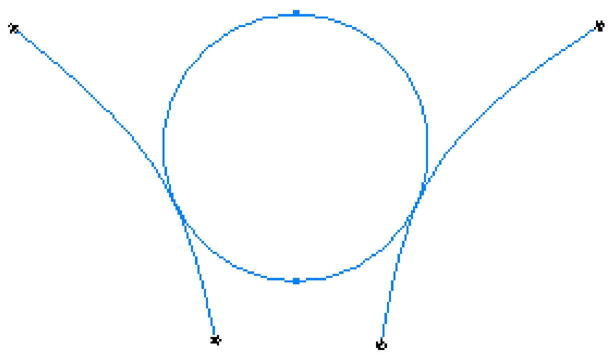
The point will be projected onto the curve.

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

This automatic support definition saves you from performing useless selections.

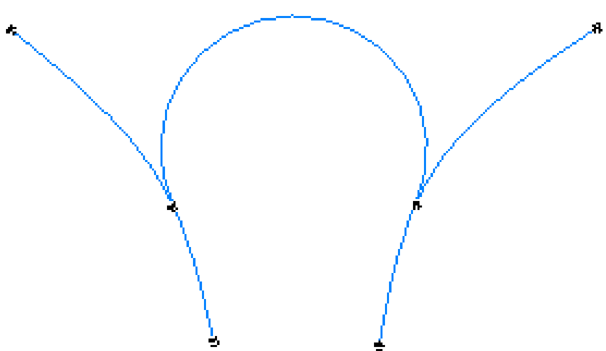
- Several solutions may be possible, so click in the region where you want the circle to be.

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

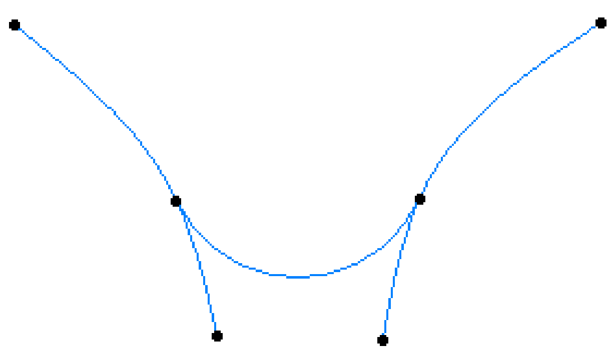


Complete circle

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



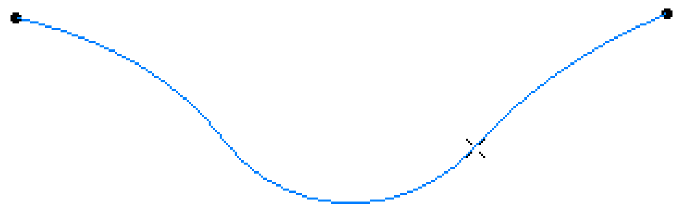
Trimmed circle



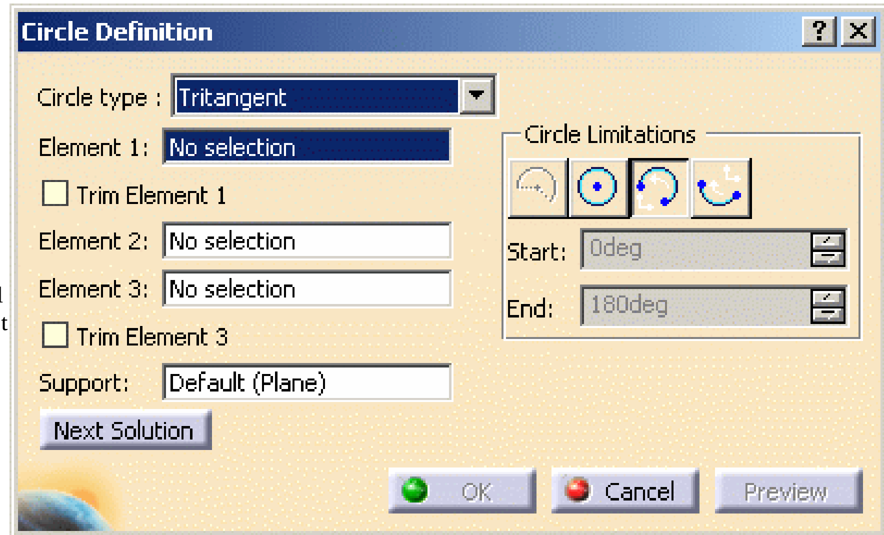
Complementary trimmed circle

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

These options are only available with the Trimmed Circle limitation.



Tritangent



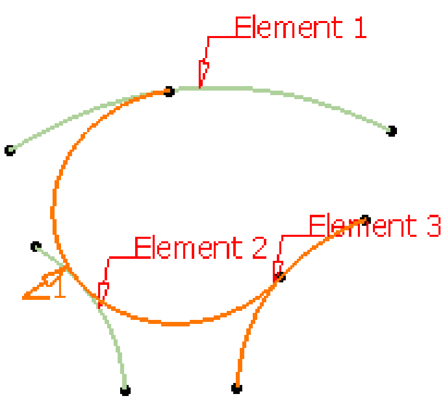
- Select three **Elements** to which the circle is to be tangent.
- Select a **Support** planar surface.

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

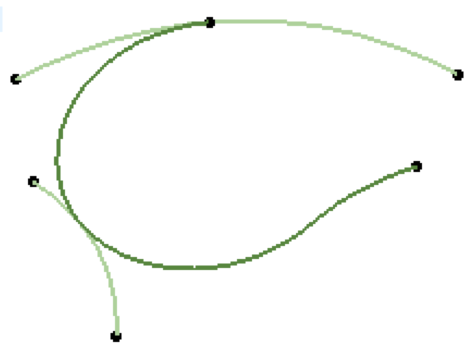
This automatic support definition saves you from performing useless selections.

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

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



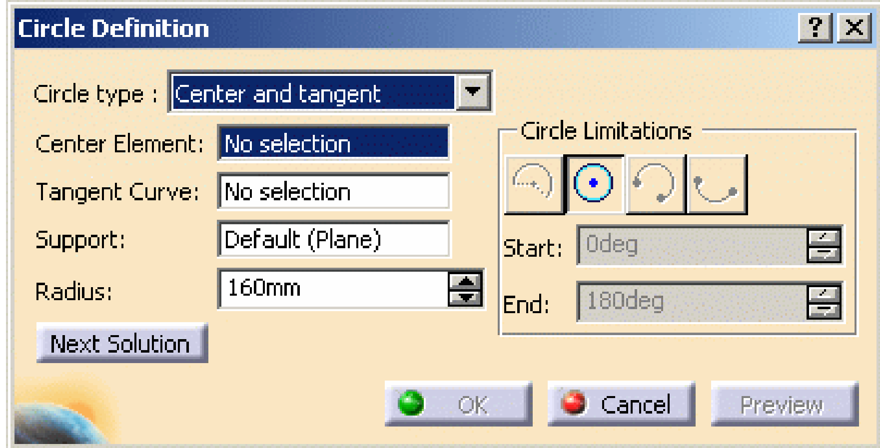
You can select the **Trim Element 1** and **Trim Element 3** check boxes to trim the first element or the third element, or both elements.
Here is an example with Element 3 trimmed.



These options are only available with the Trimmed Circle limitation.

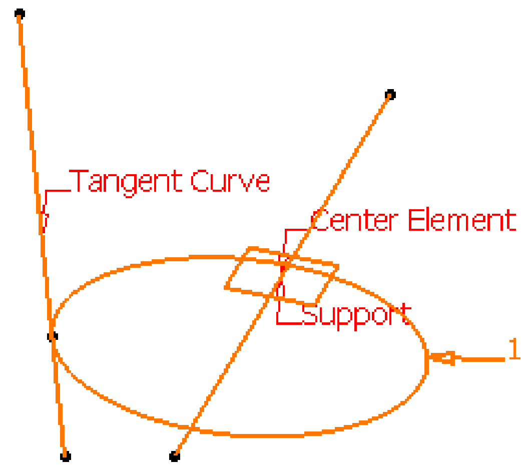
Center and tangent

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



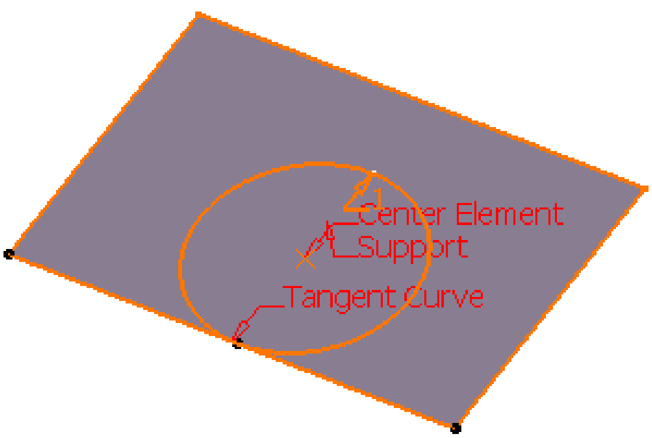
1. Center curve and radius

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



2. Line tangent to curve definition

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



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

  This automatic support definition saves you from performing useless selections.
- The circle center will be located either on the center curve or point and will be tangent to tangent curve.
- Please note that only full circles can be created.

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

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

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

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

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

Interoperability

Quick Surface Reconstruction complies with the following CATIA V5 standards:

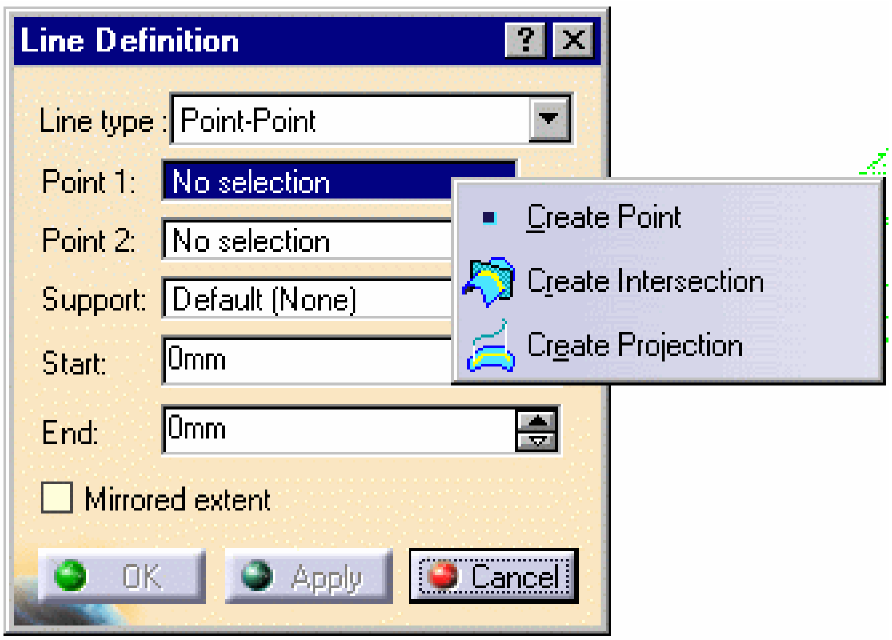
Points in Generative Shape Design
Updating Your Design
Using the Historical Graph
Creating Datums

Interoperability

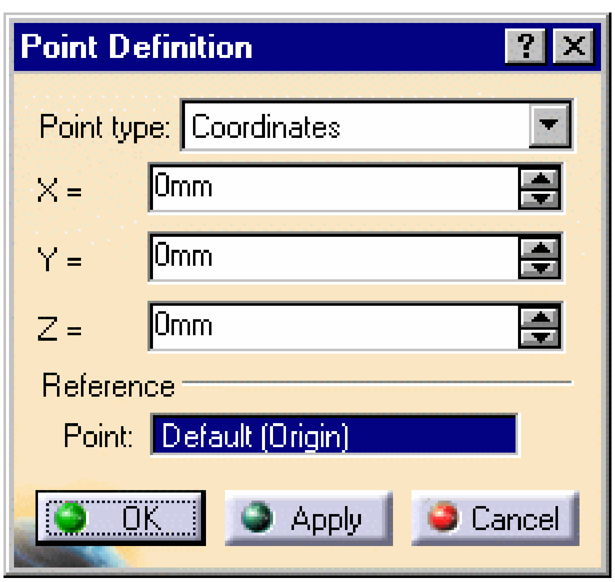
This task shows you how to use points from a cloud of points in Generative Shape Design.

Open the Interoperability.CATPart from the samples directory. We used the Line Definition for our example, but the operating mode is the same for all creation action requiring points.

1. Select the type of creation from the combo list of the dialog box.
2. Go to the next Point field and choose **Create Point** from the contextual menu.



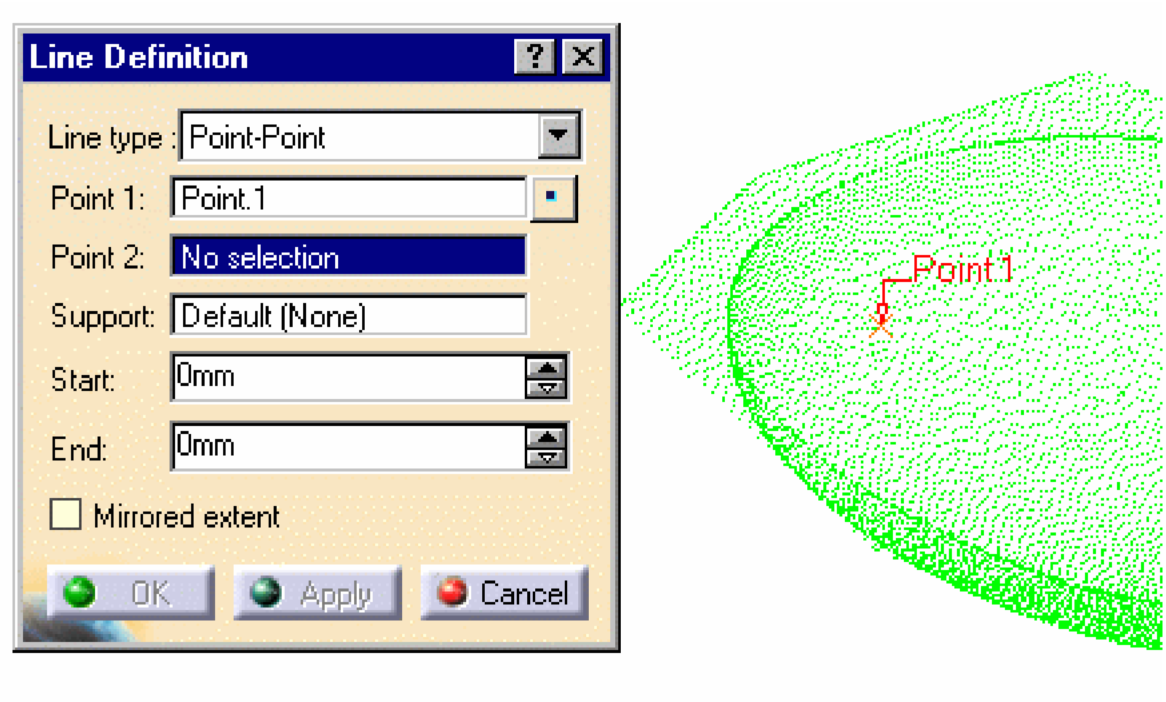
3. The **Point Definition** dialog box is displayed.



4. Click a point on the cloud. Its coordinates are displayed in the **Point Definition** dialog box. Click **OK** to confirm the creation of this point.

5. In the main dialog box, go to the next Point field and repeat the above steps as many times as necessary.

6. If necessary, push the Point icon on the right of the Point field to modify the point you have created: the **Point Definition** dialog box is displayed and updated according to your pick on the cloud.



Updating Your Design

This task explains how and when you should update your design.

The point of updating your design is to make the application take your last operation into account. Indeed some changes to geometry or a constraint may require rebuilding the part. To warn you that an update is needed, CATIA displays the update symbol next to the part name and displays the corresponding geometry in bright red.

To update a part, the application provides two update modes:

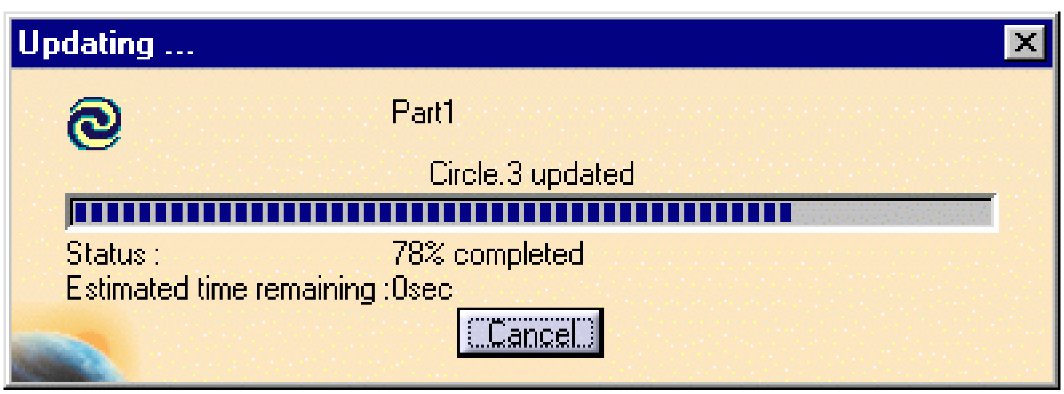
- **automatic update**, available in **Tools -> Options -> Mechanical Design -> Assembly Design -> General** tab. If checked, this option lets the application update the part when needed.

- **manual update**, available in **Tools -> Options -> Mechanical Design -> Assembly Design -> General** tab, it lets you control the updates of your part. You simply need to click the **Update** icon whenever you wish to integrate modifications.

Non-updated wireframe and surface elements are displayed in red.

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

   A progression bar indicates the evolution of the operation.



You can cancel the undergoing update by clicking the Cancel button available in the Updating... dialog box.

- Keep in mind that some operations such as confirming the creation of features (clicking **OK**) do not require you to use the update command. By default, the application automatically updates the operation.

- The Update capability is also available via **Edit -> Update** and the **Update** contextual command.

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

- Besides the update modes, you can also choose to visualize the update on the geometry as it is happening by checking the **Activate Local Visualization** option from the **Tools -> Options -> Infrastructure -> Part Infrastructure**, **General** tab.

  In this case, as soon as you have clicked the **Update** icon :

  1. the geometry disappears from the screen
  2. each element is displayed as it is updated, including elements in No Show mode. Once they have been updated, they remain in No Show mode.

Interrupting Updates

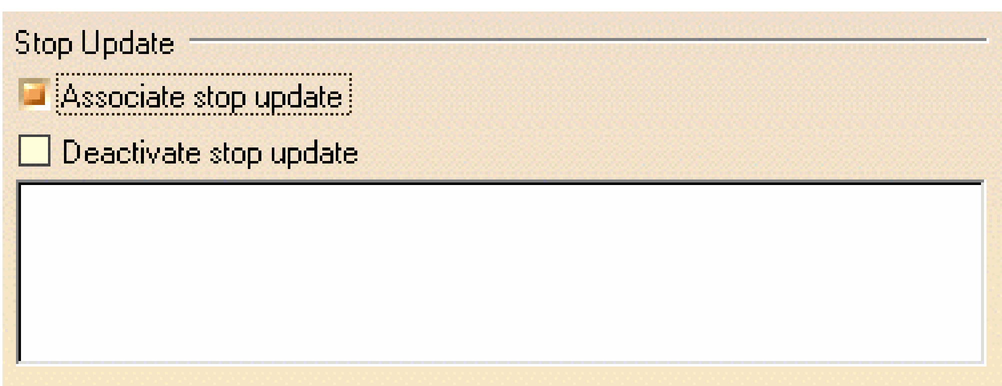
This task explains how to update a part and interrupt the update operation on a given feature by means of a useful message you previously defined.

Open any document containing geometric elements.

1. Right-click an element from the specification tree and choose the Properties contextual menu item.

   The Properties dialog box is displayed.

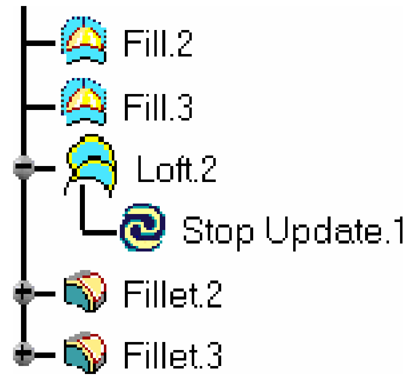
2. From the **Mechanical** tab, check the **Associate stop update** option.

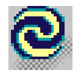


3. Enter the text to be displayed when the updating process will stop when reaching this element.

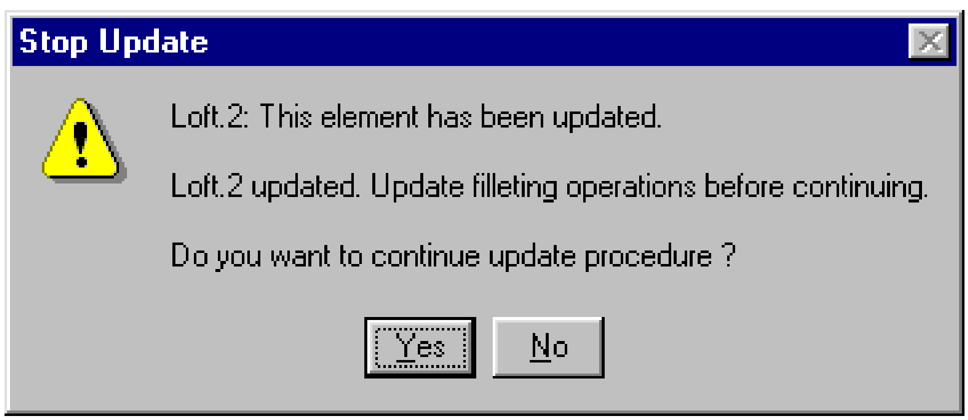
4. Click OK to confirm and close the dialog box.

The Stop Update.1 feature is displayed in the specification tree, below the element for which it was defined.



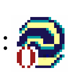
5. Whenever it is needed, click the **Update** icon  to update the whole part.

The updating process stops after having updated the element selected above, and issues the message as has been defined earlier:



6. Click Yes or No, depending on what you intend to do with the geometry created based on the selected element.

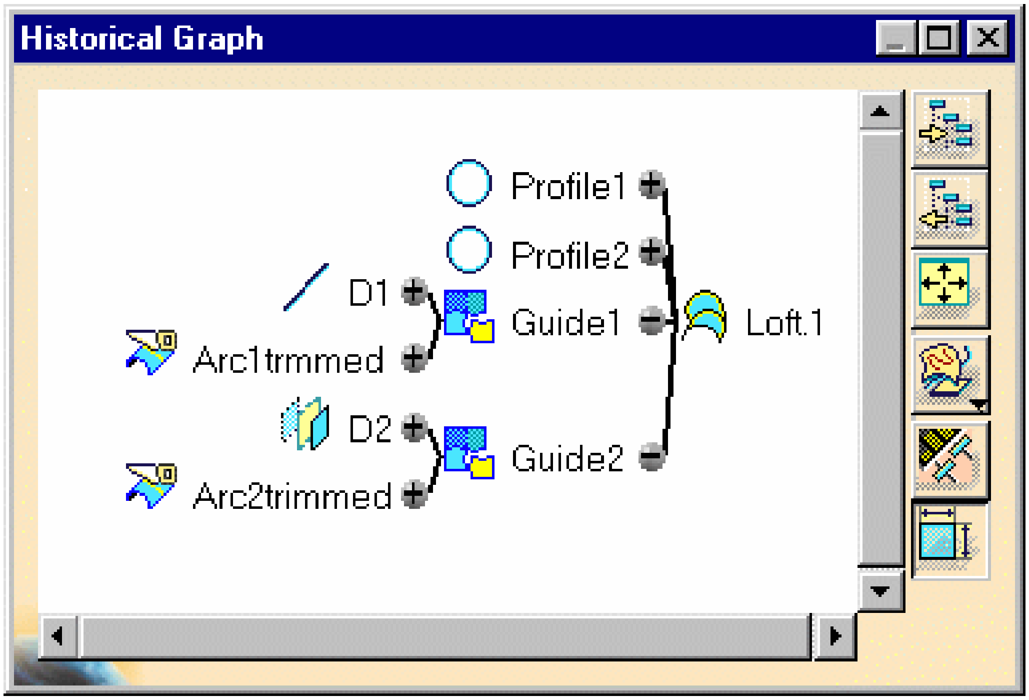
Would you no longer need this capability, you can:

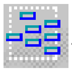
- right-click the element for which the stop was defined, choose the Properties contextual command and check the Deactivate stop update option from the Mechanical tab: the update will no longer at this element.
  You notice that when the capability is deactivated, the Stop Update icon changes to:  in the specification tree.
- right-click Stop Update.1 from the specification tree, and choose the **Delete** contextual command.

Using the Historical Graph

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

This task shows how to use the Historical Graph.



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

   The Historical Graph dialog box appears.

The following icon commands are available.

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

3. Just close the dialog box to exit this mode.

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.

Display Options and Graphic Properties

This task shows how to change the display option of clouds of points.

Open the Visu1.CATPart model the from the samples directory. It consists of four clouds of points:

- a mesh,
- a cloud of points,
- a set of scans,
- a set of grids.

Their default colors are respectively:

- orange,
- green,
- cyan,
- cyan.

The display options are available from the **Cloud Display Options box**. Further graphic properties are available from the **Edit/Properties** menu, in the **Graphic** tab.

From the **Cloud Display Options** box, you can:

- Choose the sampling of clouds of points (N of 100 points are displayed).
- Choose to display scans or grids as polylines, points, or both.
- Choose to display triangles, free edges, non-manifold edges of meshes. You can also choose their display mode: flat or smooth.

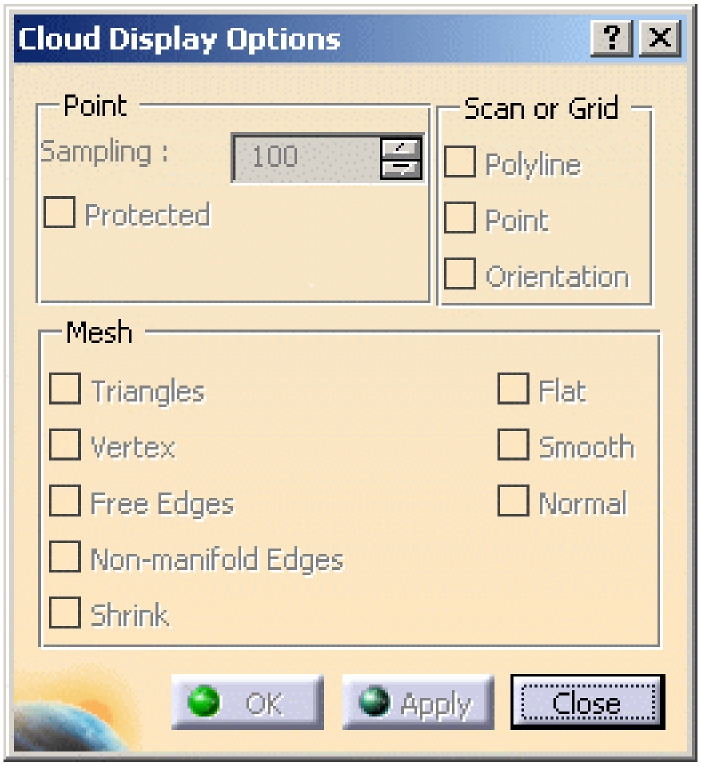
From the **Edit/Properties** menu, you can:

- Choose the fill color of the mesh and its transparency level,
- Choose the color and symbol of the points of a cloud,
- Choose the color, type and thickness of scans and grids,
- Choose to elements pickable or not.

Cloud Display Options box

The images below are only examples.

1. Click the **Cloud Display** icon at the bottom of the screen. The **Cloud Display Option** dialog box is displayed.

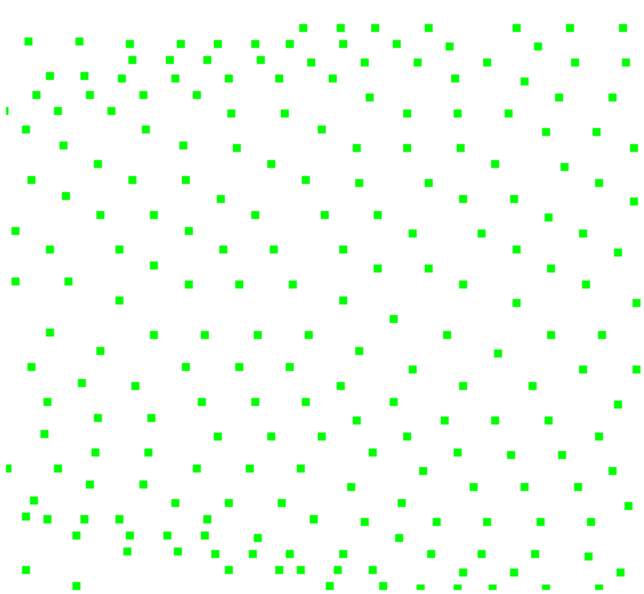


2. Select the cloud to modify. Display options are proposed according to the type of the cloud selected:

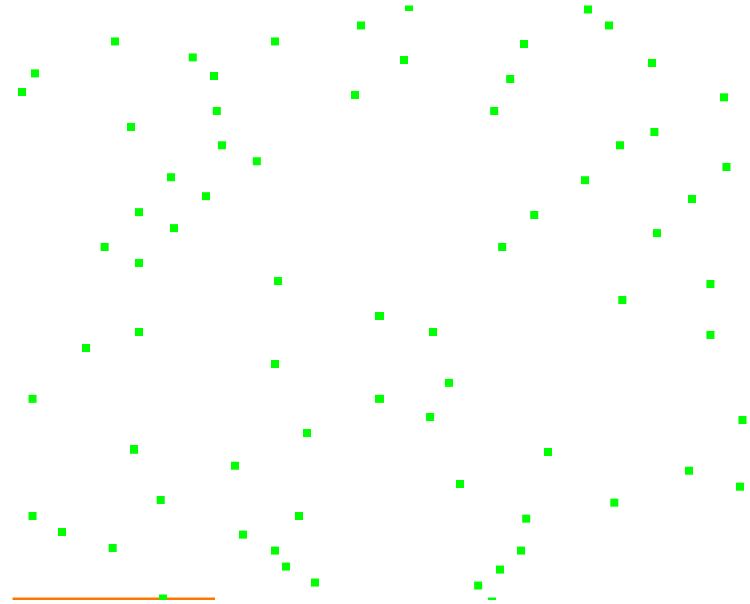
Following options are not yet available:

- Protected,
- Orientation,
- Shrink,
- Normal.

3. For the cloud of points, you can choose to display only a percentage of the points making the cloud, using the **Sampling** option. By default, 100% of the points are displayed. You can change this value with the associated spinner.



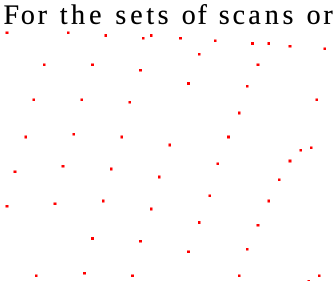
Sampling=100

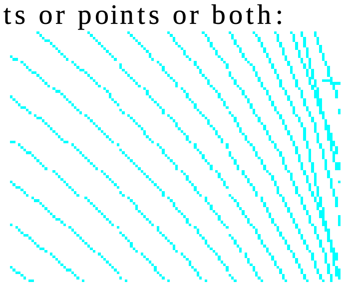


Sampling=25

The **Symbol** options are not available in that box, but in the Graphic Properties menu.

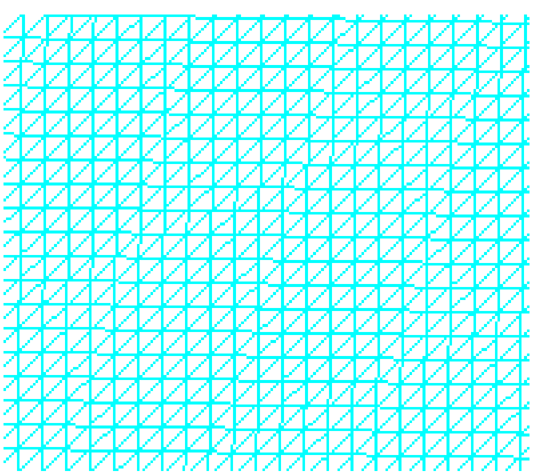
For the sets of scans or grids, you can display them as line of points or points or both:



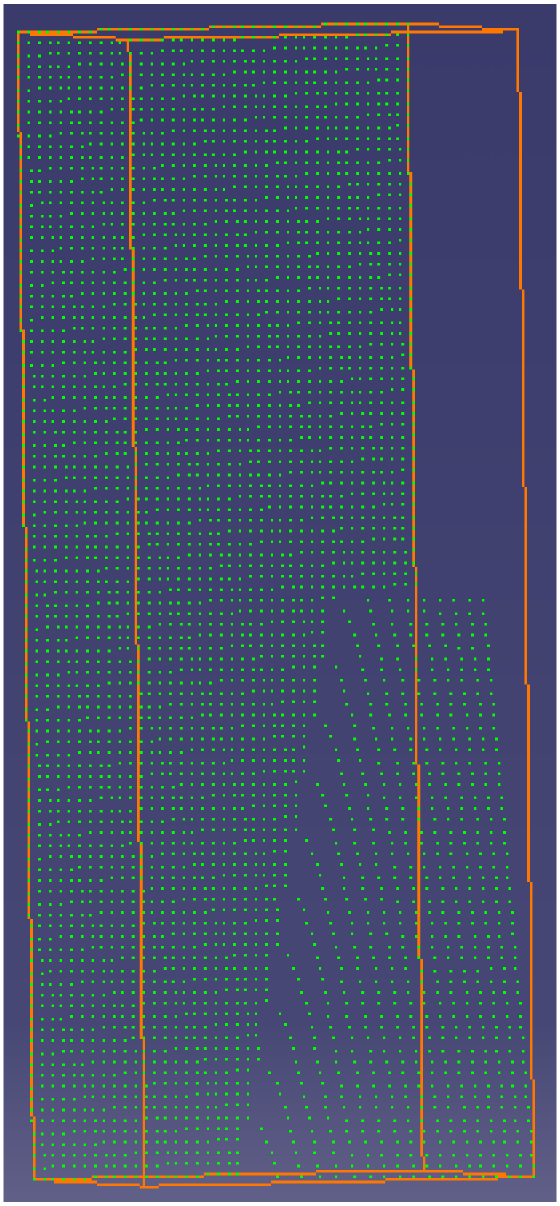
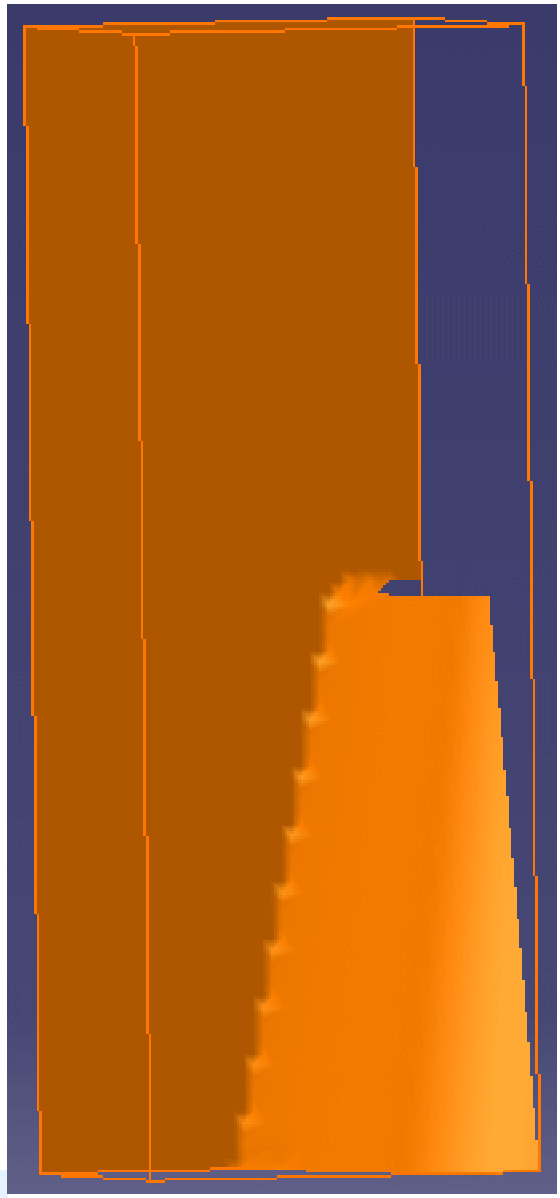


For the mesh, you can:

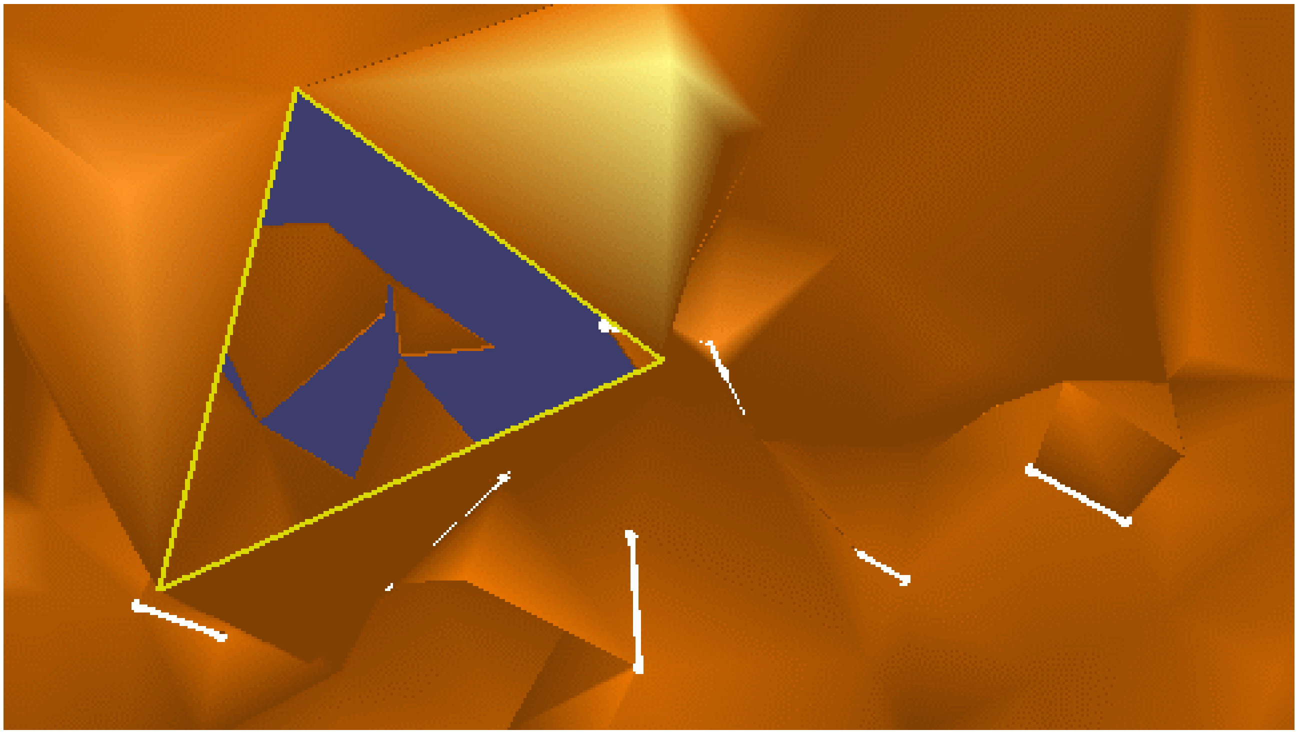
- display the triangles,



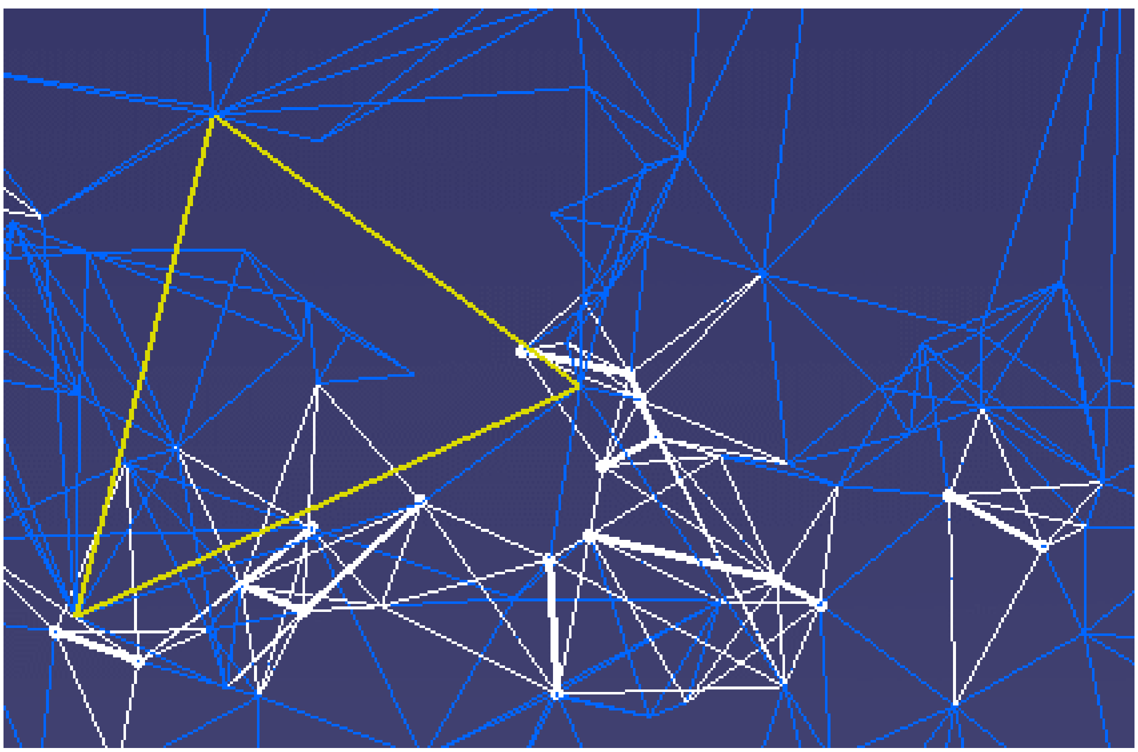
- visualize only the vertices for a lighter display (do not forget to de-activate the Smooth, Flat or Triangles options)


instead of


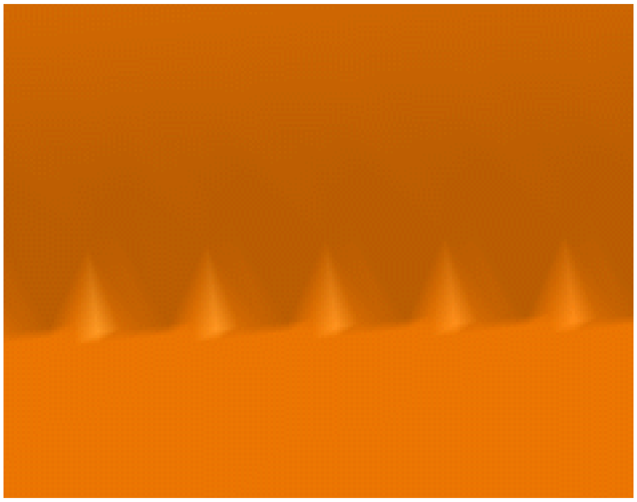
- the free edges in yellow,
- the non-manifold facets and their vertices in bold white lines.



- If you choose the display of triangles, the triangles accepting a non-manifold edge have their edges displayed as regular white lines.

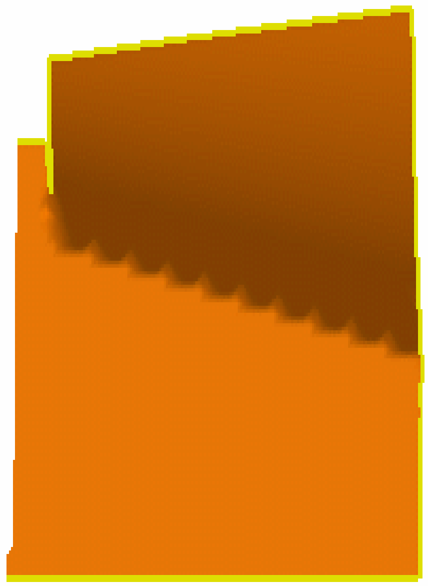


- display the mesh as a smooth or a flat mesh.

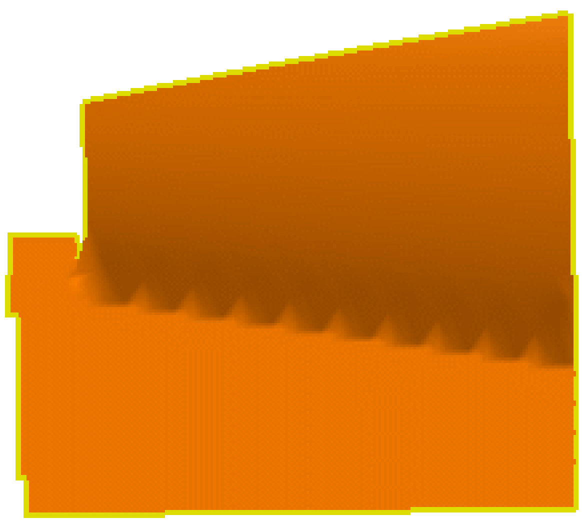
 or 

The free edges displayed are those of the complete cloud of points:

- if you activate only a portion of a cloud of points, the free edges of that portion are not displayed.



- if you remove a portion of a cloud of points, the free edges of the remaining portion are displayed.



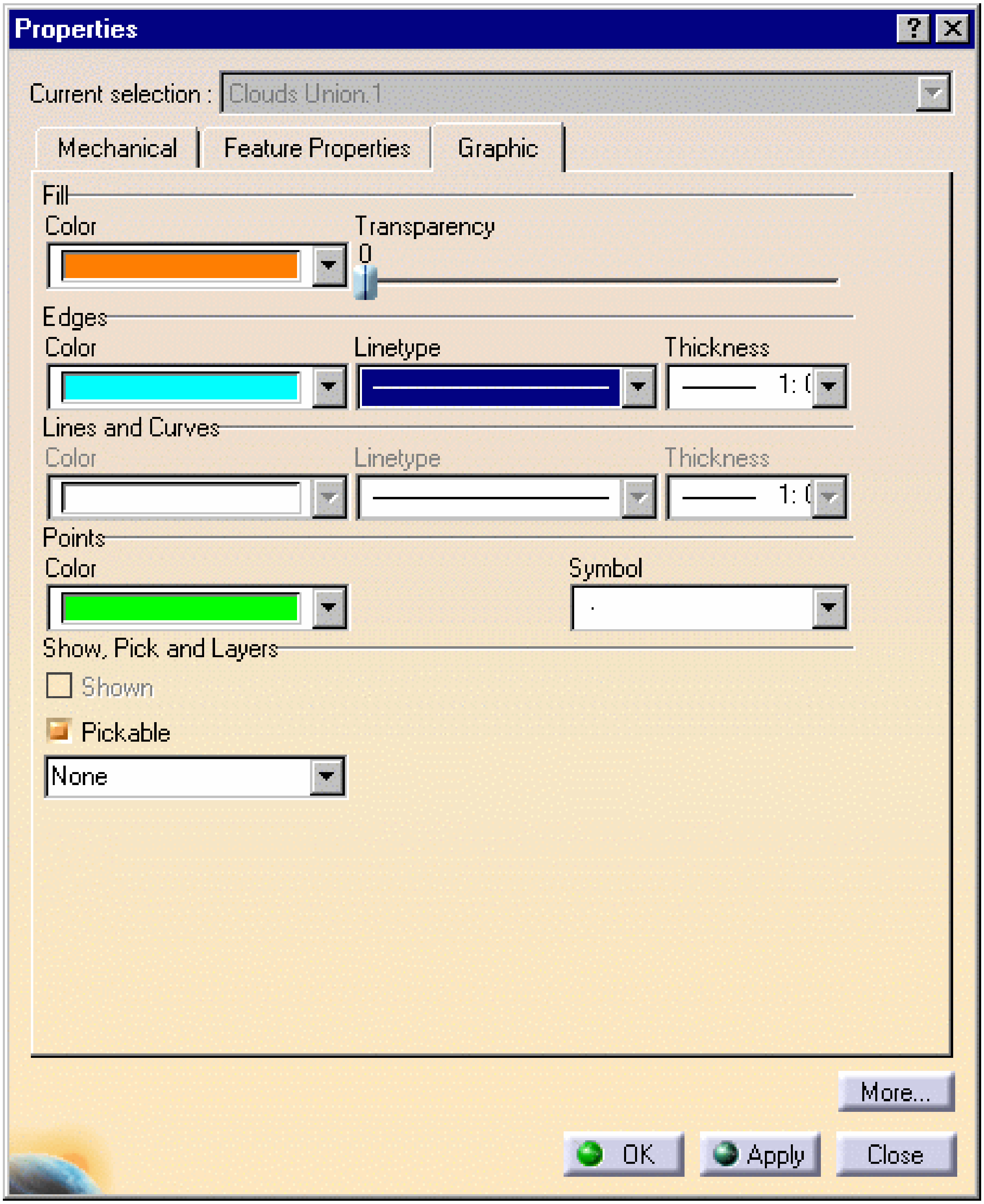
- If you move a cloud of points or a mesh, its graphic display options (not the graphic properties) are lost.
- The display options are not saved in the CATPart while the graphic properties are.

Edit/Properties menu (Graphic tab)

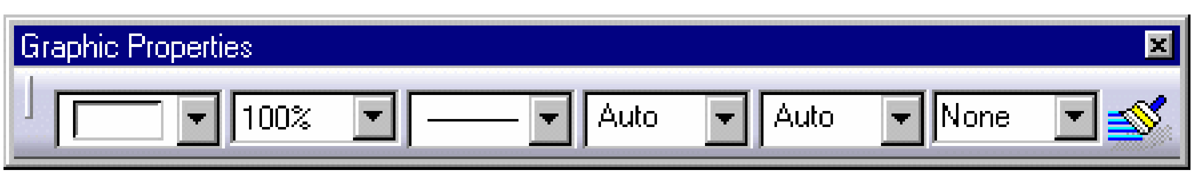
For more information about this menu, please refer to the Displaying and Editing Graphic Properties chapter in the CATIA Infrastructure user's guide.

The images below are only examples.

You can access this menu through **Edit / Properties**, or through the pop-up menu of the element, or display the **Graphic Properties** toolbar (**View / Toolbars / Graphic Properties**).

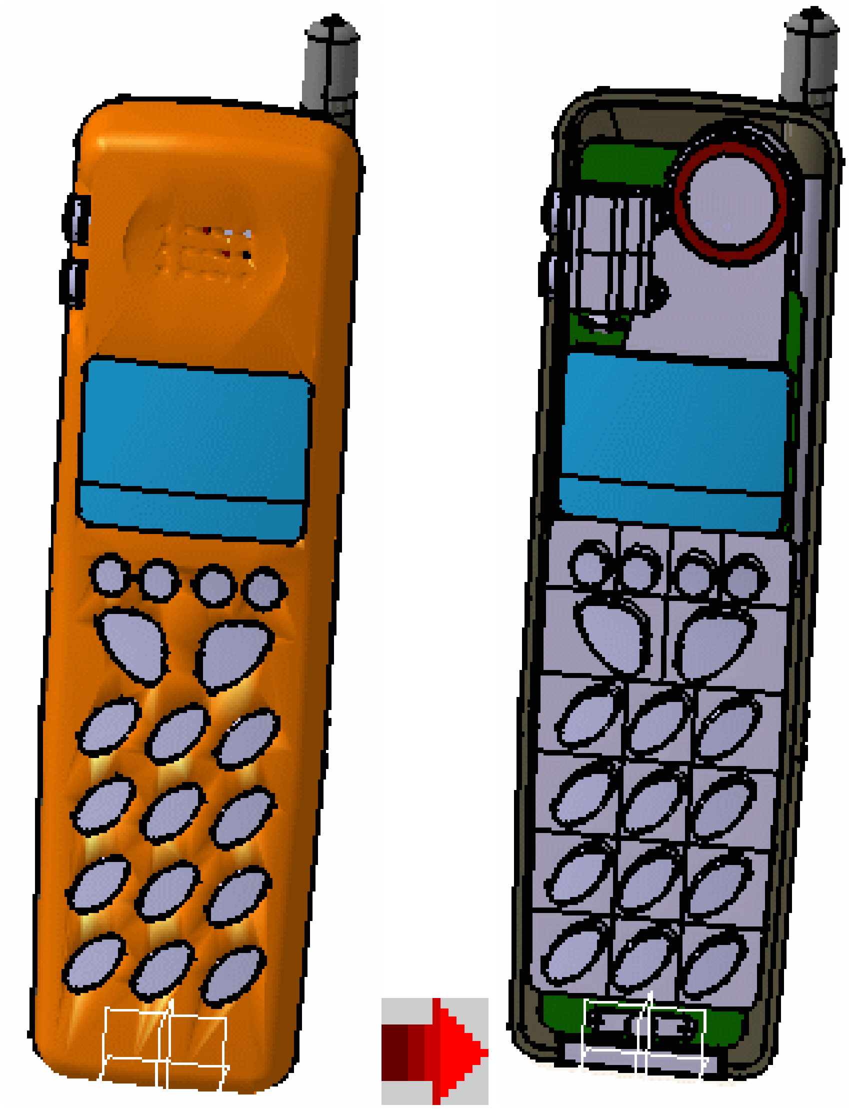


or



- The color displayed in the **Graphic Properties** toolbar applies to meshes only.
- The graphic properties are saved in the CATPart.

- Use **Fill/Color** and **Transparency** to modify the color and transparency of meshs:

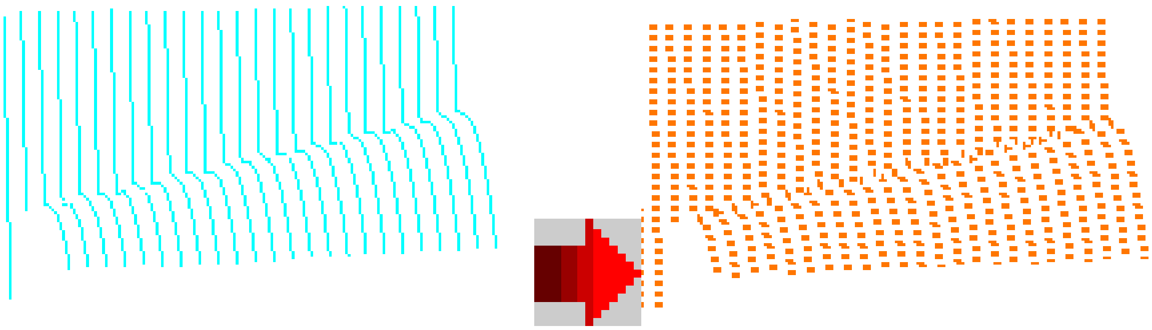


Please note that :

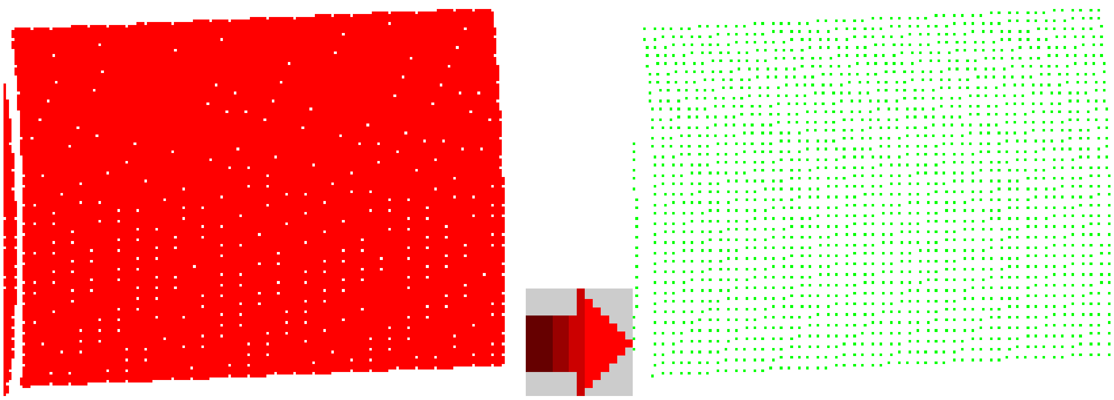
- the color of mesh free edges is yellow, and is not editable,
- the color of non-manifold edges is white, and is not editable,
- the default color of scans has changed to cyan.

For a higher transparency quality, go to **Tools/Options/Display/Performances** and check the **High (Alpha blending)** option.

- Use **Edges/Color**, **Line** type and **Thickness** to modify the display of scans and grids or of the triangles of a mesh :



- Use **Points/Color** and **Symbol** to modify the display of clouds of points:



- Use the **Pickable** check box to make an element pickable or not, and choose the pick option in the list below.

Managing Geometrical Sets

This task shows how to manage geometrical sets within the specification tree. This involves:

- inserting a geometrical set
- removing a geometrical set
- changing body
- sorting the contents of a geometrical set
- reordering components

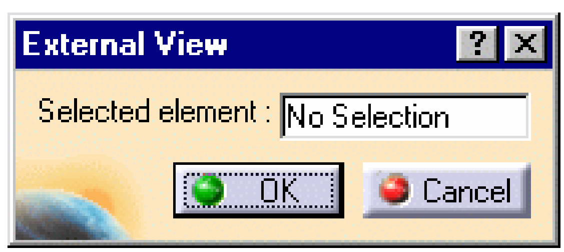
You will find other useful information in the Managing Groups and Hiding/Showing chapters.

A geometrical set enables to reorganize the specification tree when it becomes too complex or too long. You can put any element you wish in the geometrical set, it does not have to be structured in a logical way.

- You can insert and manipulate geometrical sets in the specification tree in much the same way as you manage files in folders.
- These management functions have no impact on the part geometry.
- You should refer to the Copying and Pasting section for information about how geometrical sets can be used in a part edition context.
- When loading the Generative Shape Design workbench, a Geometrical Set automatically becomes the current body.
  This also means that only the results of the Hybrid Body, i.e. the result of all the operations performed on geometry, is visible and not any intermediate state of the Hybrid Body.
- You can define the Generative Shape Design feature that is to be seen when working with another application, such as Generative Structural Analysis for example.

  To do this, while in the Generative Shape Design workbench:

  1. Choose the **Tools -> External View...** menu item
     The External View dialog box is displayed.
  2. Select the element belonging to a Geometrical Set that should always been seen as the current element when working with an external application.
  3. Click OK in the dialog box.



The selected element will be the visible element in other applications, even if other elements are created later in the .CATPart document, chronologically speaking.

To check whether an external view element has already been specified, choose the **Tools -> External View...** menu item again. The dialog box will display the name of the currently selected element. This also allows you to change elements through the selection of another element. Note that you cannot deselect an external view element and that only one element can be selected at the same time.

Open any .CATPart document containing Geometrical Sets.
You can also open the GeometricalSets2.CATPart document.

Inserting a Geometrical Set

1. In the specification tree, select an element as the location of the new geometrical set.

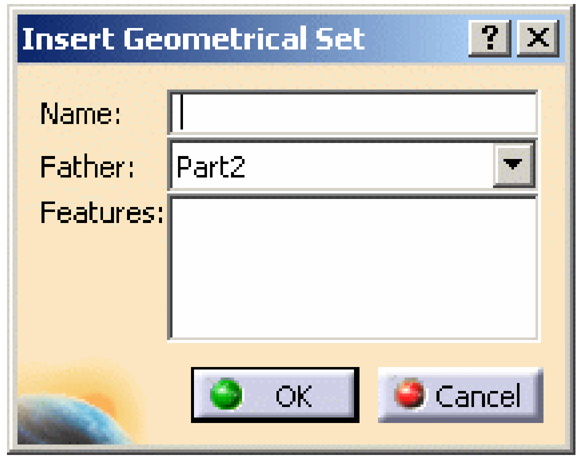
   This element will be considered as a child of the new geometrical set and can be a geometrical set or a feature.

2. Select the **Insert -> Geometrical Set** menu command.

   The Insert Geometrical Set dialog box is displayed.

   The Features list displays the elements to be contained in the new geometrical set.

3. Enter the name of the new geometrical set.
4. Use the Father drop-down list to choose the body where the new geometrical set is to be inserted.
   All destinations present in the document are listed allowing you to select one to be the father without scanning the specification tree. They can be:
   - bodies
   - geometrical sets
   - ordered geometrical sets
   - parts
5. Select additional entities that are to be included in the new geometrical set.



If all selected entities belong to the same geometrical set, the father of the new geometrical set is automatically set to the father of these entities.

6. Click OK to create the geometrical set at the desired location.

   The result is immediate. CATIA displays this new Geometrical Set.x, incrementing its name in relation to the pre-existing bodies, in the specification tree. It is created after the last current geometrical set and is underlined, indicating that it is the active geometrical set. The next created element is created within this geometrical set.

- You cannot create a geometrical set within an ordered geometrical set and vice versa.
- This Insert Geometrical Set dialog box is only available with the Generative Shape Design 2 product.

You can check the **Create a Geometrical Set when creating a new part** option in **Tools -> Options -> Infrastructure -> Part Infrastructure -> Part Document** tab if you wish to create a geometrical set as soon as you create a new part. For more information about this option, please refer to the Customizing section of the *Part Design User's Guide*.

Removing a Geometrical Set

Two methods are available:

- If you want to delete the geometrical set and all its contents:

1. Right-click the geometrical set then select the **Delete** contextual command.

- If you want to delete the geometrical set but keep its contents:
  This is only possible when the father location of the geometrical set is another geometrical set. This is not possible when the father location is a root geometrical set.

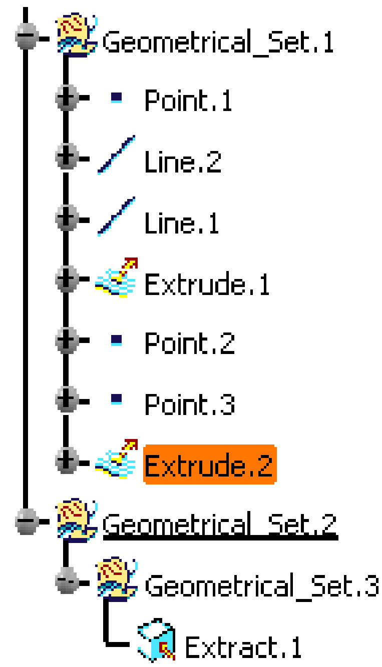
1. Right-click the desired geometrical set then select the **Geometrical Set.x object -> Remove Geometrical Set** contextual command.

The geometrical set is removed and its constituent entities are included in the father geometrical set.

You cannot delete a feature within a geometrical set created on the fly. Indeed this geometrical set is considered as private and can only be deleted globally.

Moving a Geometrical Set to a New Body

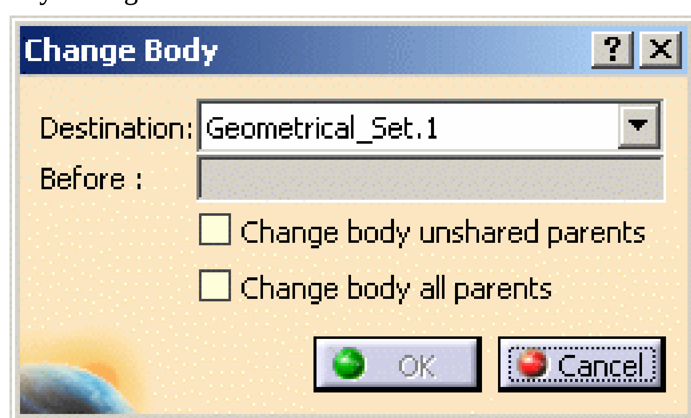
1. From the specification tree, select the element then choose the **Geometrical Set.object -> Change Geometrical Set...** item from the contextual menu.



Multi-selection of elements of different types is supported. However, note that in this case, the contextual menu is not available, and that you can access this capability using the **Edit** menu item.

The Change Body dialog box is displayed.

The list of destinations is alphabetically sorted.



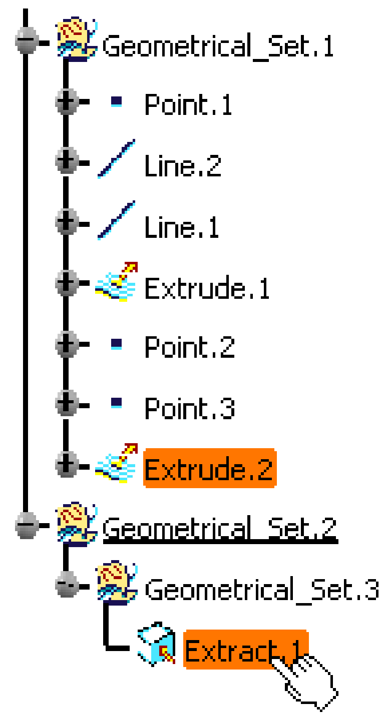
2. Select the **Destination** body where the geometrical set is to be located.

Here we selected Geometrical_Set.3.

You can do so by selecting the Body in the specification tree, or using the drop-down list from the dialog box.

By default, if you select a body, the geometrical set is positioned last within the new body. However, you can select any element in the new body, before which the moved geometrical set will be located.

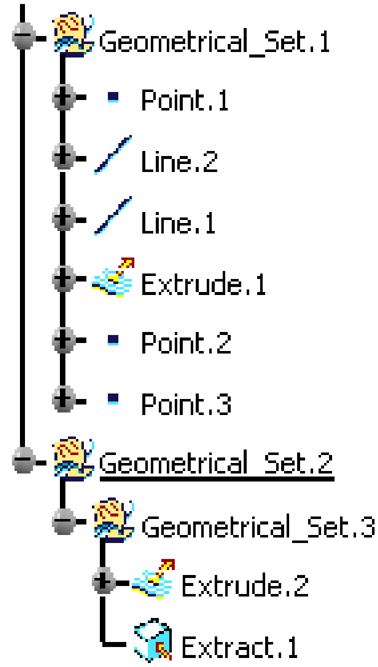
3. Select the element above which the one you already selected is to be inserted.



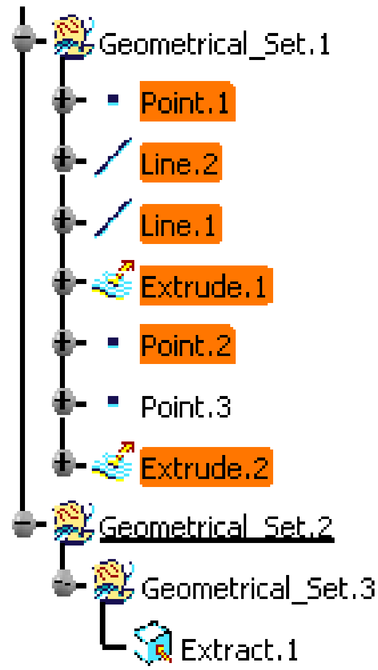
You can directly select this positioning element. In this case the **Destination** field of the Change Body dialog box is automatically updated with the Body to which this second element belongs.

4. Click OK to move the geometrical set to the new body.

The element selected first is moved to its new location in the specification tree, but geometry remains unchanged.

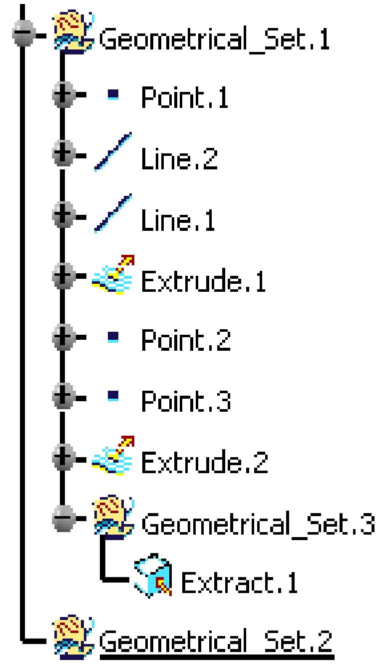


- Check the **Change body unshared parents** option to move all parents of the first selected element to its new location, provided these parents are not shared by any other element of the initial body.
In this case, all the unshared parents are highlighted prior to the move.



- Check the **Change body all parents** option to move all parents of the first selected element to its new location, regardless of whether these parents are used (shared) by any other element of the initial body.
In this case, all the parent elements are highlighted prior to the move.

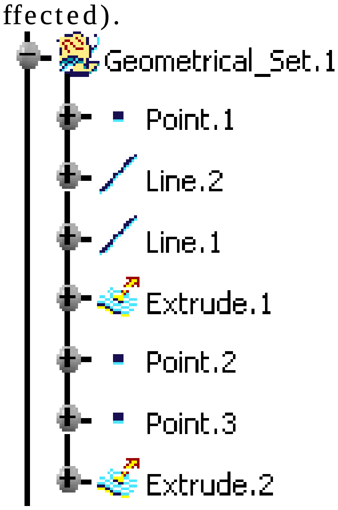
- You can move a whole branch, i.e. a whole body and its contents, at a time.
Here we moved Geometrical_Set.3 last in Geometrical_Set.1.



Sorting the Contents of a Geometrical Set

You may need to sort the contents of a Geometrical Set, when the geometric elements no longer appear in the logical creation order. In that case, use the Auto-sort capability to reorder the Geometrical Set contents in the specification tree (geometry itself is not affected).

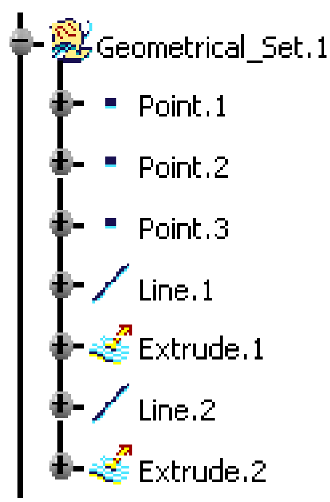
The Geometrical_Set.1 contains two extruded surfaces based on point-point lines. The specification tree looks like this:



1. Right-click the Geometrical_Set.1 from the specification and choose the **Geometrical_Set.1 object -> AutoSort** command.

   Instantly, the contents of the Geometrical Set are reorganized to show the logical creation process.

   The geometry remains unchanged.



Reordering Components within a Geometrical Set

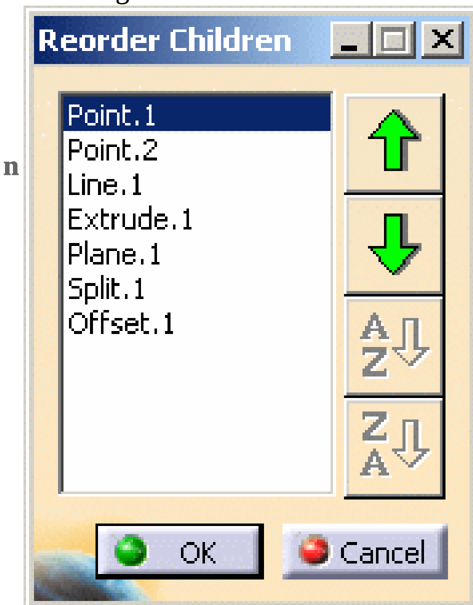
This capability enables you to reorder elements inside the same geometrical set.

1. Right-click the Geometrical_Set.1 from the specification tree and choose the **Ordered Geometrical Set.1 object -> Reorder Children** command.

   The Reorder Children dialog box is displayed.

2. Select an element.

3. Use the arrows to move an element up or down.



Reordering Features

The Reorder command allows you to move a feature in a Geometrical Set. These features can be:

- solids
- shape features
- sketches

For further information, please refer to the Reordering Features chapter in the *Part Design User's Guide*.

Replacing Features

This capability is only available on shape features.

Please refer to the Replacing or Moving Elements chapter in the *Part Design User's Guide*.
To manage this capability, the **Do replace only for elements situated after the In Work Object** option is available in **Tools -> Options -> Part Infrastructure -> General** tab. It allows you to make the Replace option possible only for features located below the feature in Work Object and in the same branch.

Operations

This chapter deals with operations in Quick Surface Reconstruction.

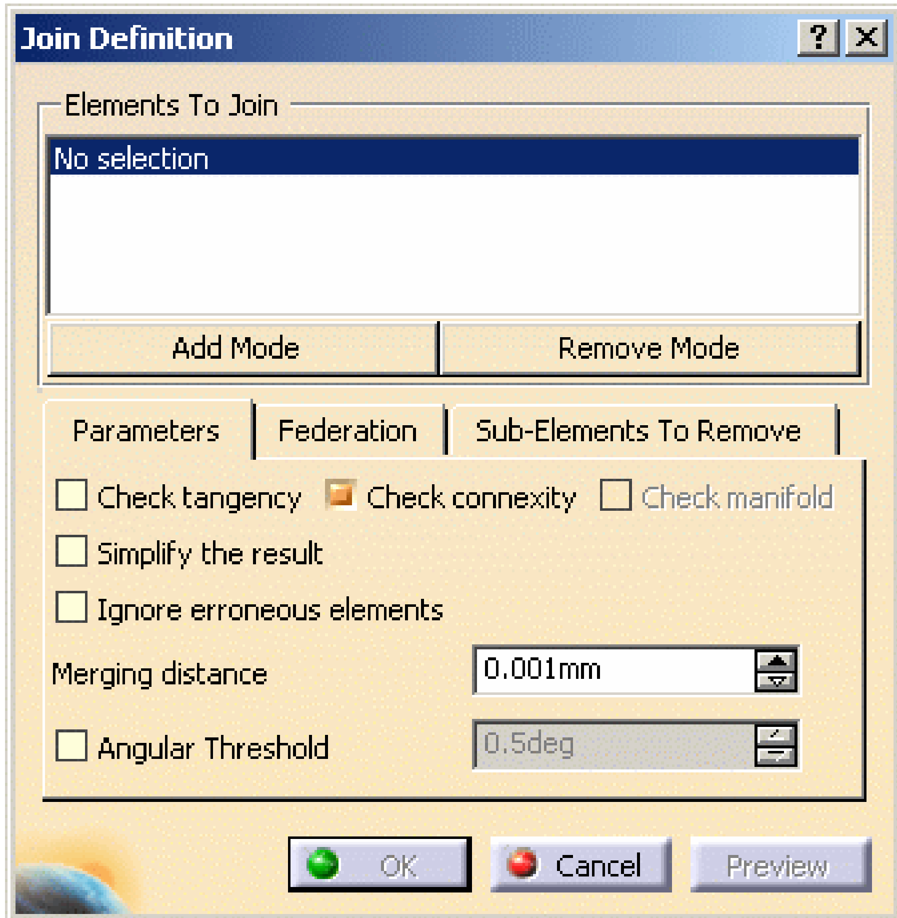
Joining Surfaces or Curves

This task shows how to join surfaces or curves.

Open the Join1.CATPart document.

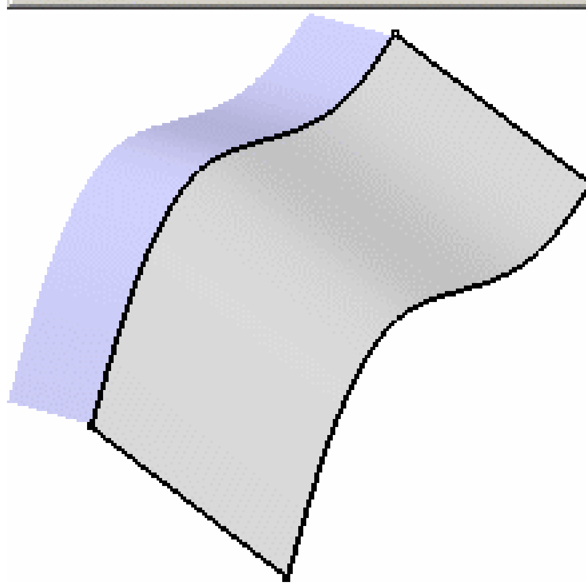
1. Click the **Join** icon.

   The Join Definition dialog box appears.



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

2. Select the surfaces or curves to be joined.

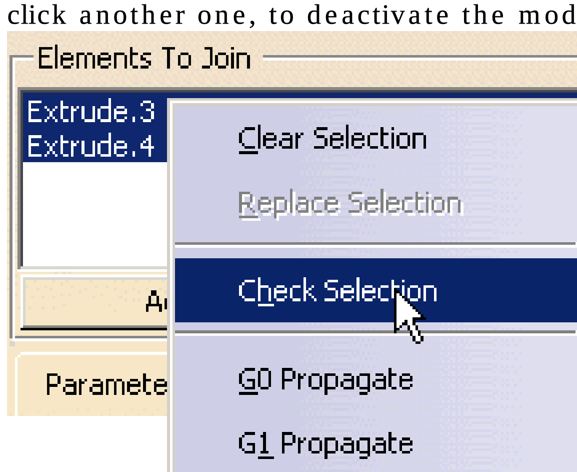


3. You can edit the list of elements to be joined:

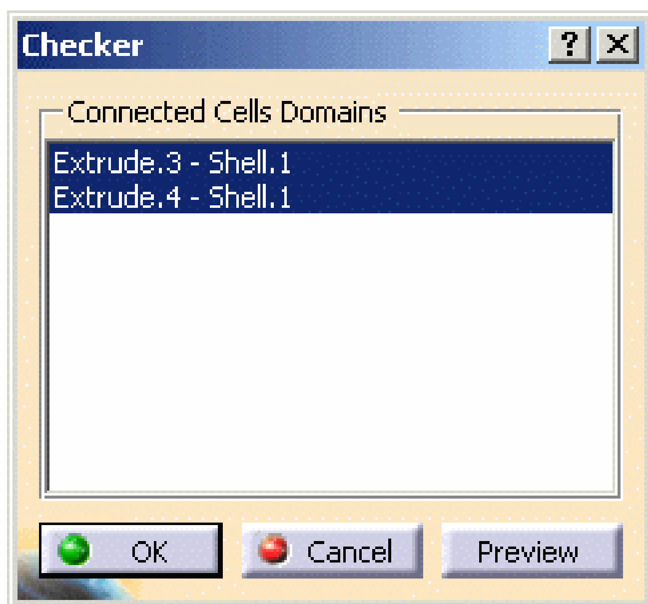
- by selecting elements in the geometry:
  - Standard selection (no button clicked):
    when you click an unlisted element, it is added to the list
    when you click a listed element, it is removed from the list
  - **Add Mode**:
    when you click an unlisted element, it is added to the list
    when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

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

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



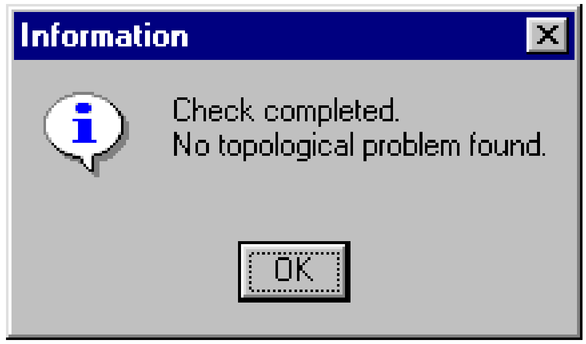
This let's you check whether any element to be joined presents any intersection (i.e. at least one common point) with other elements prior to creating the joined surface:



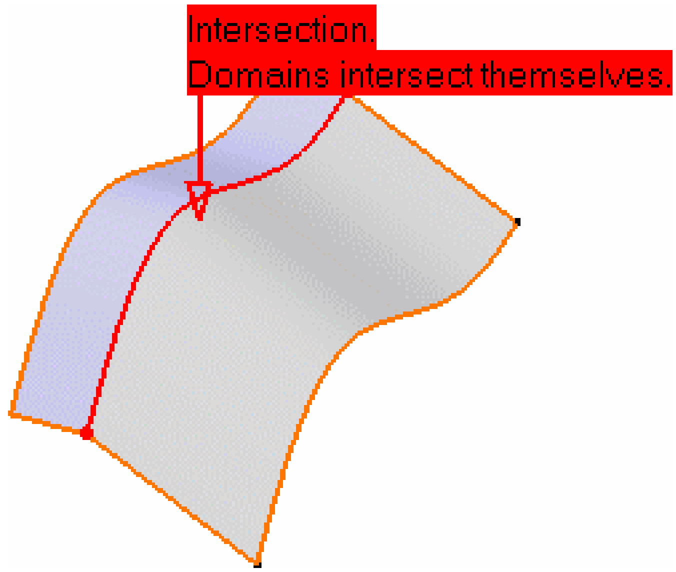
The Checker dialog box is displayed, containing the list of domains (i.e. sets of connected cells) belonging to the selected elements from the **Elements To Join** list.

5. Click Preview.

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



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



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

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

- **G0 Propagate**: the tolerance corresponds to the Merging distance value.
- **G1 Propagate**: the tolerance corresponds to the Angular Threshold value, if defined. Otherwise, it corresponds to the G1 tolerance value as defined in the part.

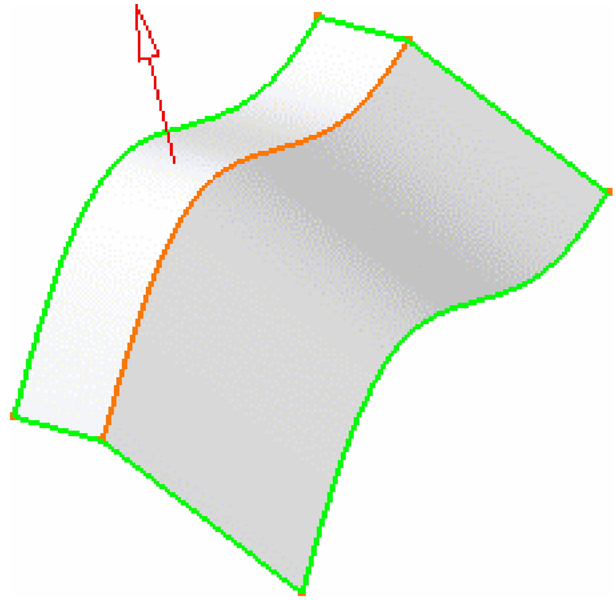
Each new element found by propagation of the selected element(s) is highlighted and added to the **Elements To Join** list.

Please note that:

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

8. Click Preview in the Join Definition dialog box.

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

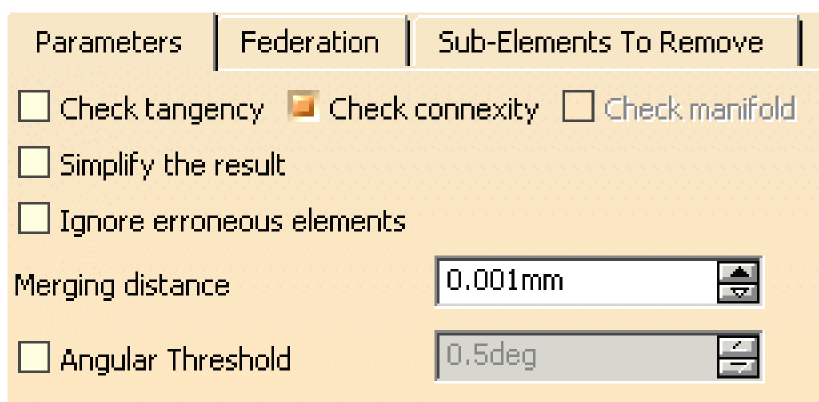


The join is oriented according to the first element in the list. If you change this element, the join's orientation is automatically set to match the orientation of the new topmost element in the list.

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

10. Check the **Check connexity** button to find out whether the elements to be joined are connex. If they are not, and the button is checked, an error message is issued indicating the number of connex domains in the resulting join.

    When clicking Preview, the free boundaries are highlighted, and help you detect where the joined element is not connex.



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

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

- The **Simplify the result** check button allows the system to automatically reduce the number of elements (faces or edges) in the resulting join whenever possible.
- The **Ignore erroneous elements** check button lets the system ignore surfaces and edges that would not allow the join to be created.

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

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

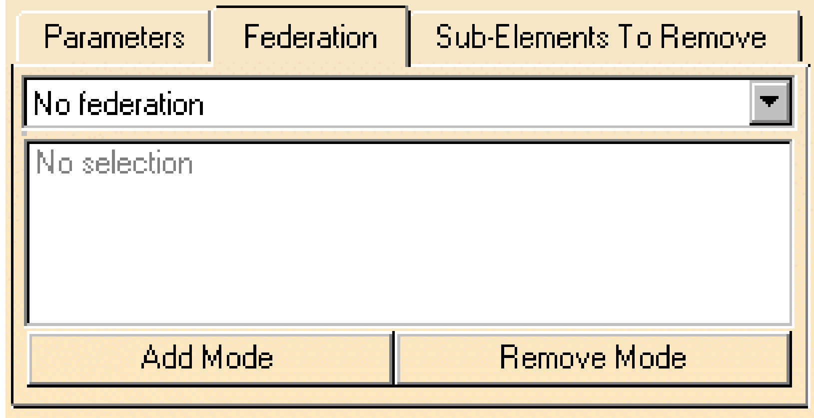
14. Click the **Federation** tab to generate groups of elements belonging to the join that will be detected together with the pointer when selecting one of them.

For further information, see Using the Federation Capability.

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

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

You can edit the sub-elements list as described above for the list of elements to be joined.



16. Check the **Create join with sub-elements** option to create a second join, made of all the sub-elements displayed in the list, i.e. those that are not to be joined in the first join.

    This option is active only when creating the first join, not when editing it.

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

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

Sometimes elements are so close that it is not easy to see if they present a gap or not, even though they are joined. Check the **Surfaces' boundaries** option from the **Tools -> Options** menu item, **General, Display**, **Visualization** tab.

Using the Federation Capability

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

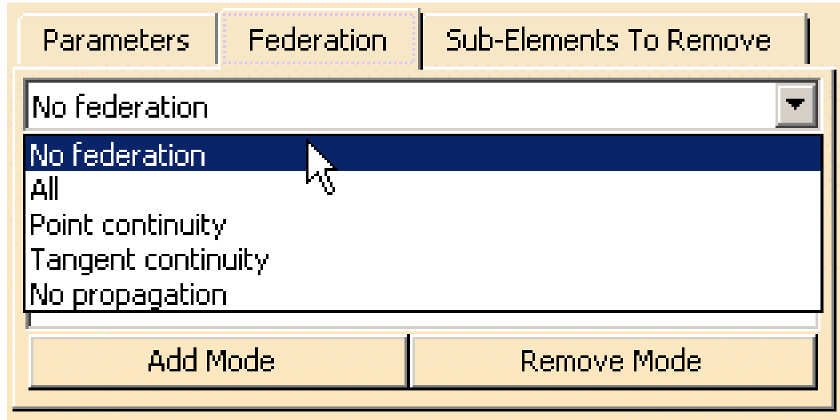
The purpose of the federation is to regroup several elements making up the joined surface or curve. This is especially useful when modifying linked geometry to avoid re-specifying all the input elements.

Open the Join2.CATPart document.

1. Create the join as usual, selecting all elements to be joined.

   (Make sure you do not select the Sketch.1).

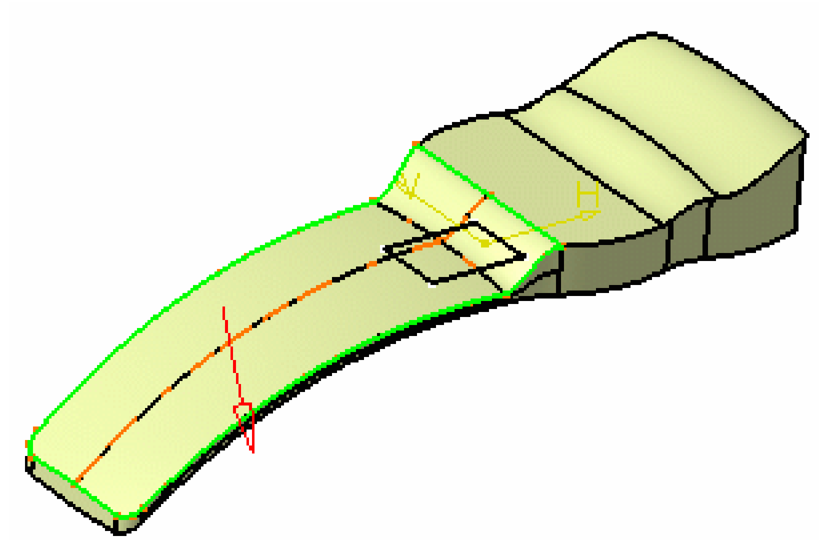
2. From the Join Definition dialog box click the **Federation** tab, then select one of the elements making up the elements federation.



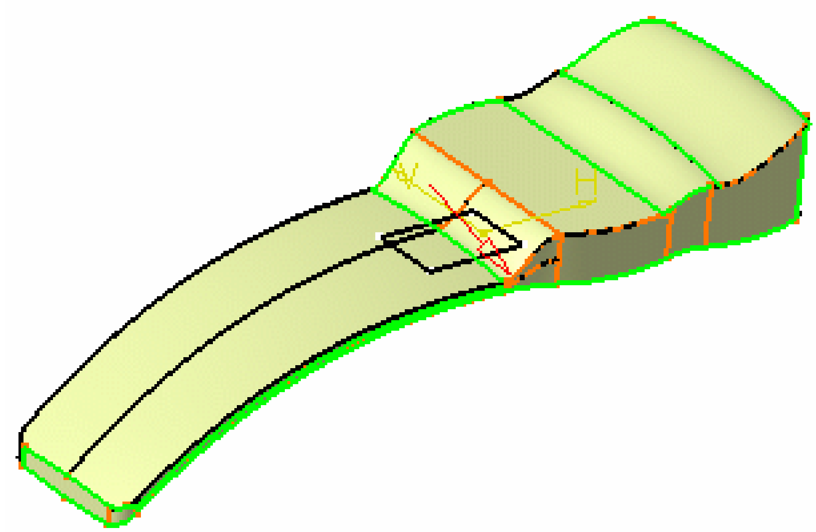
You can edit the list of elements taking part in the federation as described above for the list of elements to be joined.

3. Choose a propagation mode, the system automatically selects the elements making up the federation, taking this propagation mode into account.

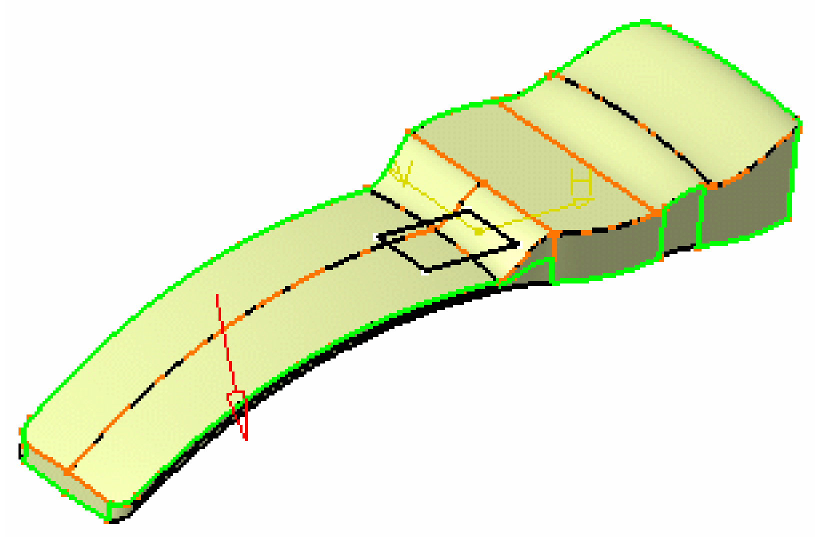
- **No federation**: only the elements explicitly selected are part of the federation



- **All**: all elements belonging to the resulting joined curve/surface are part of the federation

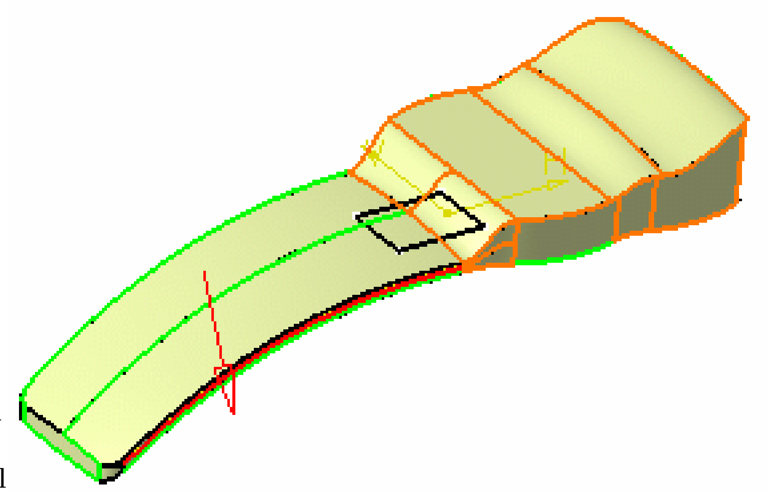


- **Point continuity**: all elements that present a point continuity with the selected elements and the continuous elements are selected; i.e. only those that are separated from any selected element is not included in the federation



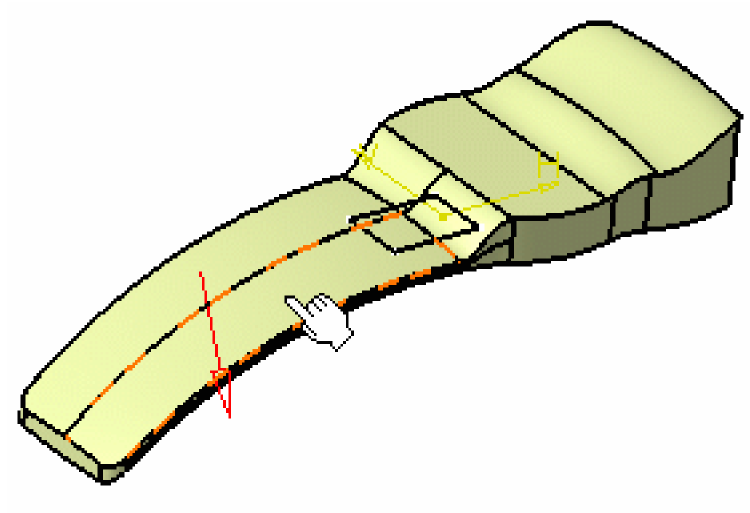
- **Tangent continuity**: all the elements that are tangent to the selected element, and the ones tangent to it, are part of the federation

  Here, only the top faces of the joined surface are detected, not the lateral faces.

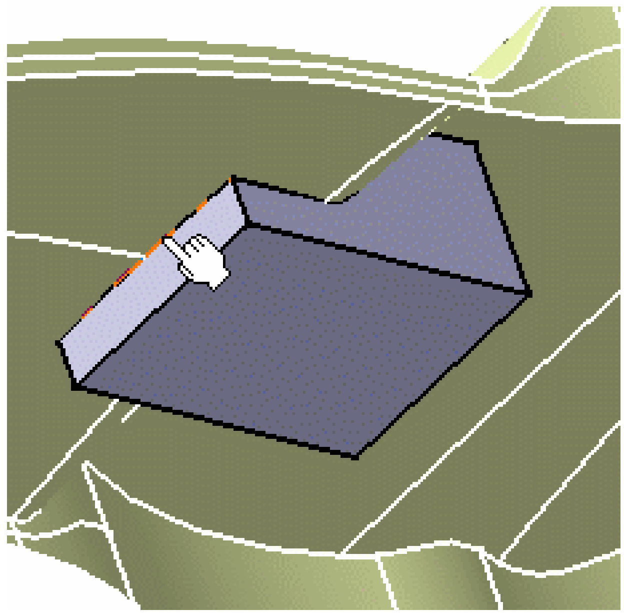


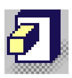
To federate a surface and its boundaries in tangency, you need to select the face as well as the edges: both face and edges will be federated.

- **No propagation**: only the elements explicitly selected are part of the propagation



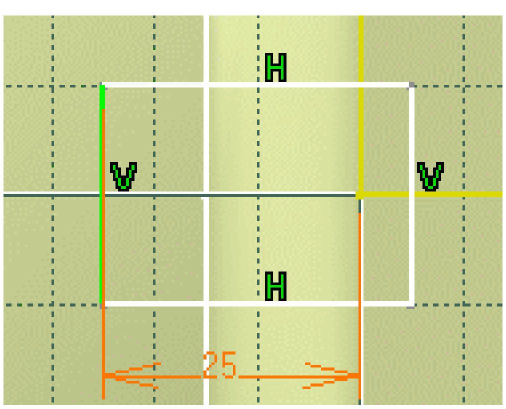
4. Choose the **Tangency Propagation** federation mode as shown above.

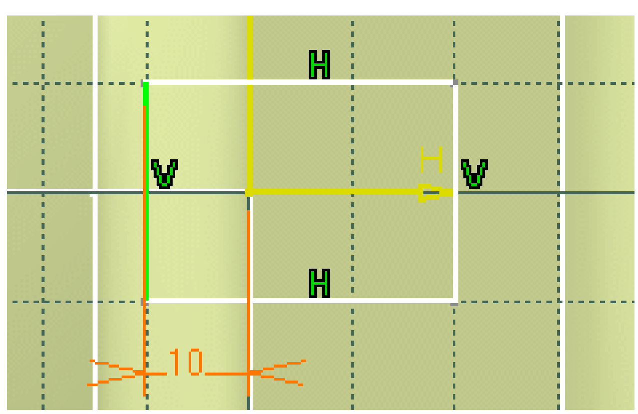


5. Move to the Part Design workbench, select the Sketch.1, and click the Pad  icon to create an **up to surface** pad, using the joined surface as the limiting surface.

6. Select the front edge of the pad, and create a 2mm fillet using the Edge Fillet  icon.

7. Double-click the Sketch.1 from the specification tree, then double-click the constraint on the sketch to change it to 10mm from the Constraint Definition dialog box.





Sketch prior to modification lying over two faces *Sketch after modification lying over one face only*

8. Exit the sketcher .

The up to surface pas is automatically recomputed even though it does not lie over the same faces of the surface as before, because these two faces belong to the same federation. This would not be the case if the federation including all top faces would not have been created, as shown below.

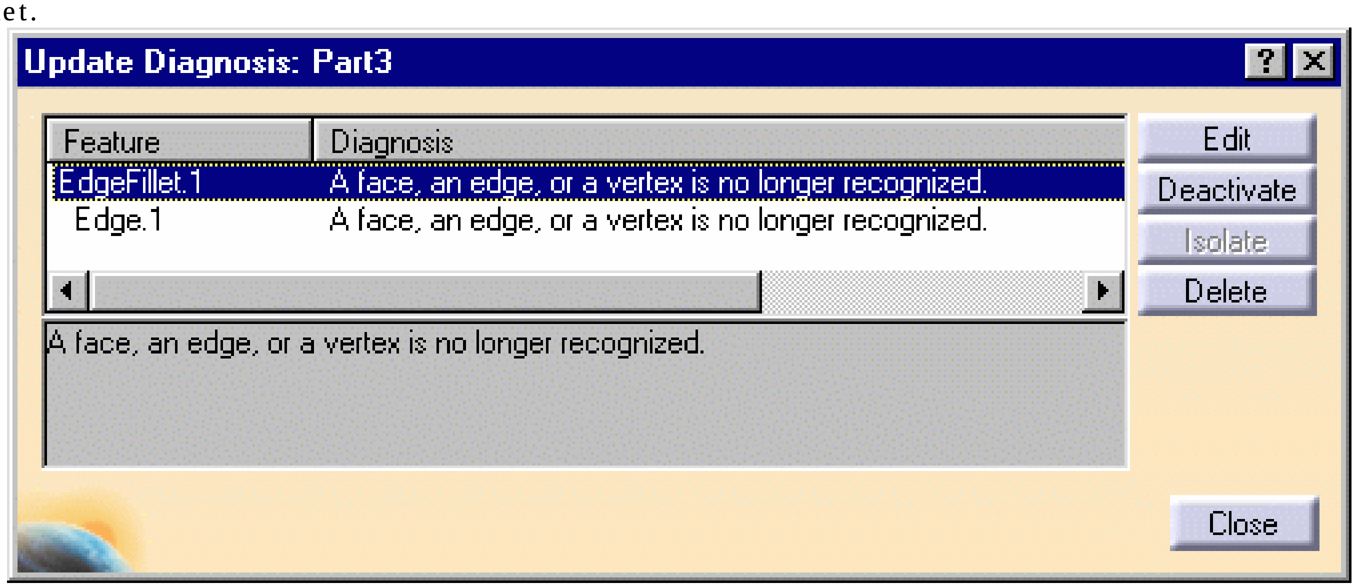
9. Double-click the joined surface (Join.1) to edit it, and choose the **No propagation** federation mode.

10. Click OK in the Join Definition dialog box.

A warning message is issued, informing you that an edge no longer is recognized on the pad.

11. Click OK.

The Update Diagnosis dialog box is displayed, allowing you to re-enter the specifications for the edge, and its fillet.



You then need to edit the edge and re-do the fillet to obtain the previous pad up to the joined surface.

12. Select the Edge.1 line, click the Edit button, and re-select the pad's edge in the geometry.

13. Click OK in the Edit dialog box.

The fillet is recomputed based on the correct edge.

Splitting Geometry

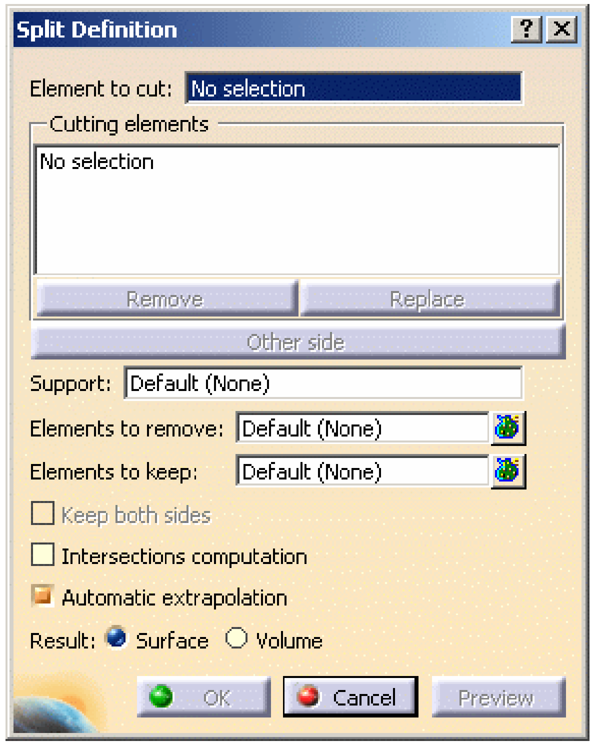
This task shows how to split a surface or wireframe element by means of a cutting element.
You can split:

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

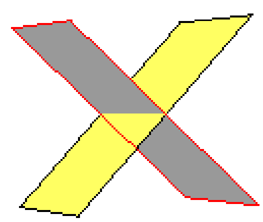
Open the Split1.CATPart document.

1. Click the **Split** icon.

The Split Definition dialog box appears.



2. Select the element to be split.

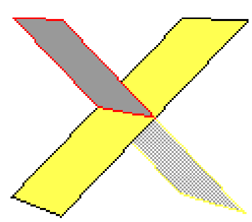


You should make your selection by clicking on the portion that you want to keep after the split.

3. Select the cutting element.

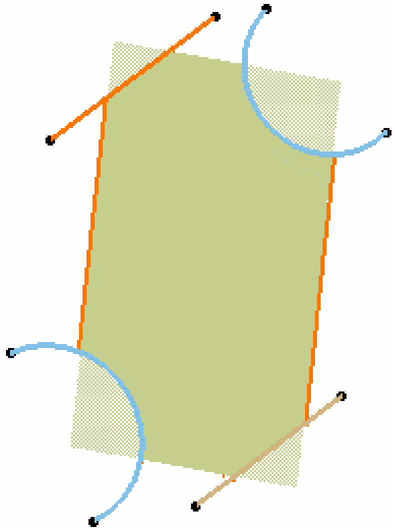
A preview of the split appears. You can change the portion to be kept by selecting that portion.

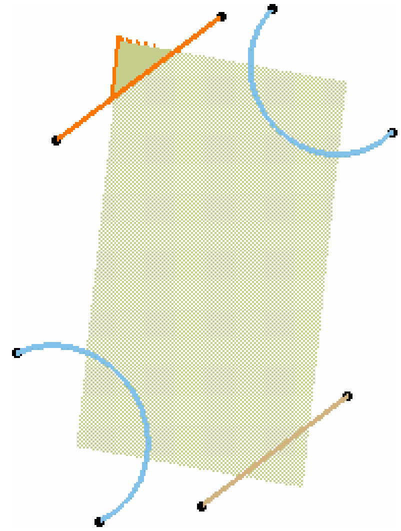
You can also select the portion to be kept by clicking the **Other side** button.



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

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

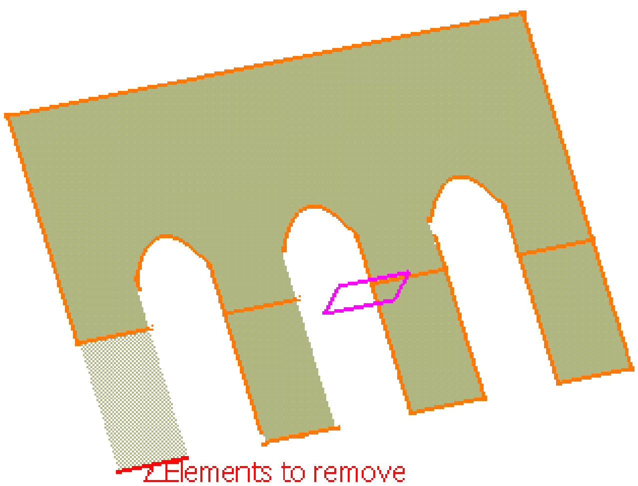




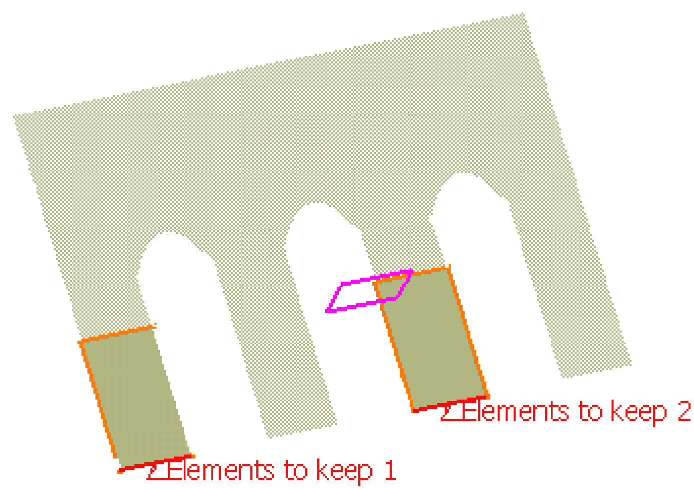
Would you need to remove, or replace, one of these cutting elements, select it from the list and click the **Remove** or **Replace** button.

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

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

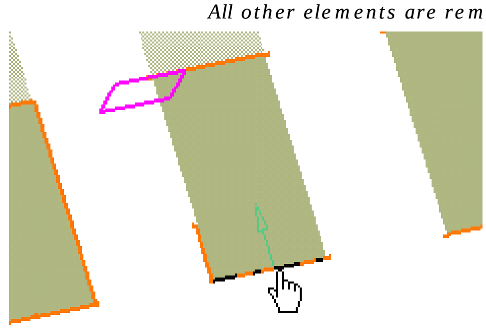


Only the selected element is removed.
All other elements are kept.



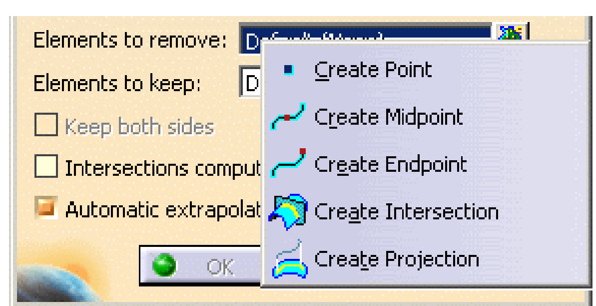
The selected elements are kept.
All other elements are removed.

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



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

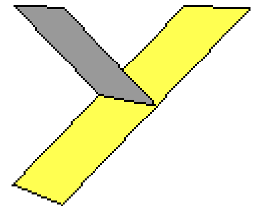
  A contextual menu is available on the **Elements to remove** and **Elements to keep** fields.



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

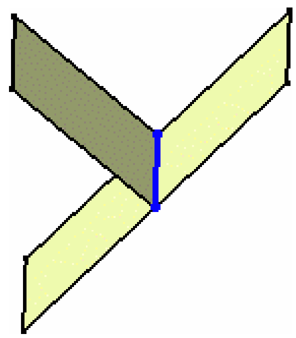
4. Click OK to split the element.

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

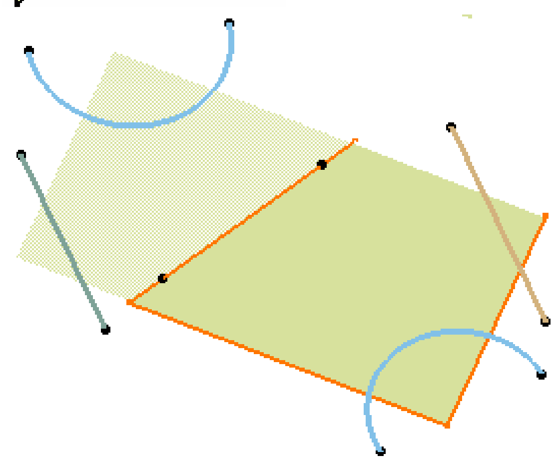


- Check the **Keep both sides** option to retain the other side of the split element after the operation. In that case it appears as aggregated under the first element.

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



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

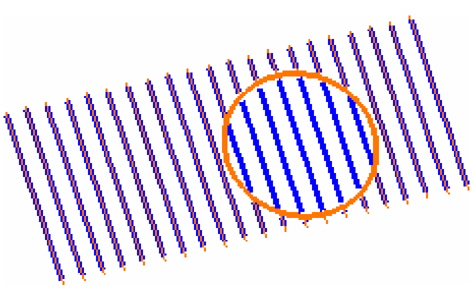


This option is available in the case of a split surface/curve or surface/surface.

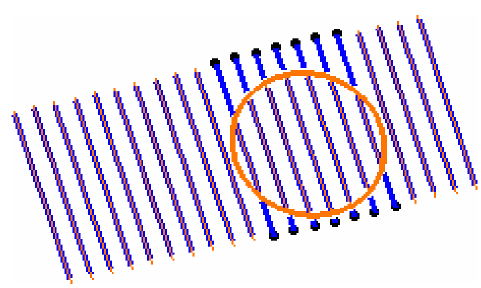
- Providing the element to cut is a volume and the cutting element is a volume or a surface, you can choose whether you want the result of the split to be a surface or a volume. To do so, switch to either **Surface** or **Volume** option. This switch only concerns volumes since the transformation of a surface can only be a surface. Thus in case of multi-selection of volumes and surfaces, the switch only affect volumes.
  If the result of the split is a volume, the split is a modification feature.
  If the result of the split is a surface, the split is a creation feature.
  To have further information about volumes, please refer to the corresponding chapter.

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

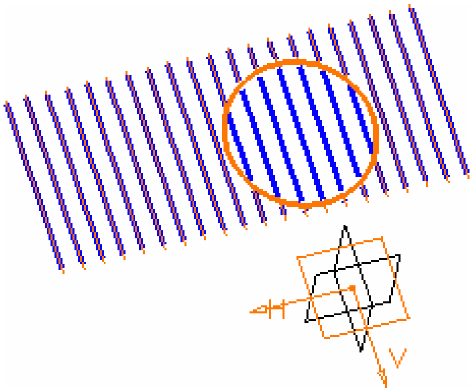
The non disconnected elements of the element to cut are kept in the result of the split.



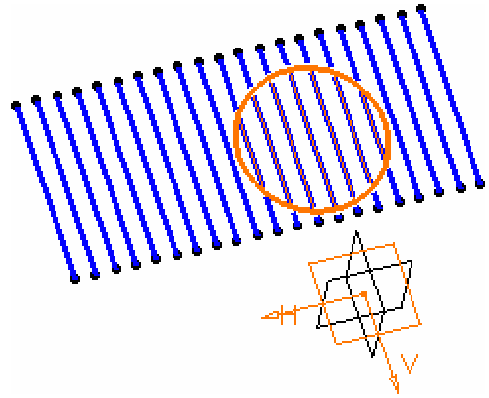
Splitting with no support selected: first solution



Splitting with no support selected: second solution



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

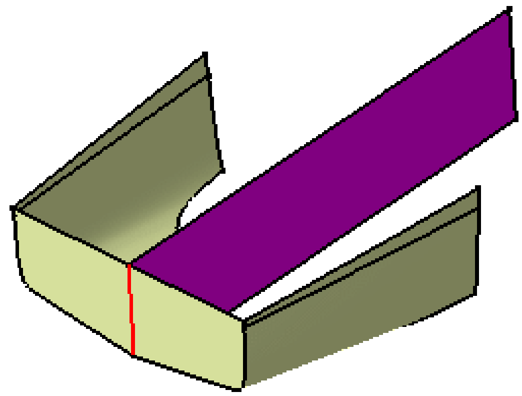


Splitting with a selected support (xy plane): second solution

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

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

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



- When splitting a closed surface or a curve by connex elements, an error message is issued. You need to create a join feature of non connex elements and cut the closed surface or curve with this join feature.

Splitting surface/curve or surface/surface

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

Split surface/curve

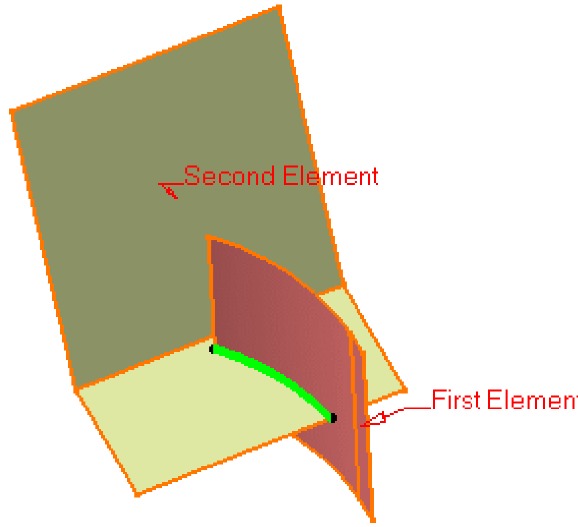
1. First, the cutting element (the curve) is laid down the surface.
2. Then, the result of step 1 is tangentially extrapolated in order to split the surface correctly (as shown in following figure). However, when this extrapolation leads to the intersection of the cutting element with itself prior to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

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

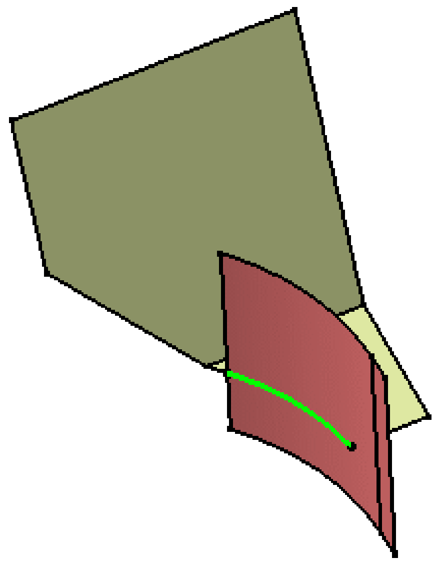
Split surface/surface

Open the Split2.CATPart document.

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



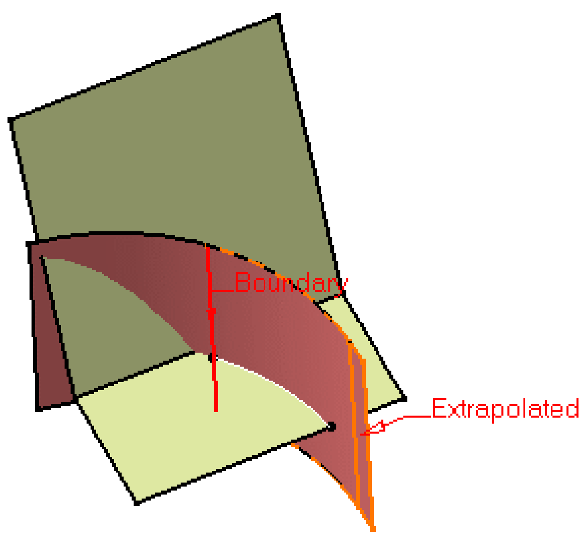
2. Then, the result of the intersection is automatically extrapolated in tangency up to the closest free edges of the element to cut. The result of the extrapolation is used as the cutting element and the split is created.



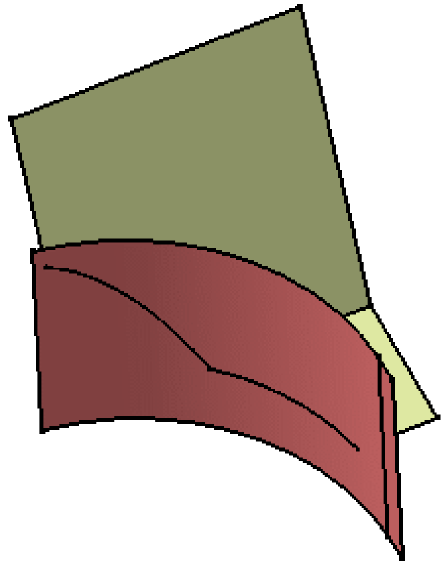
Please note that it is **not** the cutting element which is extrapolated but the result of the intersection.

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

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



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

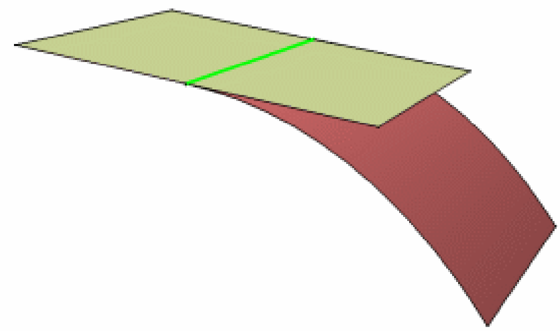


Avoid using input elements which are tangent to each other since this may result in geometric instabilities in the tangency zone.

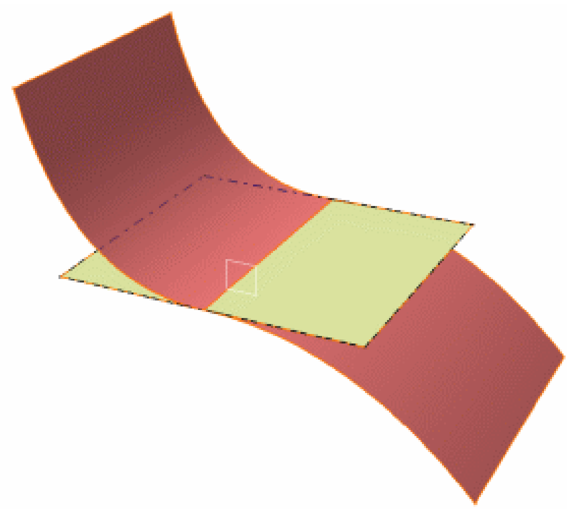
In case surfaces are tangent or intersect face edges, please process as follow in order to avoid indeterminate positioning.

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

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

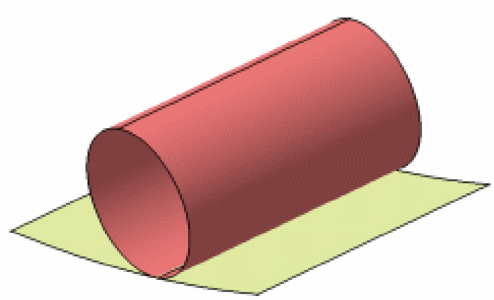


Steps 2 and 3 may be optional if the tangency constraint between the two surfaces has been clearly defined by the user during the surface creation.



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

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



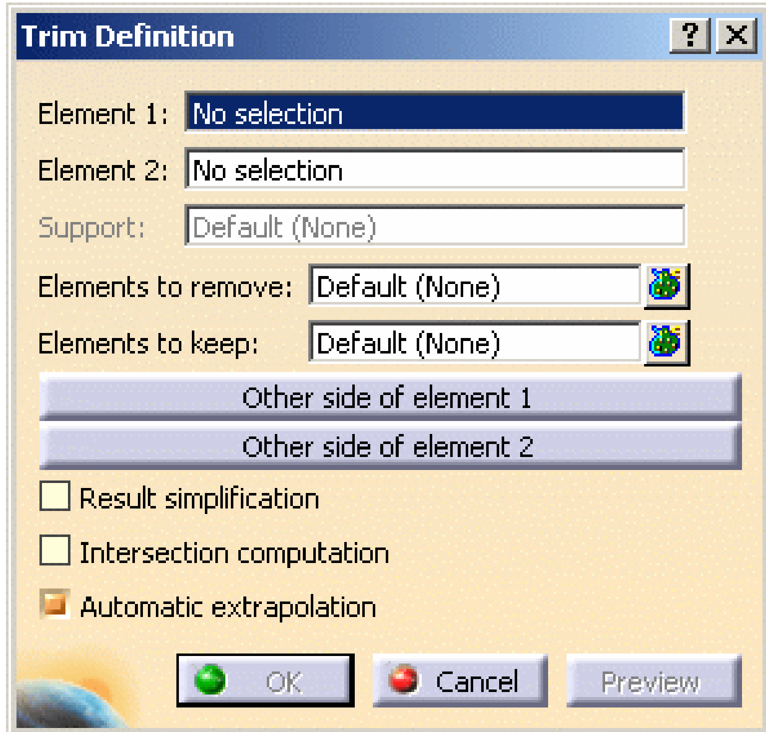
Trimming Geometry

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

Open the Trim1.CATPart document.

1. Click the **Trim** icon .

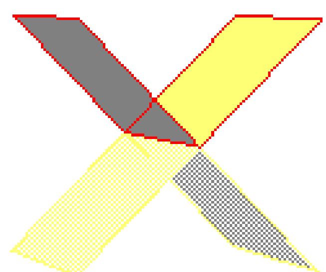
   The Trim Definition dialog box appears.



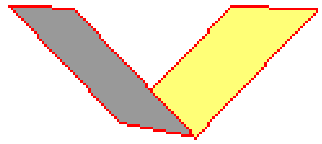
2. Select the two surfaces or two wireframe elements to be trimmed.

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

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



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

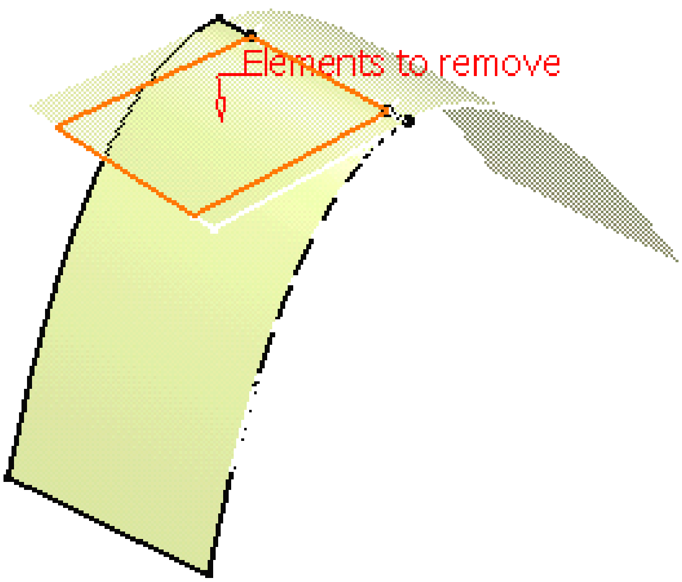


The trimmed element (identified as Trim.xxx) is added to the specification tree.

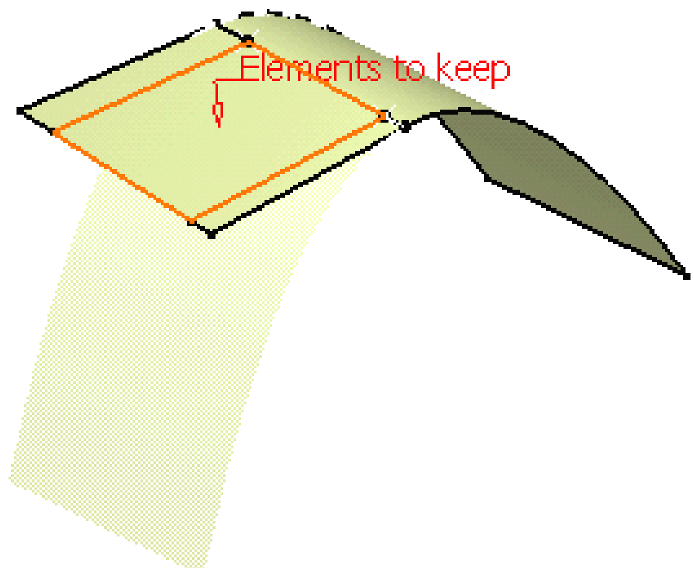
- You should make your selections by clicking on the portions that you want to keep after the trim.
- Please refer to the Splitting Geometry chapter in the case surfaces intersect face edges.

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

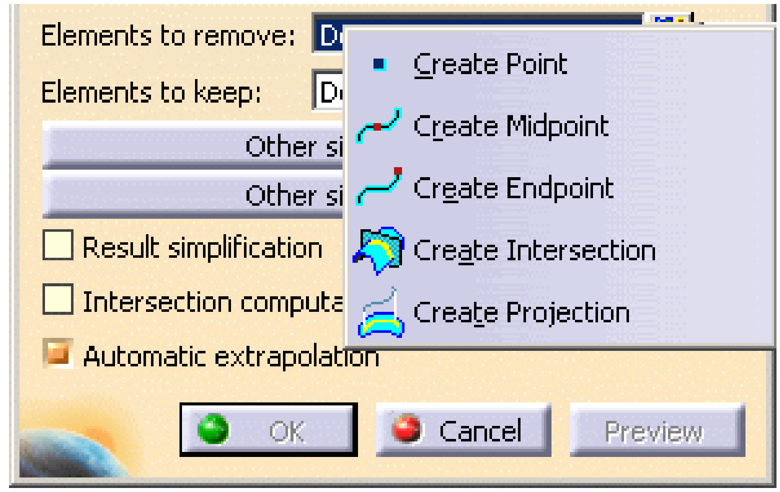
- Click in the field of your choice to be able to select the elements in the 3D geometry.
- Right-click in the field either to clear the selection or display the list of selected elements.



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



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



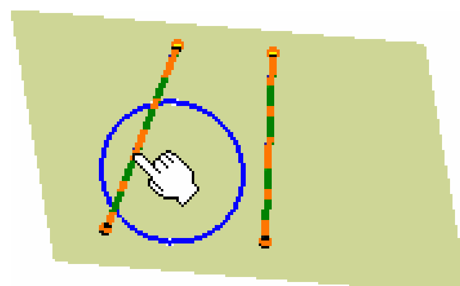
- You can also select a point to define the portion to keep or to remove.

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

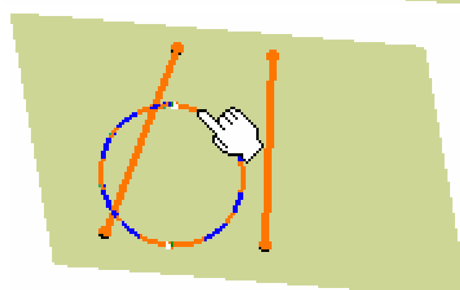
- You do not need to select elements to keep if you already selected elements to remove and vice-versa.

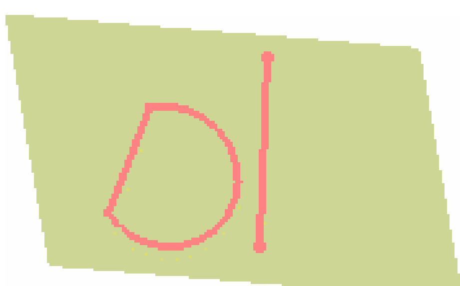
- When trimming wires (curve, line, sketch and so forth) by another wire, you can select a support to define the area that will be kept after trimming the element. It is defined by the vectorial product of the normal to the support and the tangent to the trimming element.

This is especially recommended when trimming a closed wire.

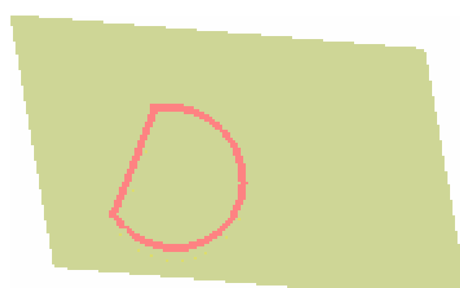


In our example, the Sketch composed of two lines (Sketch.11) is trimmed by the circle (Sketch.10).





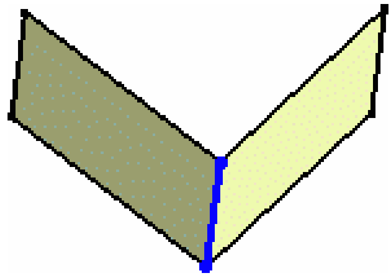
Resulting trimmed element without support selection



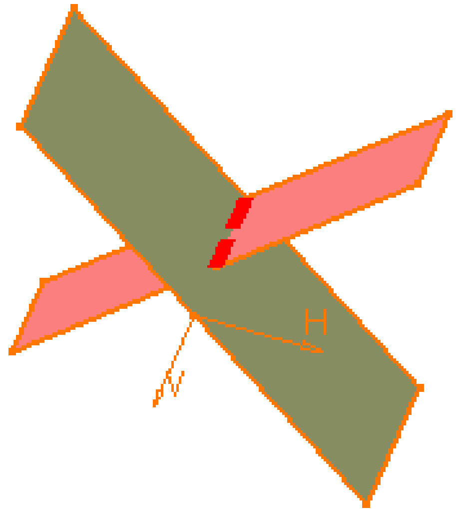
Resulting trimmed element with support selection

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

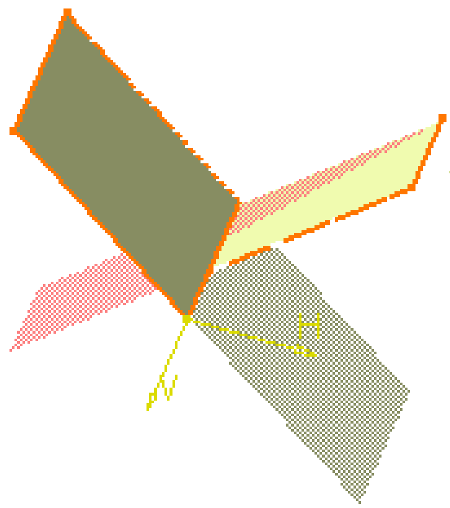
- Check the **Intersection computation** button to create a completely independent element when performing the trimming operation. In that case it appears as a separate Intersect.xxx element in the specification tree.



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



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



Extrapolating Surfaces

This task shows you how to extrapolate a surface boundary.

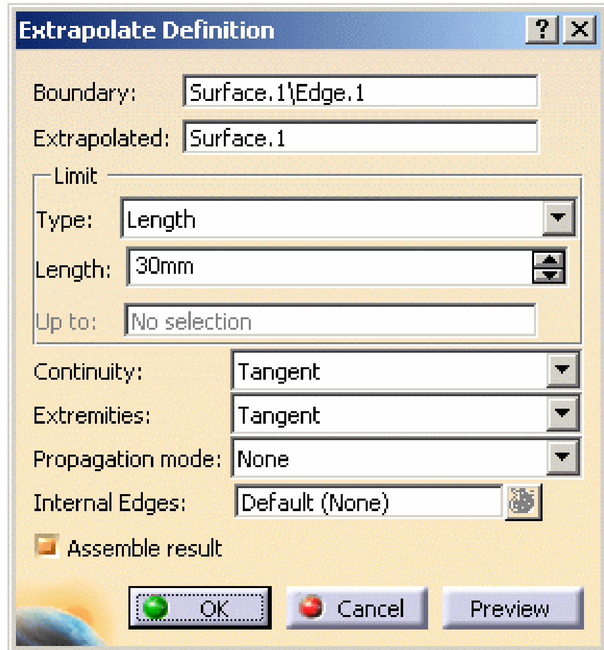
Open the Extrapolate1.CATPart document.

1. Click the **Extrapolate** icon.

   The Extrapolate Definition dialog box appears.

2. Select a surface **Boundary**.

3. Select the surface to be **Extrapolated**.

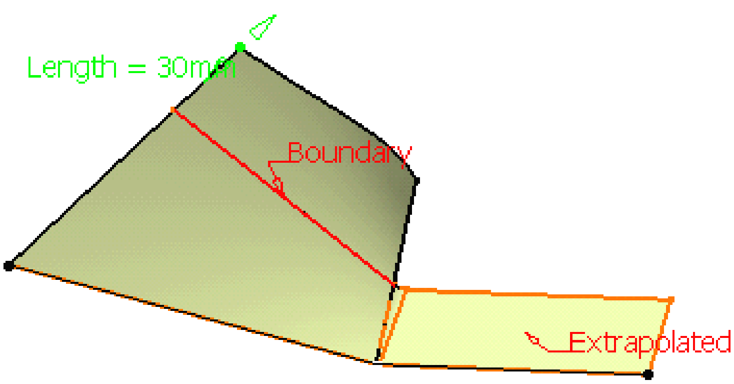


4. Specify the **Limit** of the extrapolation by either:

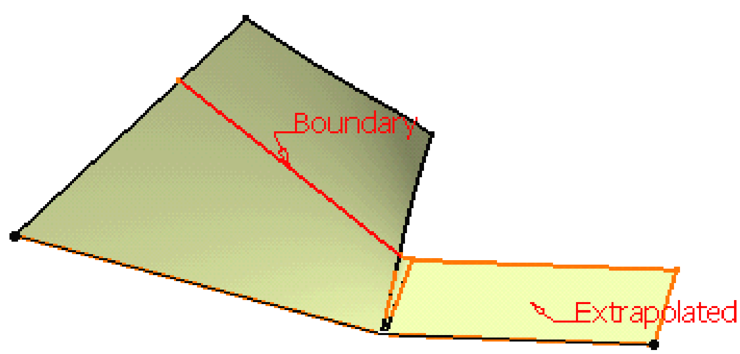
   - entering the value of the extrapolation length
   - selecting a limit surface or plane
   - using the manipulators in the geometry.

5. Specify the **Continuity** type:

   - **Tangent**
   - **Curvature**



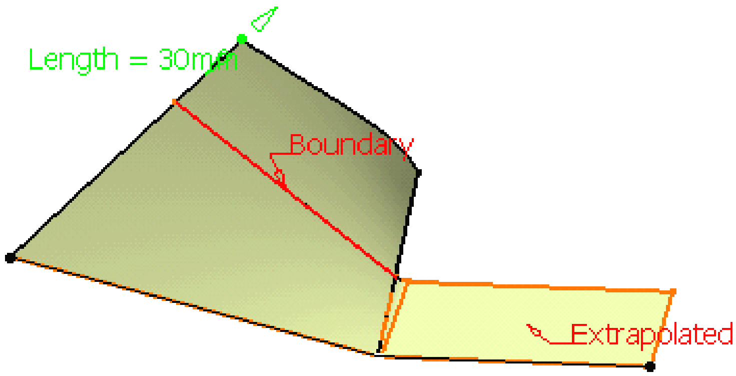
Tangent



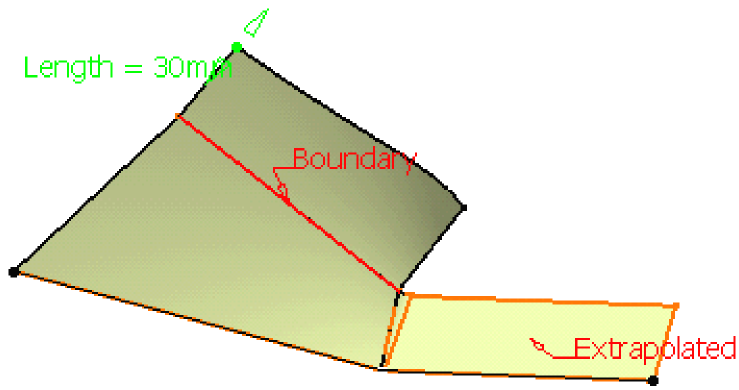
Curvature

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

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



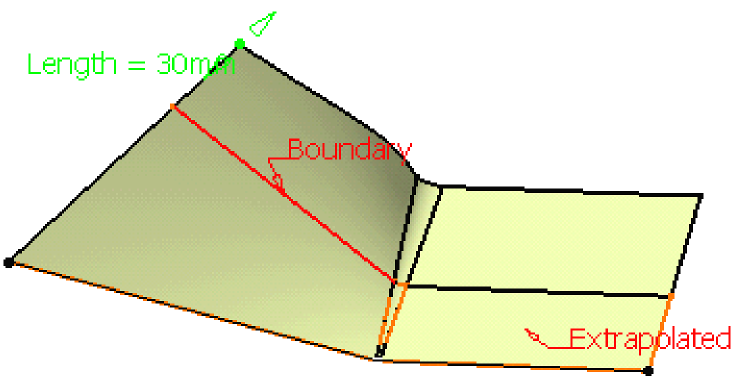
Tangent



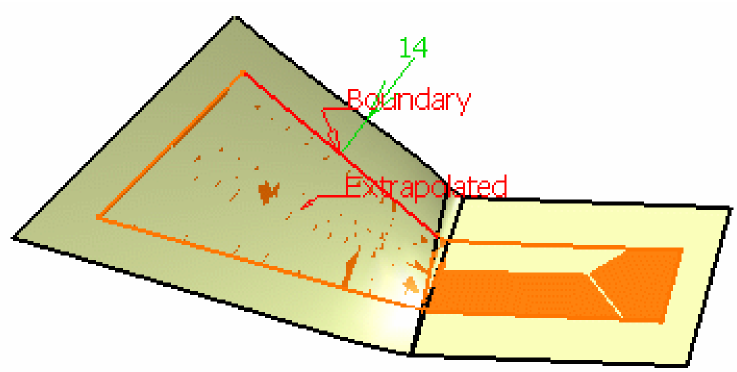
Normal

7. Specify the **Propagation** type:

- **Tangency continuity** to propagate the extrapolation to the boundary's adjacent edges.
- **Point continuity** to propagate the extrapolation around all the boundary's vertices.



Tangent continuity



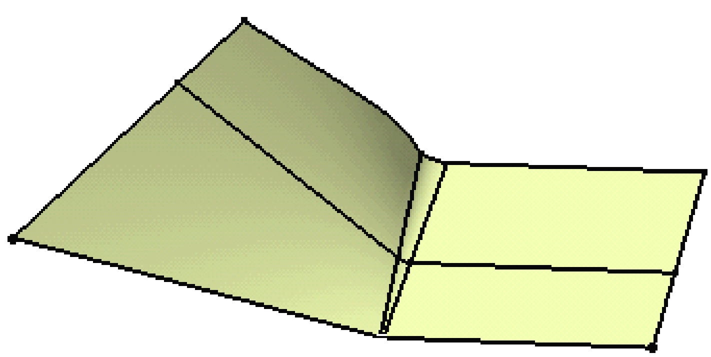
Point continuity

8. Check the **Assemble result** option if you want the extrapolated surface to be assembled to the support surface.

This option is now also available with the Curvature continuity.

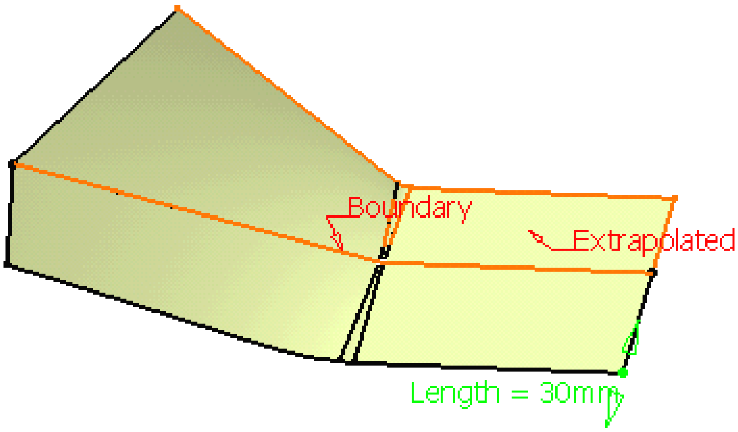
9. Click **OK** to create the extrapolated surface.

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

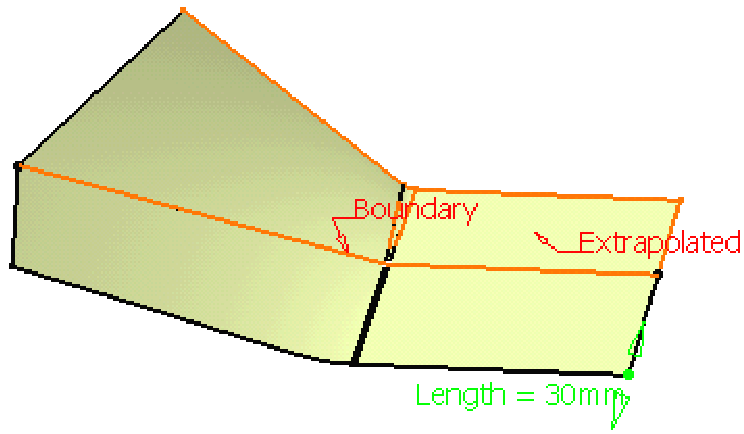


The **Internal Edges** option enables to determine a privileged direction for the extrapolation. You can select one or more edges (in the following example we selected the edge of Surface.1) that will be extrapolated in tangency. You can also select a vertex once you have selected an edge in order to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

You can only select edges in contact with the boundary.



No edges selected



One selected edge

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

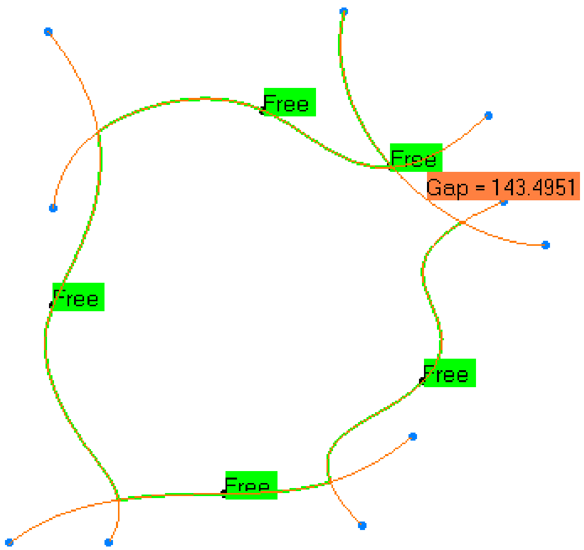
You can extrapolate several elements at a time. In this case, refer to Editing a List of Elements to find out how to display and manage the list of selected elements.

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

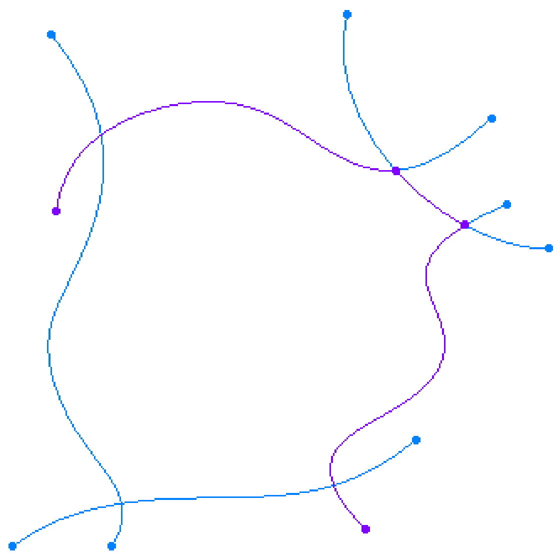
Slicing curves

This task will show you how to slice curves or edges.

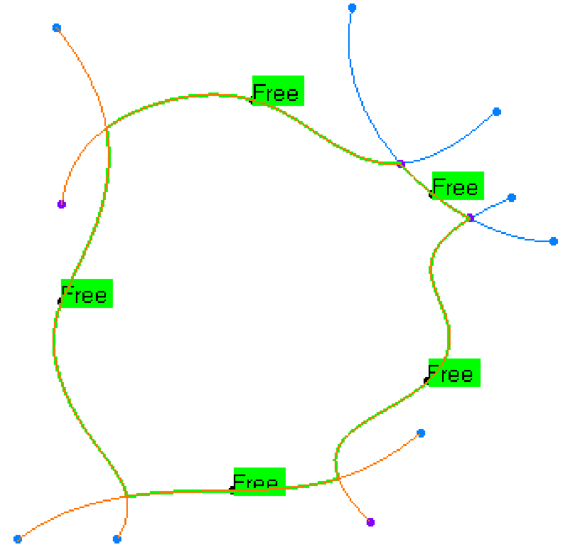
(The color of the curves have been changed to several shades of blue in the picture below).
The clean contour action sets the chaining order of the curves to create a contour. In some cases (especially with long curves) the chaining may lead to an unexpected result. You may need to slice curves or edges in order to solve this chaining incompatibility.



original curves: clean contour impossible



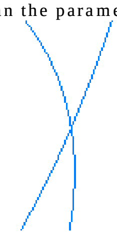
sliced curves



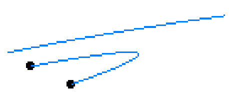
clean contour

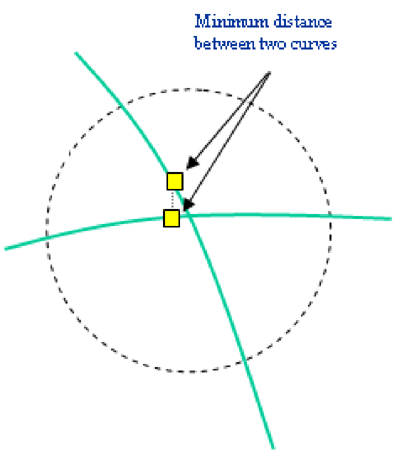
The Curves Slice action cuts curves or edges in several pieces, according to a pseudo-intersection: there is a pseudo-intersection between two curves if they intersect each other in the view direction (but not really), and if the mini 3D distance between them at this cutting point is lower than the parameter **Max. Distance**.

Pseudo-intersection of two curves in the view direction



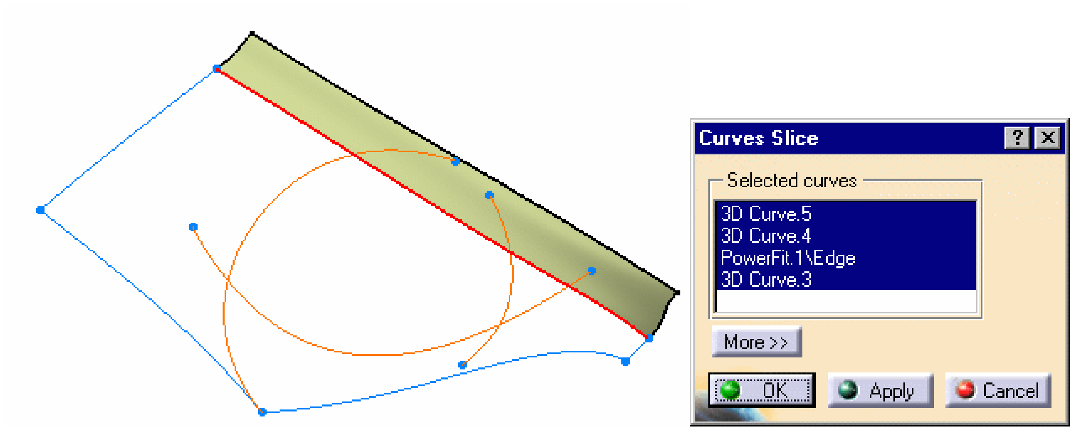
in another view.



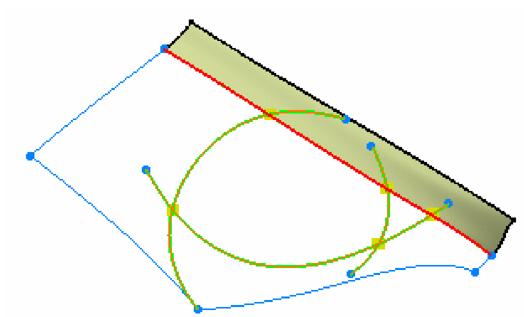


Open the Slice1.CATPart from the samples directory.

1. Activate the [illegible] set using the **Define in Work Object** contextual menu.
2. Click the **Curves Slice** icon . The **Curves Slice** dialog box is displayed.
3. Select the curves or the edges to slice. The list of the selected curves or edges is displayed in the **Selected curves** field. You can remove an element from this list by selecting it again. You can then resume the selection of the elements.

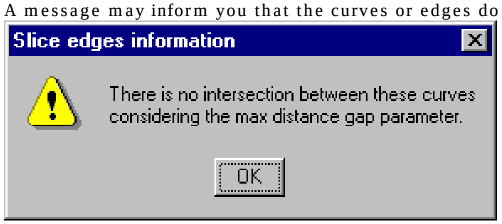


4. Click **Apply**. The resulting segments and the cutting point on each curve are pre-visualized.

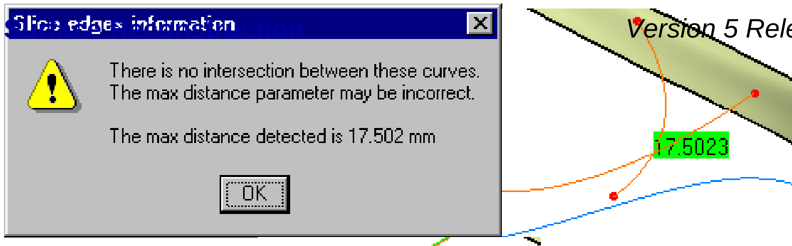


The segments smaller than the **Min length** set are not displayed.

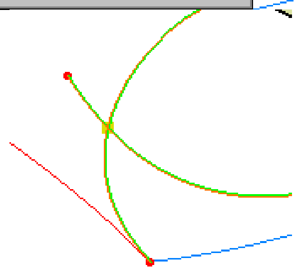
A message may inform you that the curves or edges do not "pseudo-intersect" each other with respect to the **Max distance** parameter.



These curves do not intersect each other, whatever the **Max distance** parameter. There is no solution.

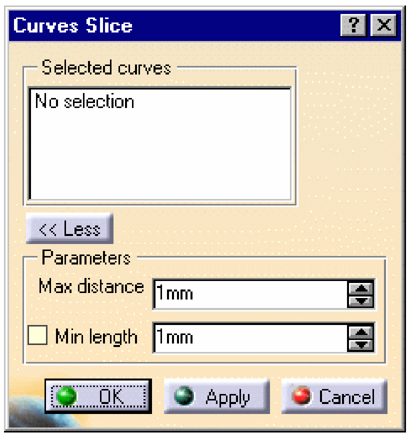


The curves will intersect each other if you set the Max distance parameter to a value higher than that indicated in the message and in the label.



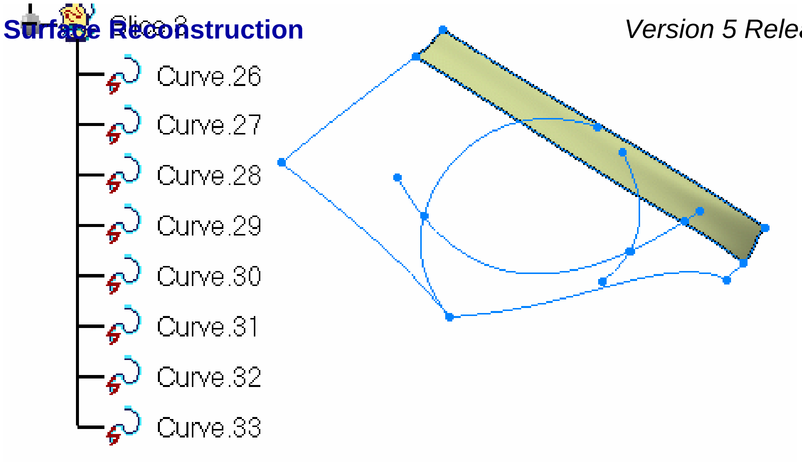
There is no information message, no distance label, only a yellow square: the curves do intersect each other.

5. Click **More >>** to display that parameter: Its default value is 1mm. **Max distance** is the minimum orthogonal distance between two curves above which one considers the two curves do not "pseudo-intersect" each other. Increase this value according to your needs (25 mm in our example).



6. Very tiny segments may be created. To avoid it, check the **Min length** box, and set the **Min length** (i.e. the minimum length of the segments created) value according to your needs.

7. Click OK to validate the result. The input curves are sent to the NoShow, A new body Slice.x is created in the specification tree, under the current working body, containing the segments created.


Curve.26
Curve.27
Curve.28
Curve.29
Curve.30
Curve.31
Curve.32
Curve.33

Adjust Nodes

It is difficult to re-create surfaces from a network of curves and to make sure that they are perfectly continuous. This task shows how to improve a node of the network to this purpose:

- the action modifies a set of curves arriving at a same node so that they have the same extremity (G0 continuity) and the same tangent plane at this node (G1 continuity).
- this ensures that all surfaces build on each mesh arriving at this node are continuous in tangency.

The curves may be simple curves, 3D curves, face edges...

Curves are deformed :

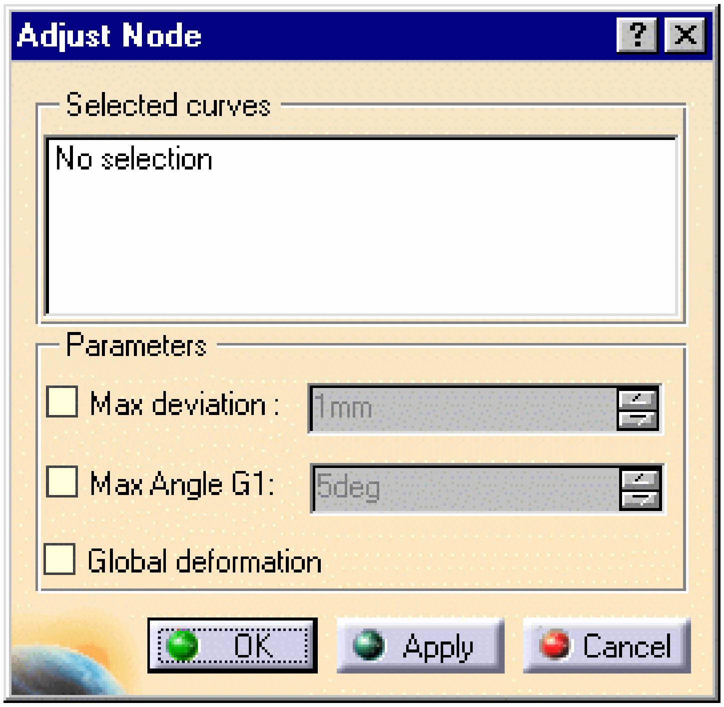
- to be made G0 continuous (passage continuity) in all cases,
- to be made G1 continuous (tangency continuity) according to your needs (for example, you do not want to apply a tangency continuity constraint on a sharp edge).

It is also possible to make two curves tangent to each other. Only the curves constrained to the tangent plane are made tangent to each other. The curves set to **Continuous** only can not be made tangent.

The G0 continuity can be tuned up using the **Maximum deviation** parameter i.e. the maximum distance between the input curve and the deformed curve.

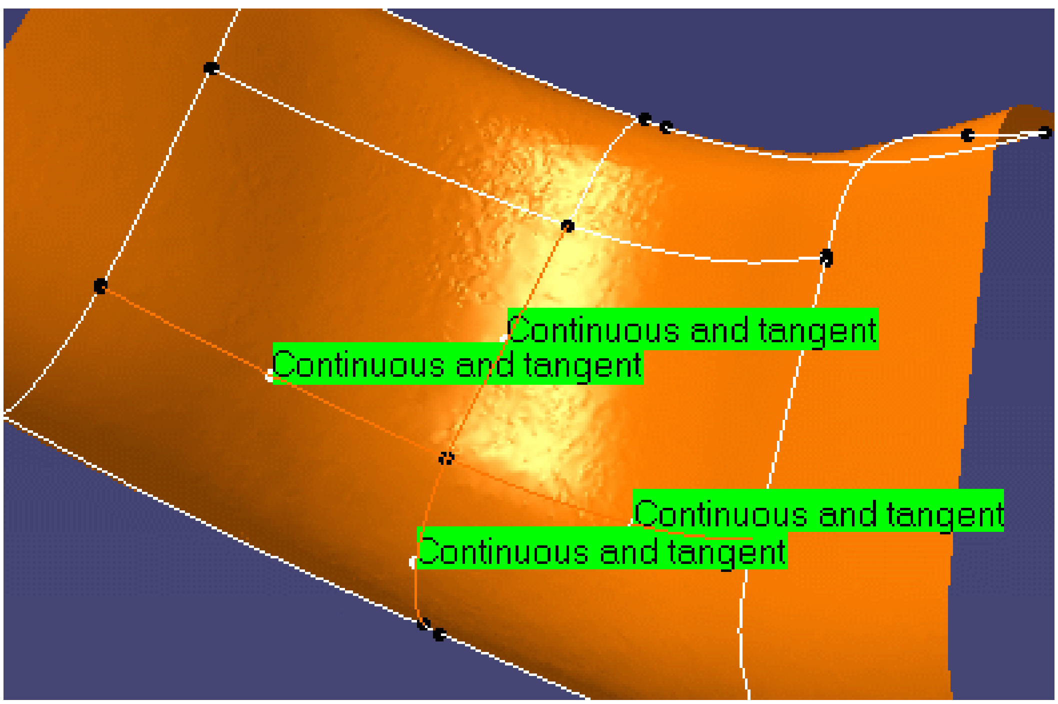
The G1 continuity can be tuned up using the **Max Angle G1** parameter, below which curves are made tangent to each other.
Open the AdjustNode01.CATPart from the samples directory.

1. Define an In Work Object.
2. Click the **Adjust nodes** icon . The **Adjust Node** dialog box is displayed:



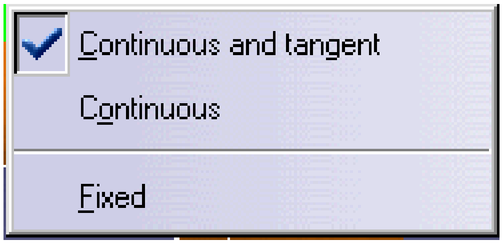
3. Select the curves to adjust. They are listed in the **Selected Curves** field.

You can deselect a curve by picking it, or select another curve.



By default, the curves are "**Continuous and tangent**"

Click of the label of a curve, or right-click to launch its contextual menu, to change its status.



The available statuses are:

Continuous and tangent: the deformed curves are G0 continuous and tangent to the computed tangency plane.

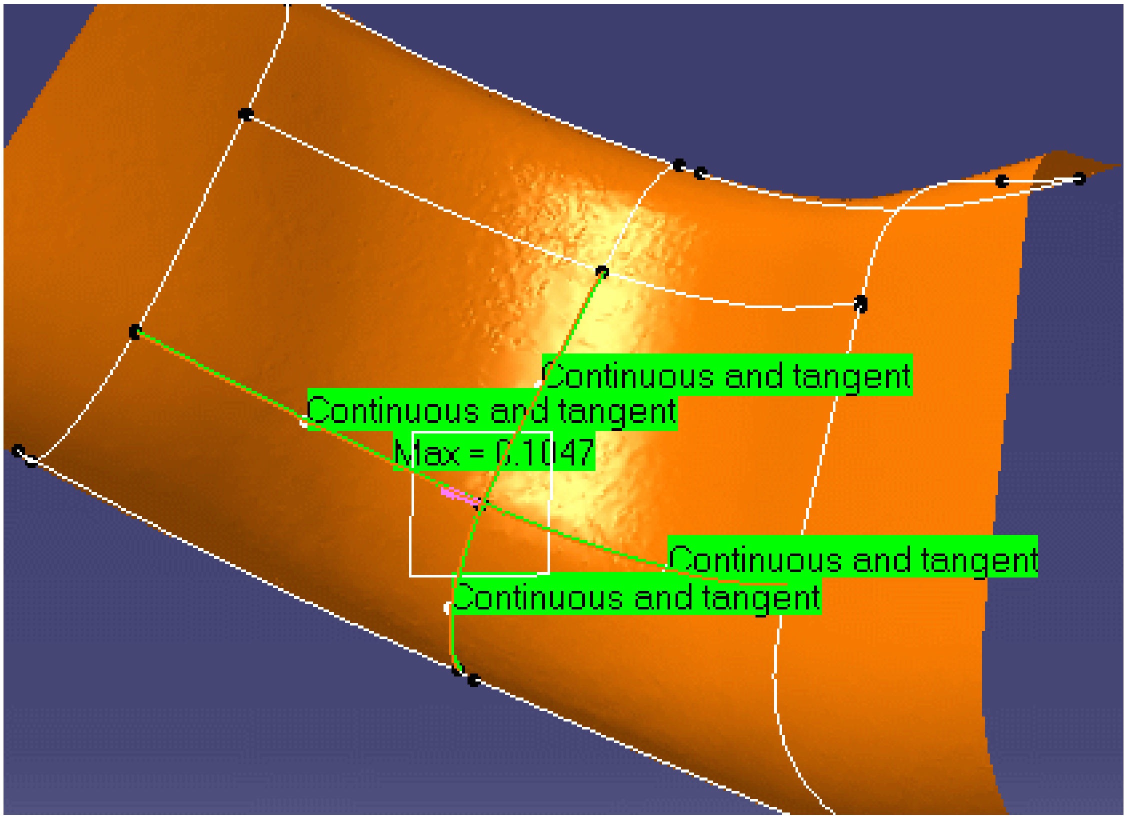
Continuous: the deformed curves are only G0 continuous.

Fixed: the curves are not deformed. In particular, face edges are fixed and remain fixed.

4. Click Apply. The deformations are computed:

- The curves selected are highlighted,
- The curves modified are displayed in green,
- The tangency plane is displayed in white,
- Its vector are displayed in pink,

The maximum deviation is also displayed on the most deformed curve, not necessarily on that deformation spot.



The curves are deformed to reach the required continuity. By default, this deformation is local, meaning the curves are deformed at their extremities. You can check the **Global deformation** option to distribute the deformation more evenly on the whole curves. The degree and the structure of the curves are kept.

The result may not be satisfactory. In that case, you can:

- Check the **Max deviation** option and enter the value of the maximum allowed deviation. At the apply, the deformation of the curves is computed and displayed. If the deformation is greater than the requested tolerance, the value of the maximum deformation is displayed in red, with an error message. This is only an information, the node can be created.

- Check the **Max Angle G1** option to force a tangency constraint on the curve end points when the angle of the tangents at those ends is lower than the **Max Angle G1** value.

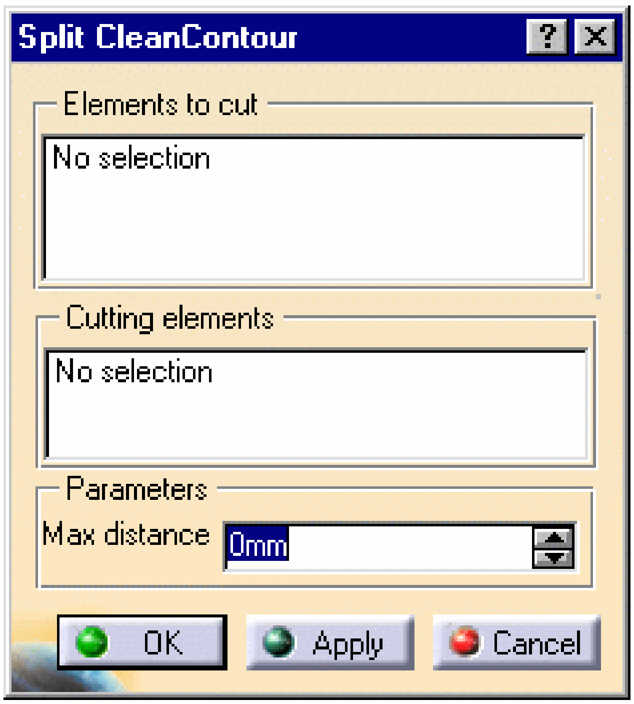
- This action processes one node at a time.
- There must be more than one curve to adjust.
- Several non continuous fixed curves may lead to inconsistencies.
- If some curves are deformed too far apart with respect to the tolerance, the adjustment cannot be computed and an error message is displayed.

Splitting CleanContours

This task will show you how to split a closed CleanContour by a curve. The output is two open CleanContours.

This may be necessary to create satisfying PowerFit surfaces: when the surface created from a polygon and a CleanContour does not respect the accuracy requested, splitting the CleanContour in two and creating two surfaces may be the solution.
Open the SplitCleanContour1.CATPart from the samples directory.

1. Click the CleanContour Split icon . The **Split CleanContour** dialog box is displayed.



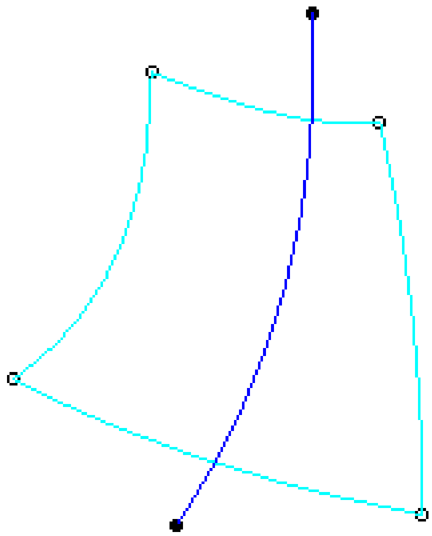
2. Select a CleanContour to split. The CleanContour must be closed. You can select it either by one vertex or by selecting its components one by one.

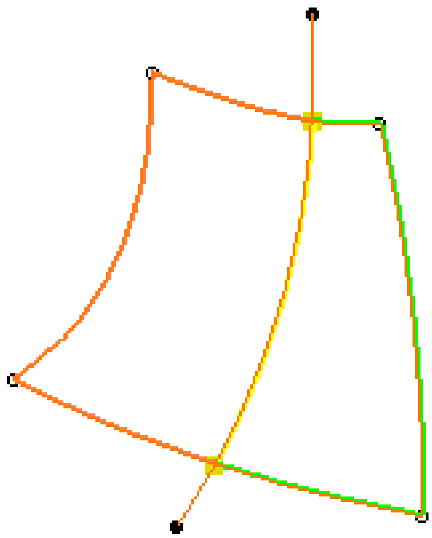
3. Go to the Cutting elements field and pick **No selection**. When it is highlighted in blue, select a or several cutting curves. There must be two and only two pseudo-intersections between the cutting curve(s) and the CleanContour.

   The name of the curves of the CleanContour and of the cutting curve(s) are displayed in the dialog box.

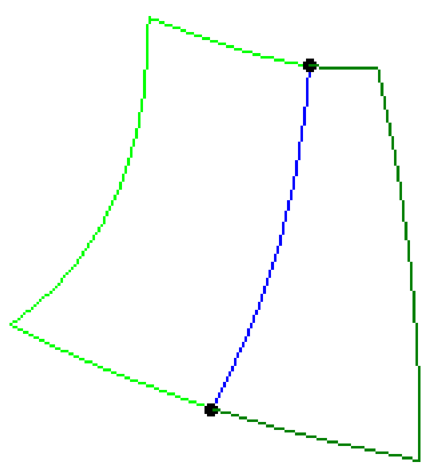
The **Split CleanContour** action cuts a CleanContour into two CleanContours, according to a pseudo-intersection: there is a pseudo-intersection between two curves if they intersect each other in the view direction, and if the mini 3D distance between them at this cutting point is lower than the parameter **Max. Distance**.

4. Click **Apply**. The split is displayed.





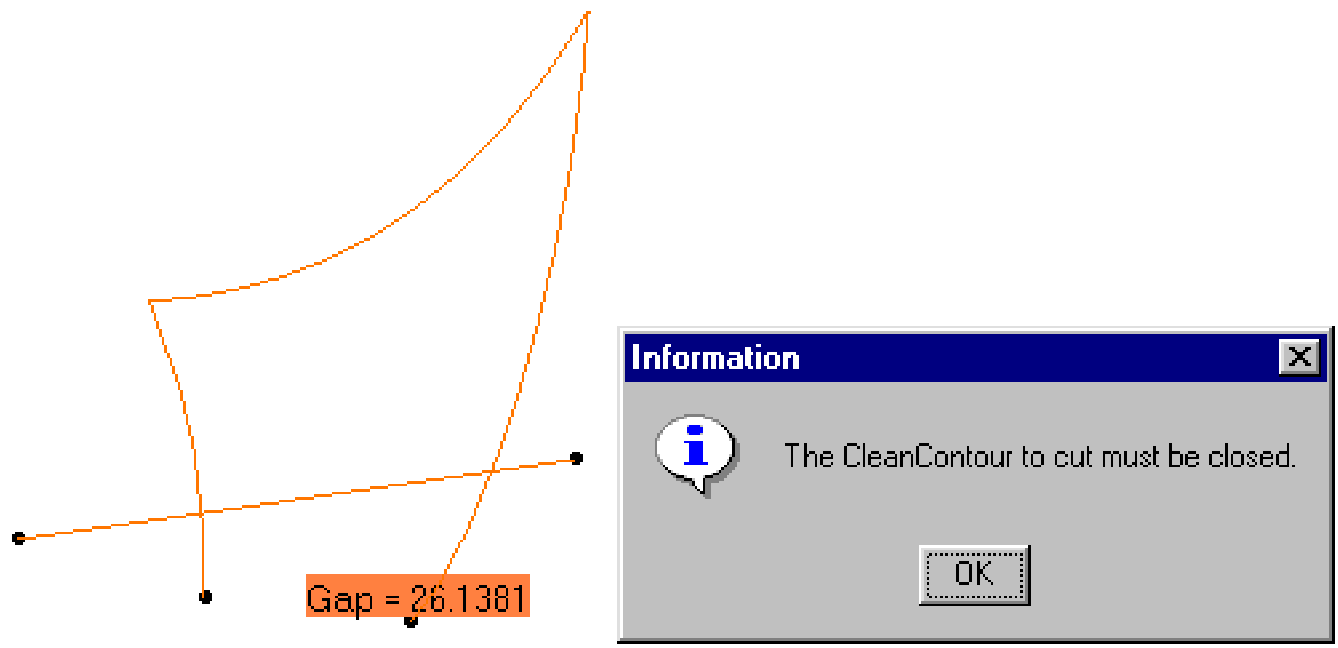
5. Click OK. The two open CleanContours (joins) are created, the curve it trimmed accordingly. A **split.x** element is created in the specification tree. It contains the two joins created.



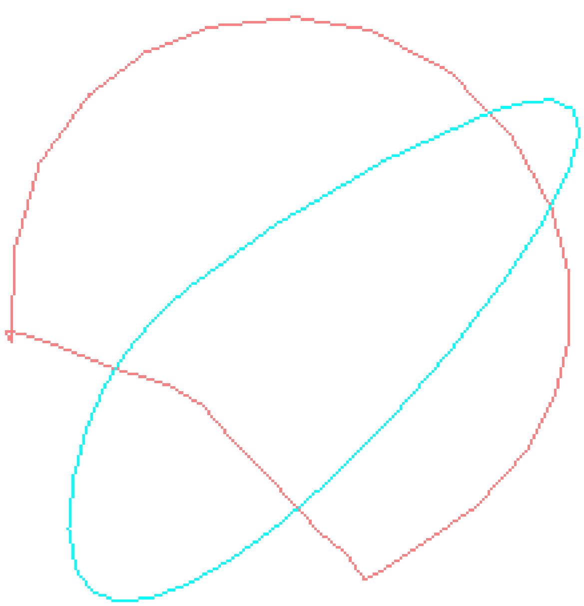
- You may want to select a join. If this join contains a sliced surface edge, or a split CleanContour that contains a sliced surface edge, with a tangency constraint that you want to keep, pick the curves one by one, graphically, i.e. do not select a join by picking one vertex, nor select the elements in the specification tree.

Possible problems

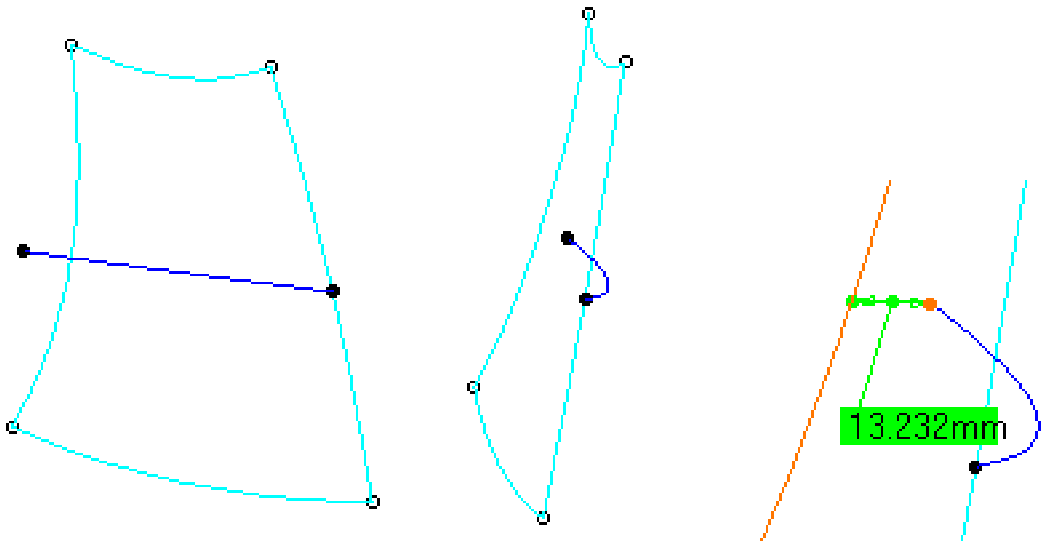
If the input CleanContour is not closed, the gap is displayed and no computation is started.



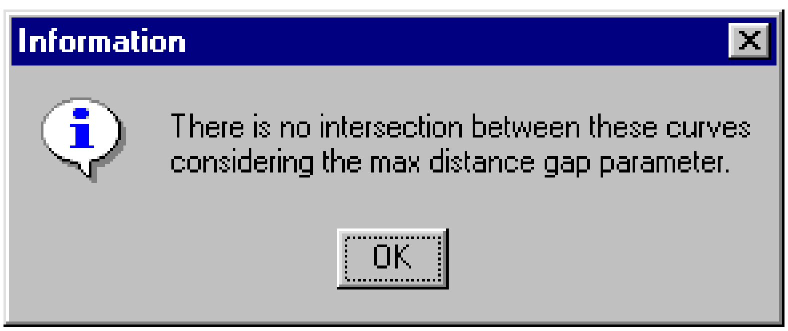
A closed CleanContour as a cutting element may cause an ambiguity. A message asks you which part of the cutting CleanContour you want to use.



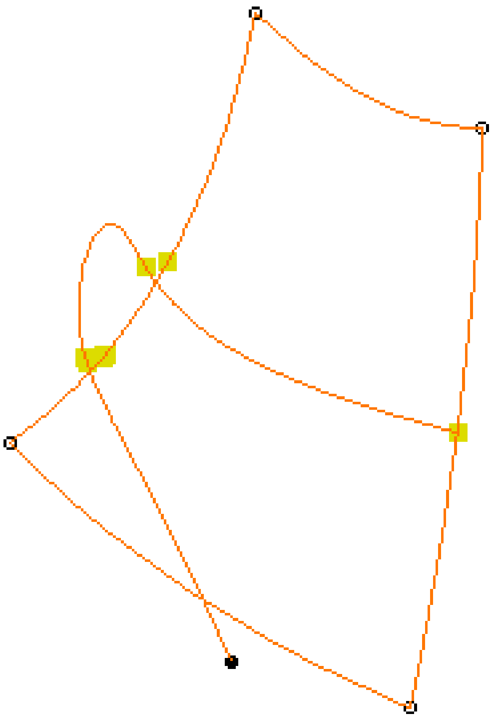
The **Max distance** parameter may be too low. Set a higher value and try again.

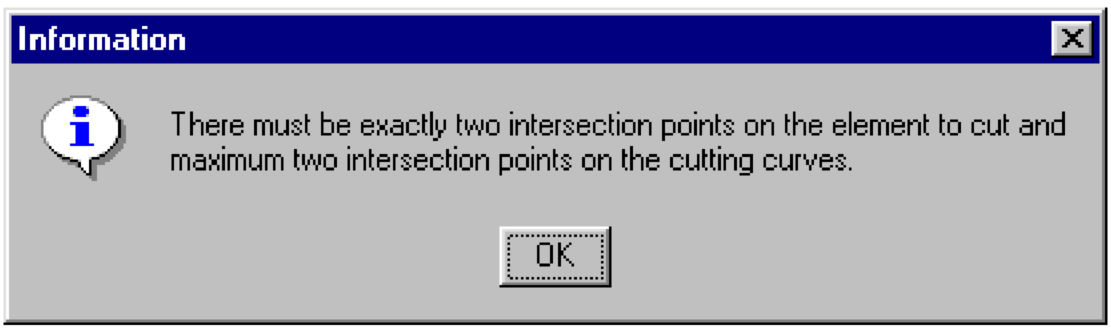


For example, the curve seems correct in the view direction, but the 3D distance is higher than the set Max distance (in our example, 1mm).



The cutting curve is not suitable. Modify it and try again.



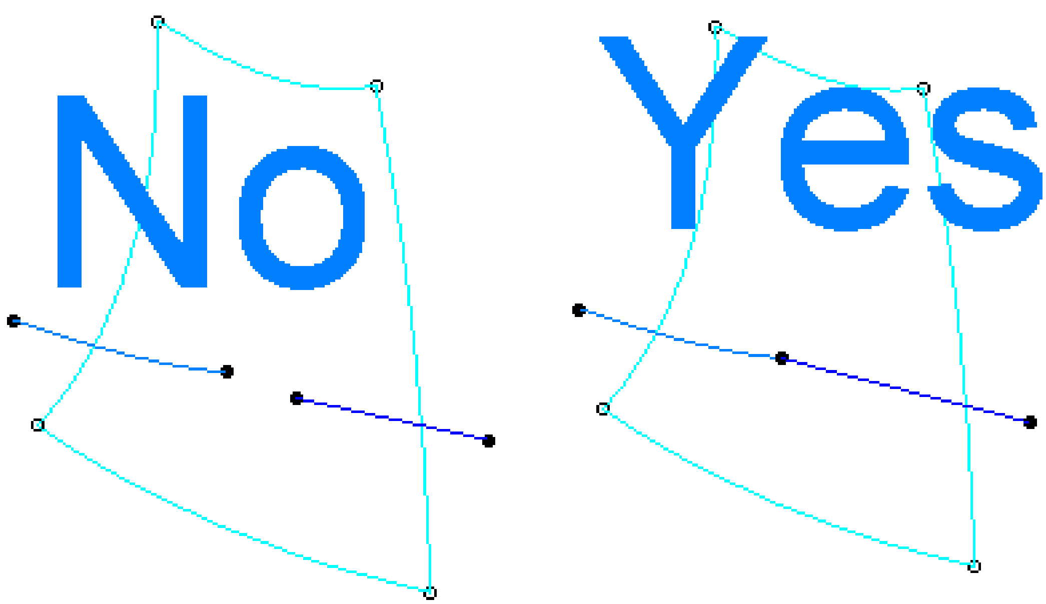


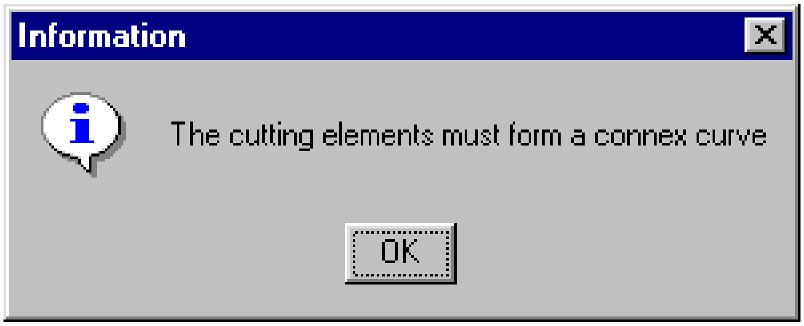
Information

There must be exactly two intersection points on the element to cut and maximum two intersection points on the cutting curves.

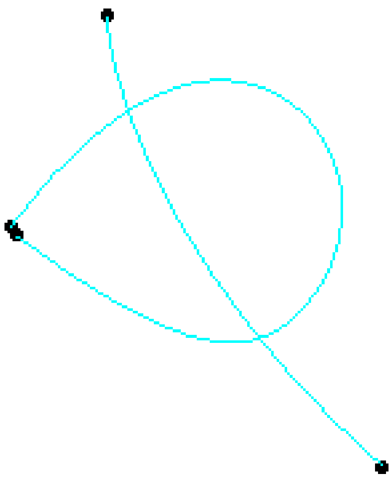
OK

You may enter several curves as cutting elements, but they must be connex:





It is not possible to split a CleanContour made of one element like this one



Creating Edge Fillets

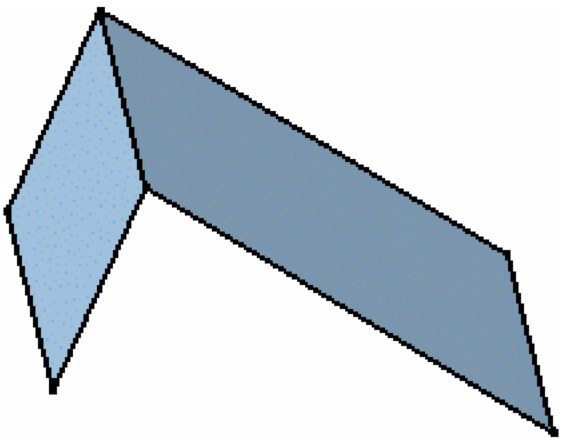
Edge fillets are useful to provide a transitional surface along a sharp internal edge of a surface.

This task shows how to create a constant radius fillet along the internal edge of a joined surface.
The fillet surface is obtained by rolling a sphere over the selected edge.

Open the EdgeFillet1.CATPart document.

1. Click the **Edge Fillet** icon .
2. Select the edge to be filleted.

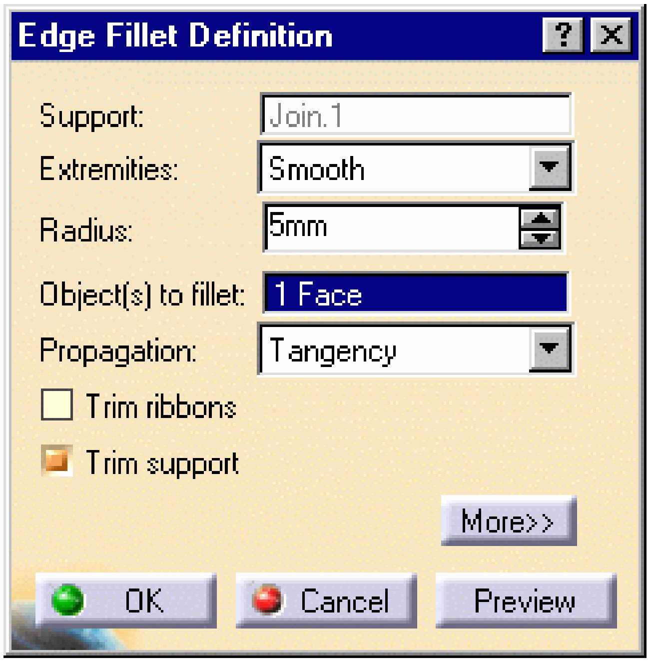
   You can also select a face, provided there is no ambiguity as to the edge(s) to be filleted.



The **Edge Fillet** Definition dialog box appears.

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

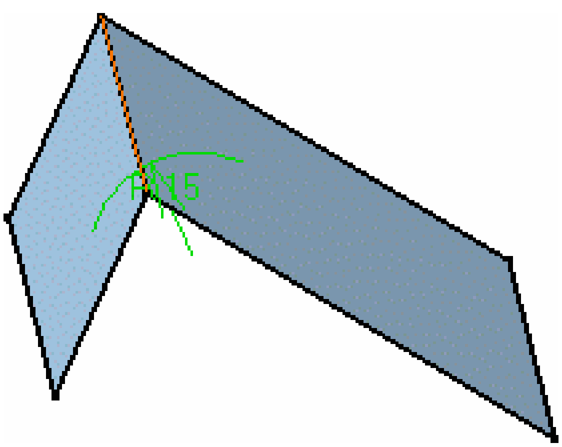
- **Straight:** no tangency constraint is imposed at the connecting point between the fillet and the initial support, generating sometimes a sharp angle.



- **Smooth:** a tangency constraint is imposed at the connection between the fillet surface and the support surfaces, thus smoothing the connection
- **Maximum:** the fillet surface is limited by the longest selected edge
- **Minimum**: the fillet surface is limited by the shortest selected edge

4. Enter the value of the fillet **Radius**.

   A preview of the fillet appears.



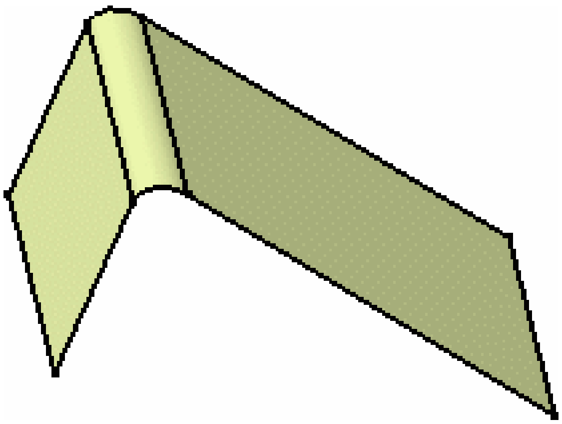
5. You can choose the **Propagation** type:

- **Tangency**: the fillet is propagated up to the first edge that is not continuous in tangency.
- **Minimal**: the fillet is propagated up to the first geometric limitation.

Use the More >> button to access further options: **Edge(s) to keep** and **Limiting element** and **Blend corner.**

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

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

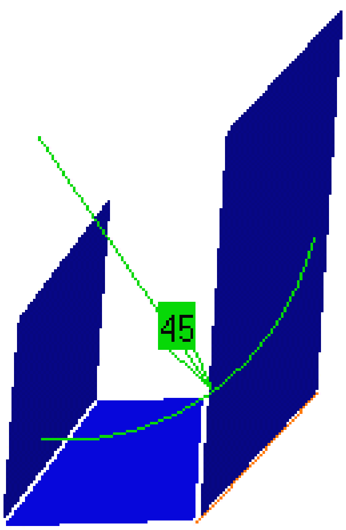


- Check the **Trim support elements** option to relimit the support elements and assemble them to the fillet.

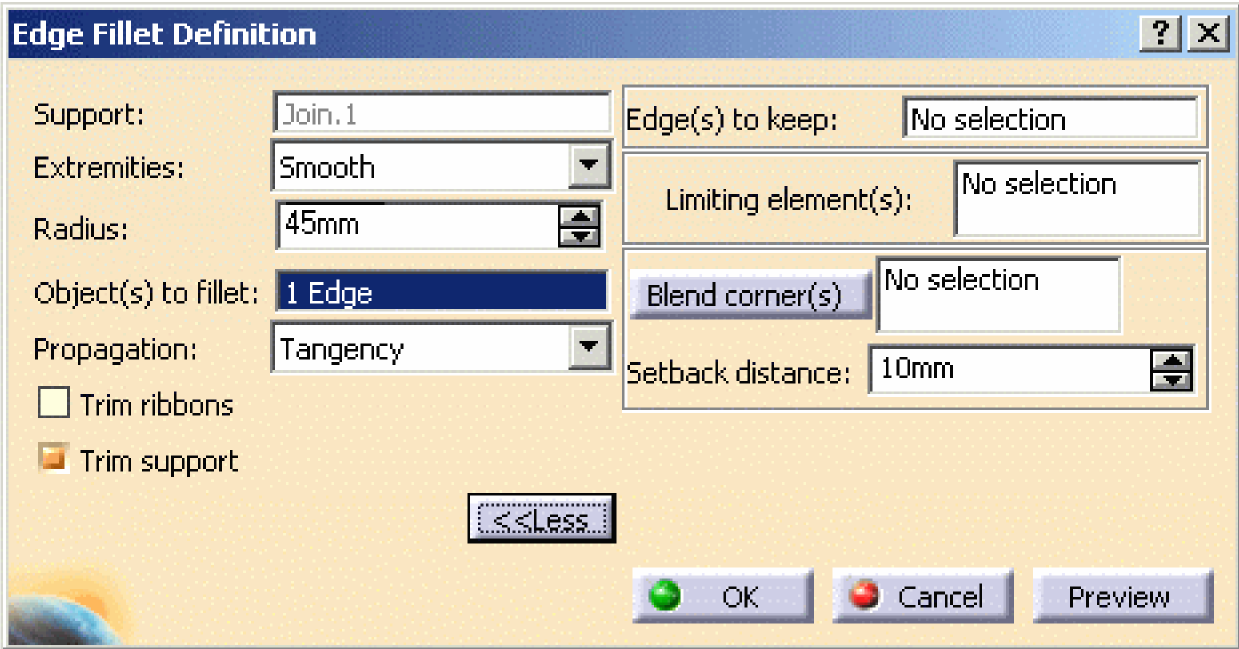
Keeping Edges

You may also need to explicitly indicate edges that should not be filleted, if a radius is too large for example. In this case you cannot select boundary edges to be kept, but only internal edges, i.e. edges limiting two faces.

Open the EdgeFillet2.CATPart document.



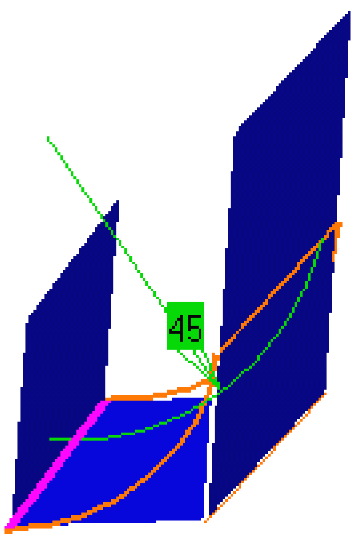
To do this, proceed as above, but once you have selected the edge to be filleted, click the **More** to expand the dialog box, then click the Edge(s) to keep field and select the edge you wish to keep.



If you have difficulties selecting the edge, use the up/down arrows to display the preselection navigator.

This edge is displayed in pink, meaning that it will not be affected by the fillet operation.

Then, click **OK** to create the fillet surface.



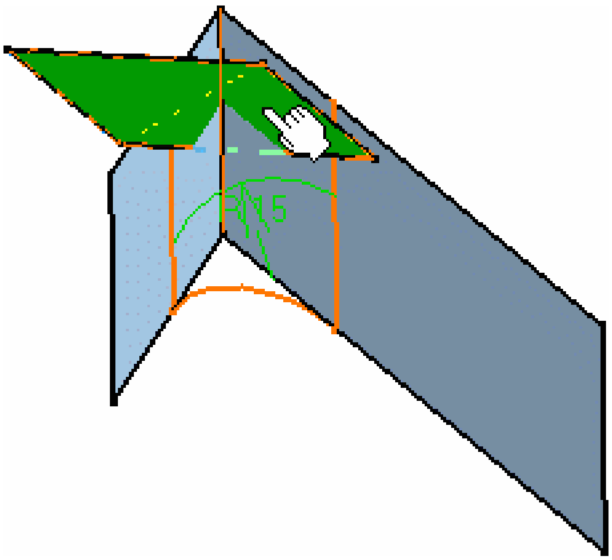
Limiting Fillets

While creating the fillet, you can limit it by selecting an element (plane or surface) that intersects it completely:

1. Once the edge to be filleted has been selected, and the radius keyed in, click **Preview** then the More button.

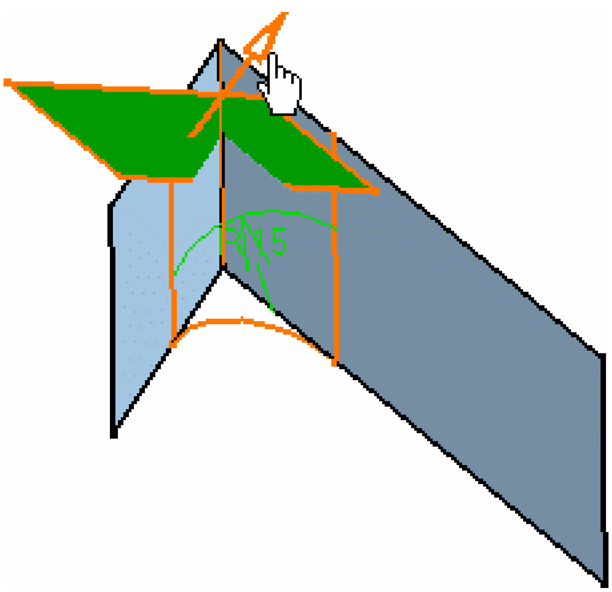
2. Click in the **Limiting element** field, then select the trimming element. These elements can be either surfaces, planes or points on edges.

   An arrow indicates which portion of the fillet is to be retained.



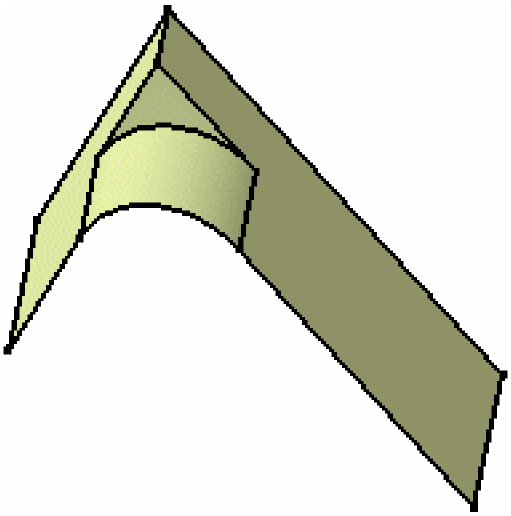
- It is now possible to use one or more limiting elements.
- You can define a limiting element just by clicking a point on one of the selected edges to be filleted.

3. Click on this arrow to inverse it, if needed, to retain the opposite side of the fillet.



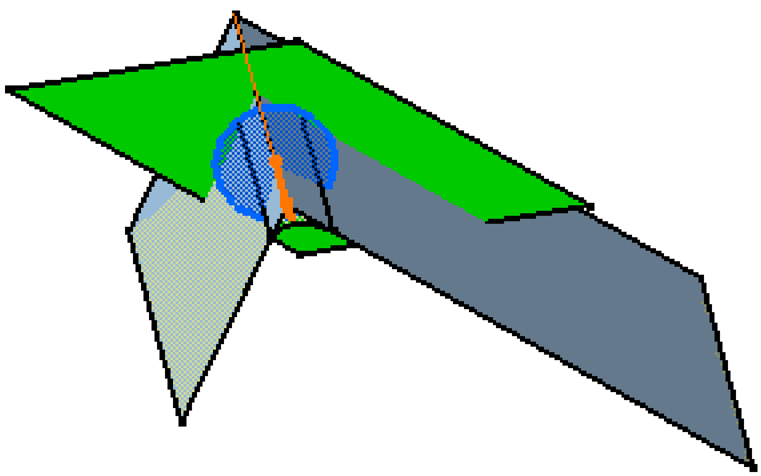
4. Click **OK** to create the limited fillet.

In the illustration, the limiting surface has been hidden.

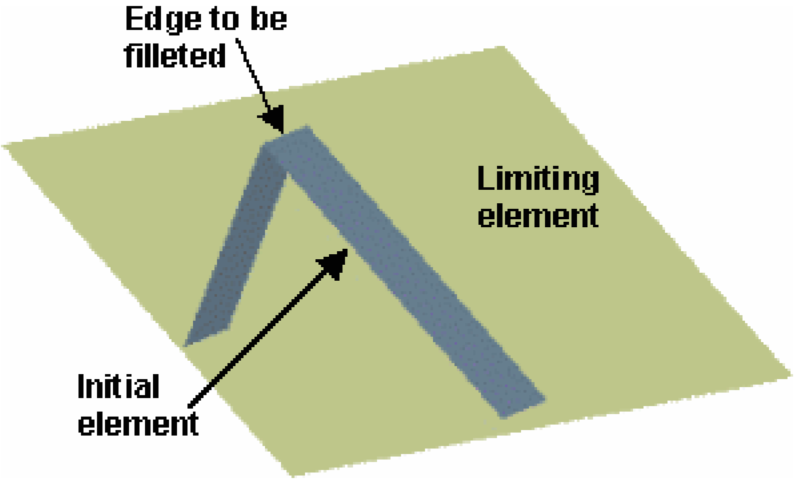


You can create limiting elements just by clicking on the edge to be filleted.

The application displays this element as a blue disk.



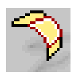
Make sure that the limiting element is not larger than the initial element, as illustrated here. In this case, decrease the size of the limiting element as prompted by the warning message.

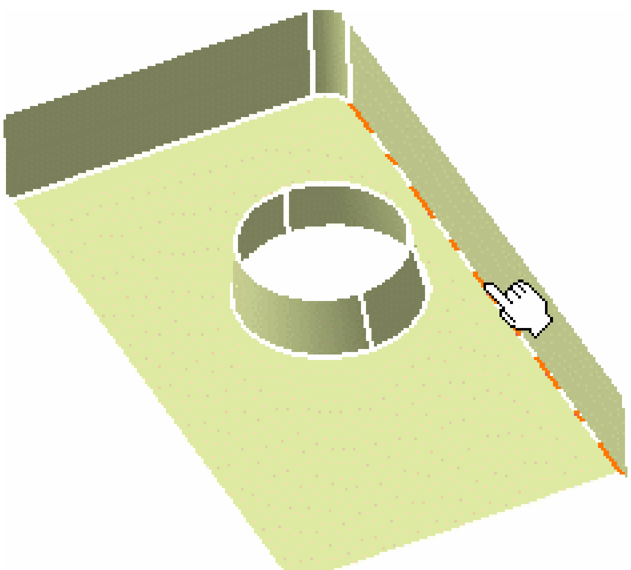


Trimming Overlapping Fillets

In some cases, fillets may be overlapping. The **Trim ribbons** option lets you solve this by trimming the fillets where they overlapping.

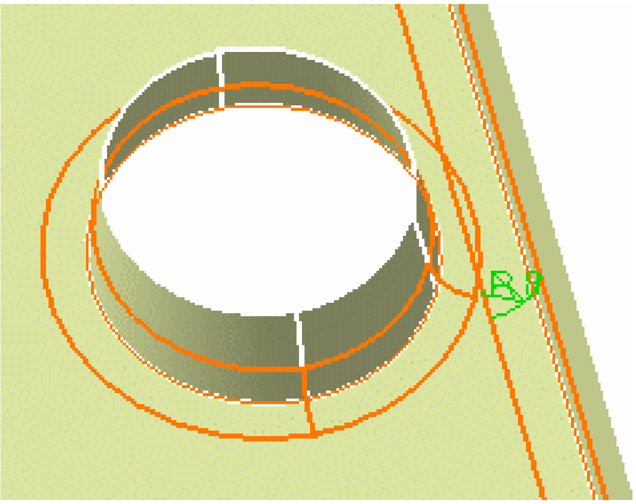
Open the EdgeFillet3.CATPart document.

1. Click the **Edge Fillet** icon  and, using the Ctrl key, select the edges at the base of the cylinder and the one along the vertical surface.

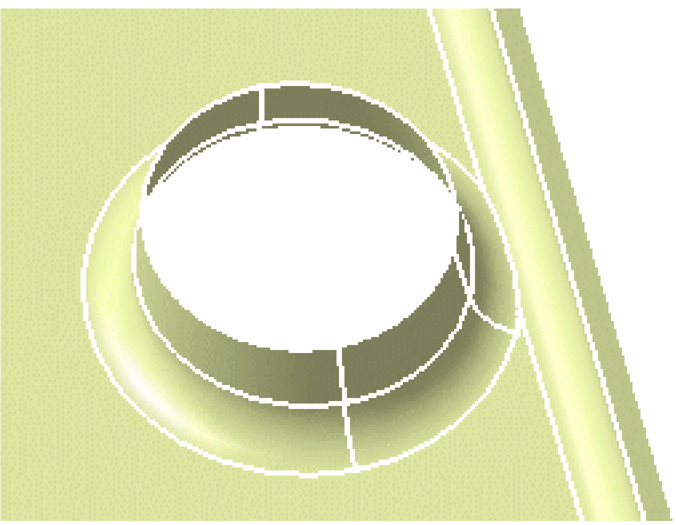


2. Click Preview.

The two fillets clearly overlap.

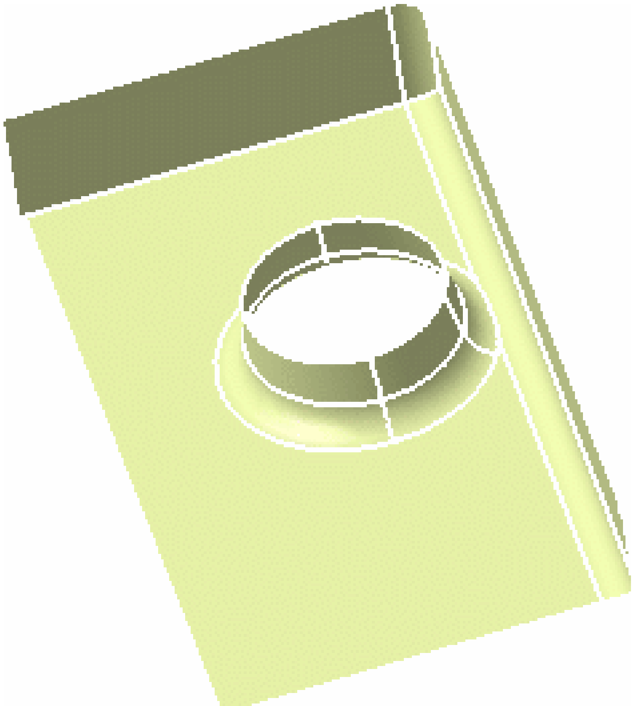


3. In the **Edge Fillet** Definition dialog box, check the **Trim ribbons** option and click OK.

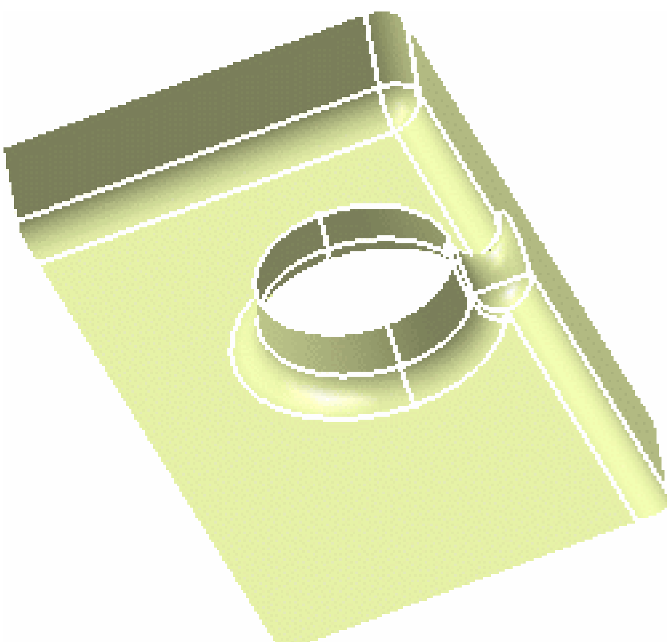


Note that the **Trim ribbons** option is available with the **Tangency** propagation mode:

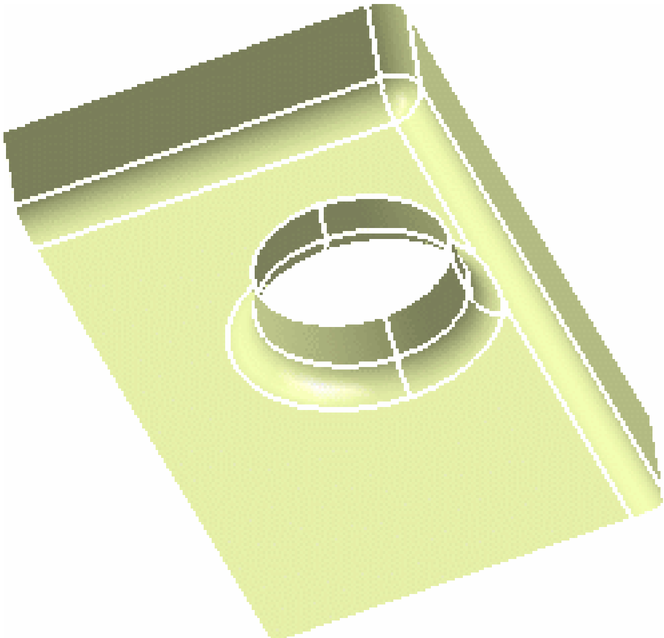
- In **Minimal** mode, the **Trim ribbons** option is grayed, as it is implicitly active. The results would be trimmed fillets, and no propagation:



- In **Tangency** mode, with the **Trim ribbons** option unchecked, the fillets intersect, with no trimming, and the propagation is performed



- In **Tangency** mode, with the **Trim ribbons** option checked, the fillets are trimmed and the propagation is performed.

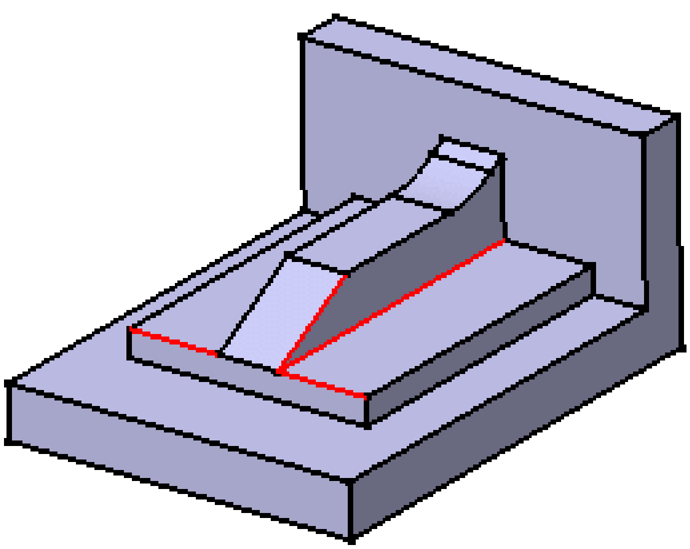


Reshaping corners:

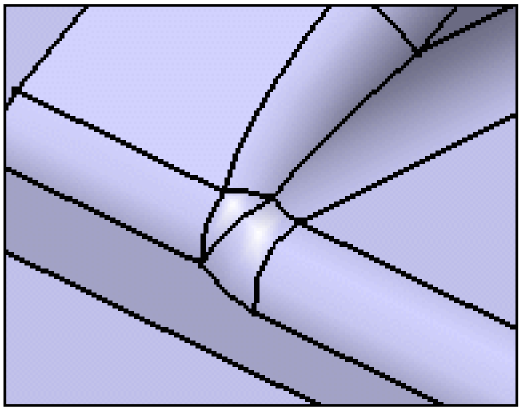
Sometimes, while filleting, you can see that corners resulting from the operation are not satisfactory. The new capability "**Blend Corners**" lets you quickly reshape these corners.

Open the BlendCorner1.CATPart document.

1. Click the **Edge Fillet** icon and fillet the edges as shown using 5mm as the radius value.

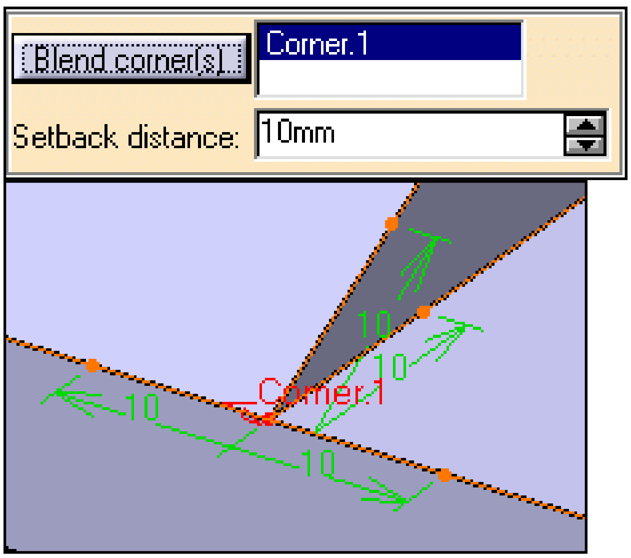


2. Taking a closer look at the corner, you can notice that the edges need to be rounded again.



3. After launching the **Edge Fillet** dialog box to edit the fillet, click the **More>>** button to access additional options.

4. Click the **Blend corner(s)** button to detect the corner to reshape. In our example, only one corner is detected. The application shows it in the geometry area (3D text).



When the application detects several corners, it is not possible to reshape just a few of them: all of them will be edited.

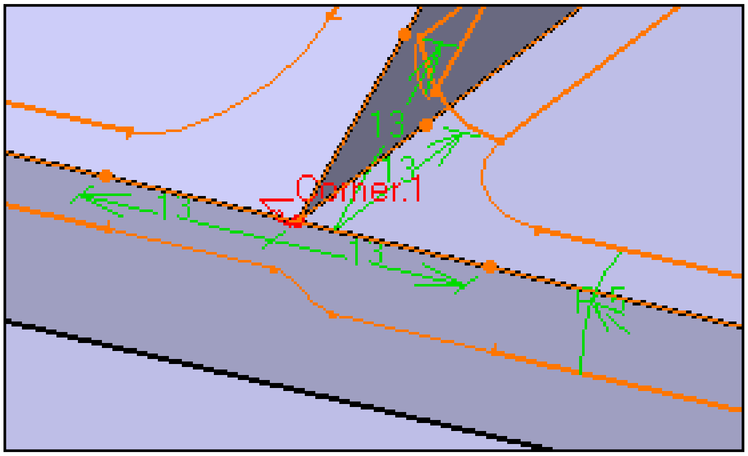
The setback distance field determines for each edge a free area measured from the vertex along the edge. In this area, the system adds material so as to improve the corner shape.

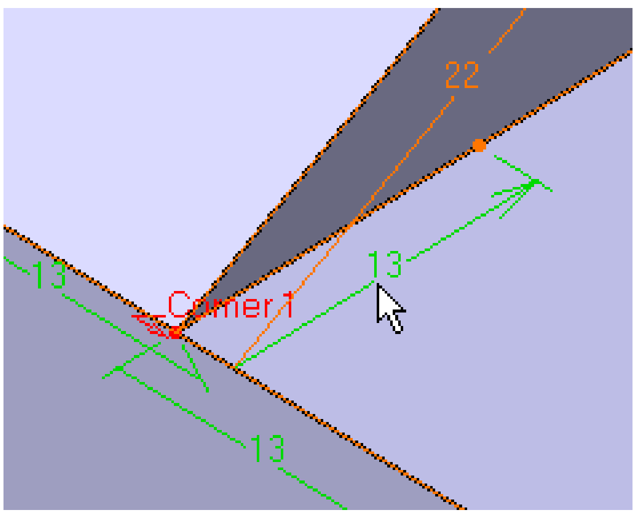
5. Enter a value in the setback distance field. For example 13.

6. Click **Preview** to examine the result.

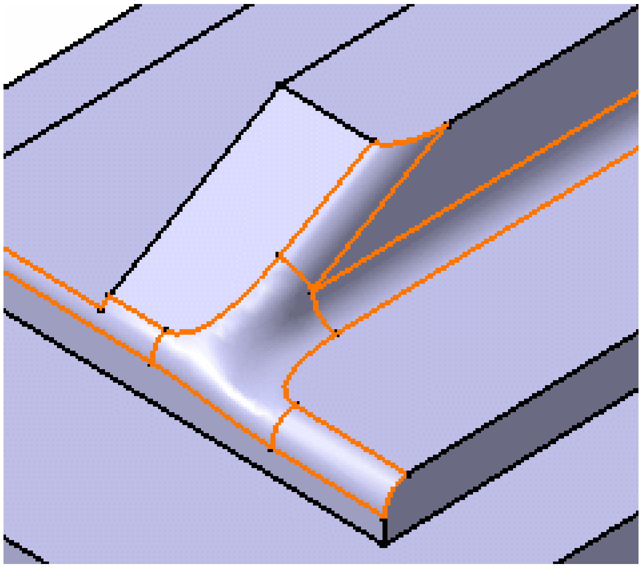
To edit the distance for the top edge, click "13" and enter "22" as the new value in the **Setback distance** field.

7. Repeat the operation for the edge below using the same distance value



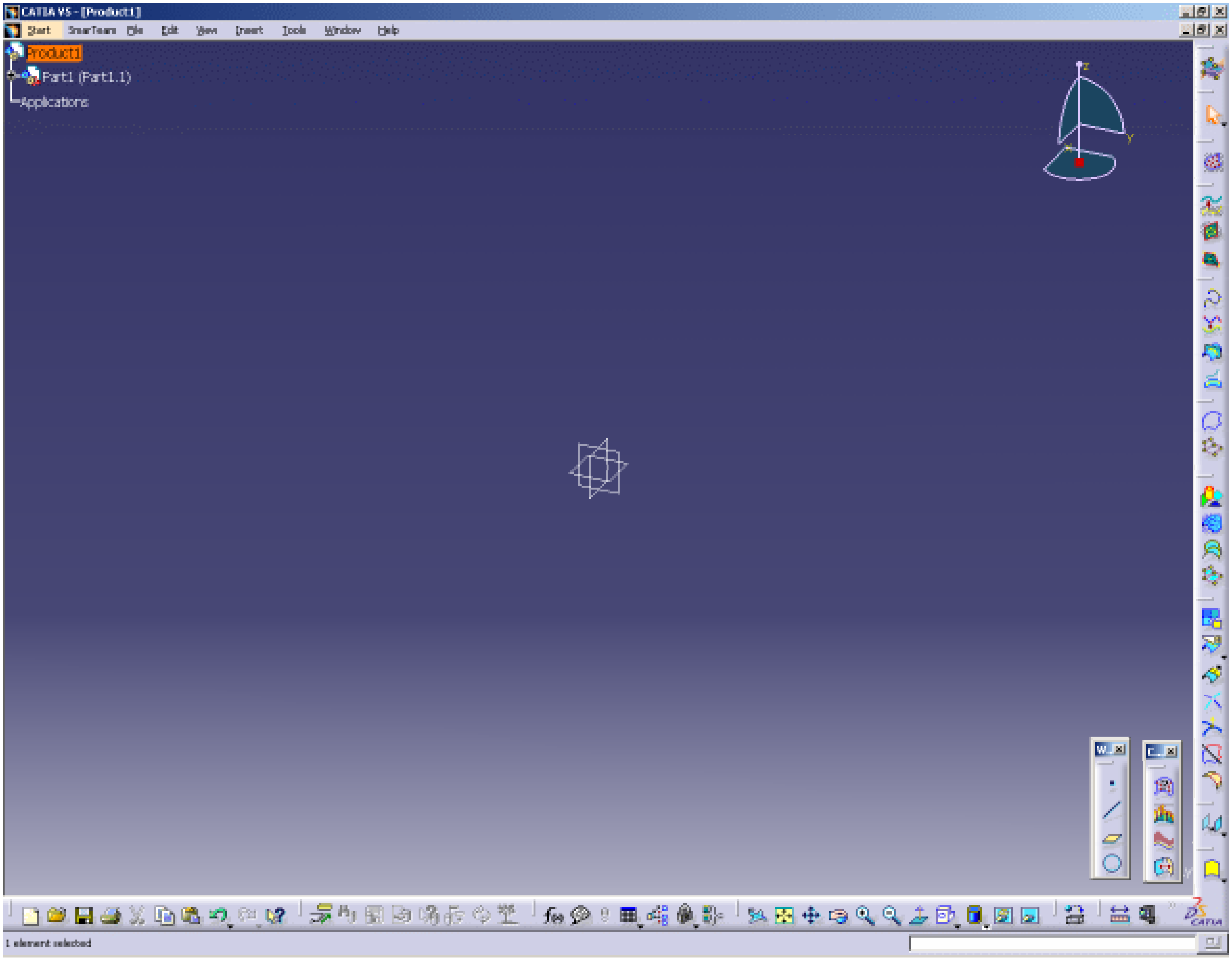


8. Click **OK** to confirm the operation. The corner is reshaped.



Workbench Description

This chapter describes the menus, sub-menus, items and toolbars of the Quick Surface Reconstruction.

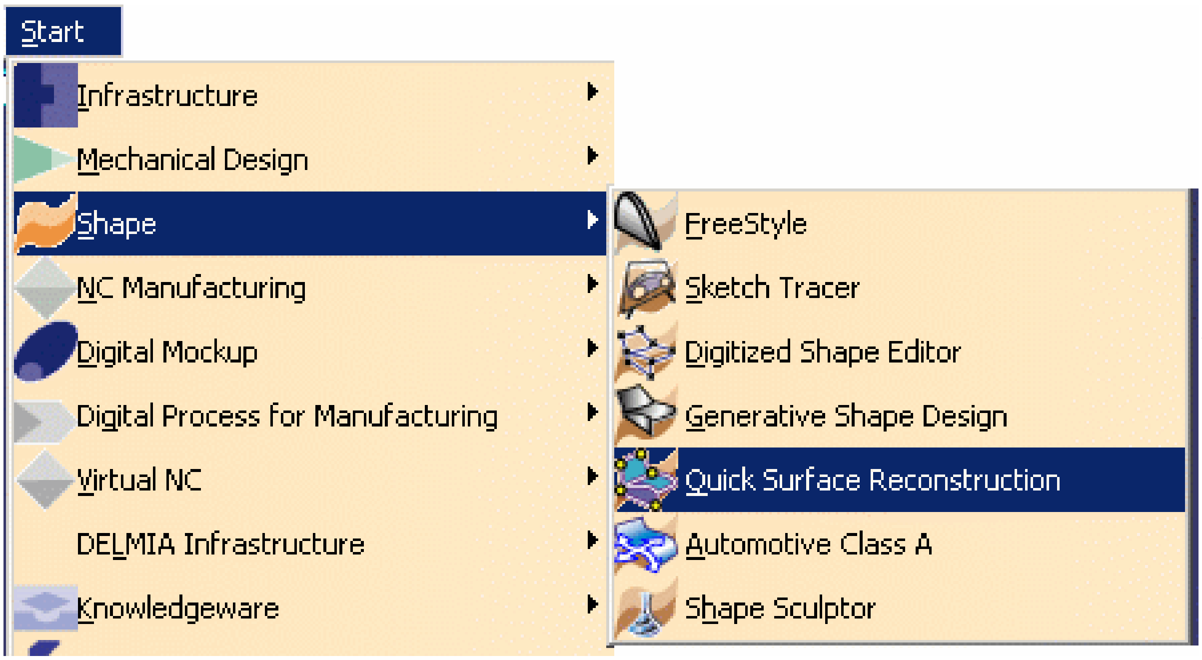


Menu Toolbar

This chapter describes the menus available in Quick Surface Reconstruction. Other menus are documented in the Infrastructure User's Guide.

Start	SmarTeam	File	Edit	View	Insert	Tools	Windows	Help

Start



Starts Quick Surface Reconstruction Workbench

Insert

Geometrical set	Managing Geometrical Sets
Cloud Edition	Cloud Edition
Scan Creation	Scan Creation
Curve Creation	Curve Creation
Domain Creation	Domain Creation
Surface Creation	Surface Creation
Operations	Operations
Transformations	Transformations
Segmentation	Segmentation

Analysis	Analysis
WireFrame	WireFrame

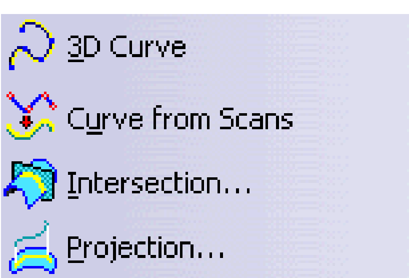
Cloud Edition

Activation	Activating a Portion of a Cloud of Points

Scan Creation

Curve Projection	Projecting Curves
Planar Sections	Cutting a Cloud of Points or a Polygon by Planar Sections
Free Edges	Creating Free Edges

Curve Creation

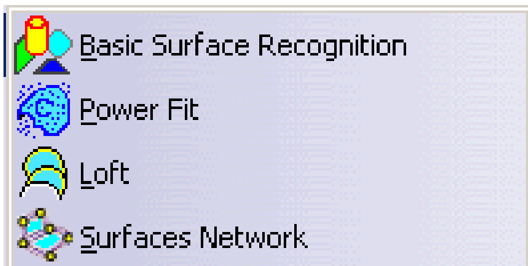


3D Curve	Creating Associative 3D Curves and Creating Associative 3D Curves on Scans
Curve from Scans	Curves from Scans
Intersection	Creating Intersections
Projection	Creating Projections

Domain Creation

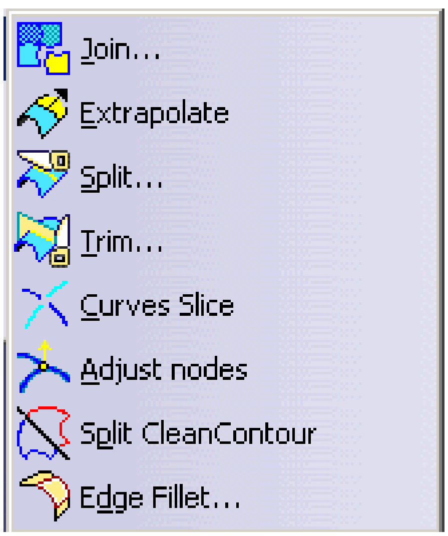
Clean Contour	Clean Contour Creation
Curves Network	Curves Network

Surface Creation



Basic Surface Recognition	Basic Surface Recognition
Power Fit	PowerFit
Loft	Creating Lofted Surfaces
Surfaces Network	Surfaces Network

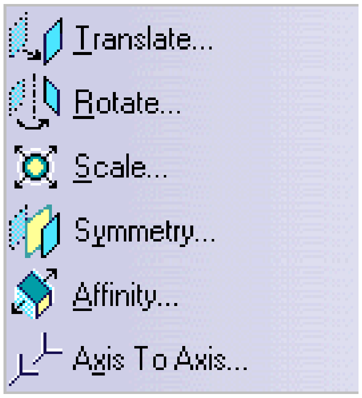
Operations



Join	Joining Surfaces or Curves
Extrapolate	Extrapolating Surfaces
Split	Splitting Geometry
Trim	Trimming Geometry
Curves Slice	Slicing curves
Adjust Nodes	Adjust Nodes
Split CleanContour	Splitting CleanContour
Edge Fillet	Creating Edge Fillets

Transformations

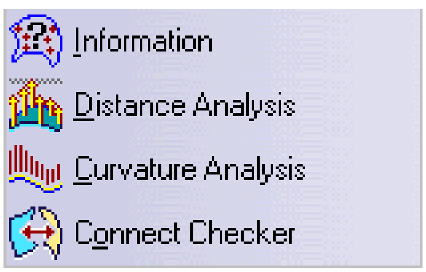
Translate	Translating Geometry
Rotate	Rotating Geometry
Scale	Transforming Geometry by Scaling
Symmetry	Performing a Symmetry on Geometry
Affinity	Transforming Geometry by Affinity

	Axis To Axis	Transforming Elements from an Axis to Another

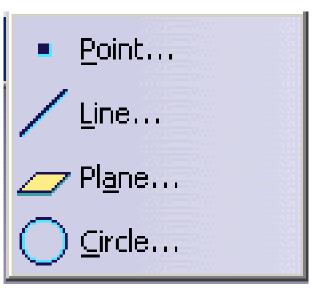
Segmentation

	Segmentation by Curvature	Segmentation by Curvature Criterion
	Segmentation by Slope	Segmentation by Slope Criterion

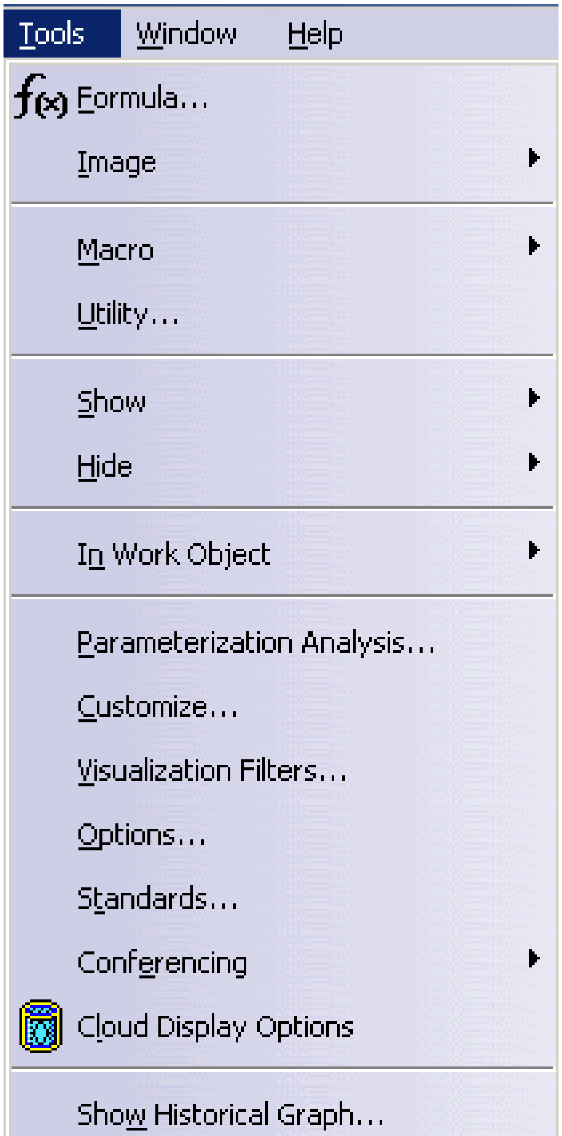
Analysis

	Information	Information
	Distance Analysis	Analyzing Distances Between Two Sets of Elements
	Curvature Analysis	Performing a Curvature Analysis
	Connect Checker	Checking Connections Between Surfaces

WireFrame

	Point	Creating Points
	Line	Creating Lines
	Plane	Creating Planes
	Circle	Creating Circle

Tools

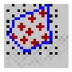


	Cloud Display Options	Display Options and Graphic Properties

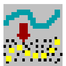
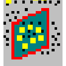
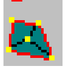
Creation Toolbars

This chapter describes the menus available in Quick Surface Reconstruction. Other menus are documented in the Infrastructure User's Guide.

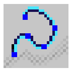
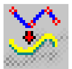
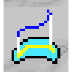
Cloud Edition

	For	See
	**Activate**	Activating a Portion of a Cloud of Points

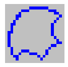
Scan Creation

	For	See
	**Project Curves**	Projecting Curves
	**Planar Sections**	Cutting a Cloud of Points or a Polygon by Planar Sections
	**Free Edges**	Creating Free Edges

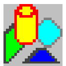
Curve Creation

	For	See
	**3D Curve**	Creating Associative 3D Curves and Creating Associative 3D Curves on a Scan of Cloud
	**Curves from Scans**	Curves from Scans
	**Intersection**	Creating Intersections
	**Projection**	Creating Projections

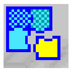
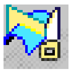
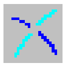
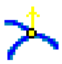
Domain Creation

	For	See
	**Clean Contour**	Clean Contour Creation
	**Curves Network**	Curves Network

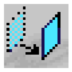
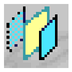
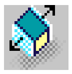
Surface Creation

	For	See
	**Basic Surface Recognition**	Basic Surface Recognition
	**PowerFit**	PowerFit
	**Loft Surface**	Creating Lofted Surfaces
	**Surfaces Network**	Surfaces Network

Cloud and Curve Operations

	For	See
	**Join**	Joining Surfaces or Curves
	**Extrapolate**	Extrapolating Surfaces
	**Split**	Splitting Geometry
	**Trim**	Trimming Geometry
	**Curves Slice**	Slicing Curves
	**Adjust nodes**	Adjust Nodes
	**Split CleanContour**	Splitting CleanContours
	**Edge Fillet**	Creating Edge Fillets

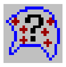
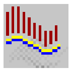
Transformations

	For:	See:
	**Translate**	Translating Geometry
	**Rotate**	Rotating Geometry
	**Symmetry**	Performing a Symmetry on Geometry
	**Scale**	Transforming Geometry by Scaling
	**Affinity**	Transforming Geometry by Affinity
	**Axis to Axis**	Transforming Elements from an Axis to Another

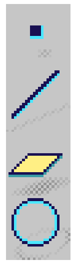
Cloud Segmentation

For	See
Segmentation by Curvature Criterion	Segmentation by Curvature Criterion
Segmentation by Slope Criterion	Segmentation by Slope Criterion

Cloud Analysis

	For	See
	**Information**	Information
	**Distance Analysis**	Analyzing Distances between two Sets of Elements
	**Curvature Analysis**	Performing a Curvature Analysis
	**Connect Checker**	Checking Connections between Surfaces

WireFrame



For	See
Point	Creating Points
Line	Creating Lines
Plane	Creating Planes
Circle	Creating Circle

Tools Toolbar

This chapter deals with:

**Cloud Display Options** Display Options and Graphic Properties

Specification Tree

Within the Quick Surface Reconstruction workbench, you can generate a number of elements that are identified in the specification tree by the following icons. If you open a CATPart generated with other CATIA applications, other icons may appear in the specification tree.

Please note that the pictures below are only example. The names and contents of elements will vary according to your actions.

Further information on general symbols in the specification tree are available in Symbols Used in the Specification Tree.

Icon	Action	Icon	Action
Geometrical_Set.1 object.1 Mesh Creation.1 Planar Sections.1	Geometrical Sets	Group-Geometrical_Set.1 . Group-Geometrical_Set.1 object.1 Mesh Creation.1 Planar Sections.1	Group
Curve Projection.1	Project Curves	Planar Sections.1	Planar sections
Free Edges.1	Create Free Edges	3D Curve.4	3D Curve
		Curve.2	Curve from Scans
Intersect.1	Intersection	Project.2	Projection
CleanContour.2 Curve.1 Curve.2 Curve.3 Join.1 `Merge distance`	Clean Contour		

Sphere.1 Center.1 R AngV1 AngV2 AngH1 AngH2	Basic Surface Recognition	Plane.1 Profile.1 Line.1 L1 L2 PlaneNormal.1 L3 L4	Basic Surface Recognition
CylinderAxis.2 Center.2 Point.10 Cylinder.2 Circle.1 L1 L2	Basic Surface Recognition	ConeAxis.1 Center.1 Top.1 Cone.1 Profile.1 Ang1 Ang2	Basic Surface Recognition
Surface.1	Powerfit, Surfaces Network	CurveNetwork.24	Curves Network
Loft.1	Loft Surface		
Symmetry.1	Symmetry	Axis to axis transformation.1	Axis to Axis
Scaling.1 Ratio	Scale	Rotate.1 Angle	Rotate
Translate.1 Length	Translate	Affinity.1 X Y Z	Affinity

Scans.1	Segmentation by Curvature or Slope Criterion	SubPolygon.1	Segmentation by Curvature or Slope Criterion
Join.1 `Merge distance`	Join	Adjusted node1 Curve.47 Curve.48 Curve.49 Curve.50	Adjust Nodes
Split.1	Split	Trim.1	Trim
Extrapol.1 Length	Extrapolate	EdgeFillet.1	Edge Fillet
Slice.2 Curve.11 Curve.12 Curve.13 Curve.14	Curves Slice	Split.3 Curve.5 Curve.6 Curve.7 Curve.8 Curve.9 Join.2 Join.3	Split CleanContour
Distance Analysis.1	Distance Analysis	Curvature Analysis.1	Curvature Analysis
Surface Connection Analysis.1	Connect Checker		

Glossary

B

boundary A boundary of trimmed or natural surfaces is defined by a clean contour.

C

clean contour A clean contour is a set of curves or surface edges, joined, ordered, eventually trimmed, with G0, G1 or G2 continuity constraints.

curves They may be created through:

- sketches on polygons,
- smoothing of scans,
- blending of curves,
- canonic recognition (line, circle),
- accuracy checking.

S

scans They can be obtained by :

- the identification of feature points (boundaries, sharp edges, breaks in curvature, inflection lines),
- sketches on polygons.

segmentation This operation defines zones on the cloud of points through:

- curvature criteria,
- normal criteria (isoslope, outlines),
- feature recognition (sphere, cylinder, plane, cone),
- threshold of distance deviation from a given surface,
- It may include scans or curves.
- It may be automatic, manual or both.

Index

9 A B C D E F G H I J L M N O P R S T U W

Numerics

A

D

E

F

G

H

I

J

L

M

R

S

T

U

W