Shape Sculptor

Overview

Shape Sculptor Version 5 application is a new powerful product enabling users to quickly create, edit, modify and sculpt polygonal models. Within Shape Sculptor, a scanned model can be imported and quickly edited to add details and new features. Polygonal models can also be created by tessellating existing surface models or generated from 3D curves. With its powerful and intuitive polygonal modeling tools, new features and details can be added to the polygonal model.

Thanks to its ease of use, Shape Sculptor is the perfect tool to rapidly edit and sculpt polygonal models and complements surface-based conceptual modeling tools when surfacing becomes too complex.

Shape Sculptor can be used with other curve and surface-based modeling tools, including *Generative Shape Design*, *FreeStyle* and *Automotive Class A* and also in cooperation with other polygonal products including *Digitized Shape Editor*, *STL Rapid Prototyping* and *Quick Surface Reconstruction*.

Using This Guide

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

To get the most out of this guide, we suggest that a beginning user reads the Getting Started chapter first of all and the Workbench Description to find his way around the Shape Sculptor workbench.

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

Where to Find More Information

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

The *Generative Shape Design*, *FreeStyle*, *Automotive Class A*, *Digitized Shape Editor*, *STL Rapid Prototyping* and *Quick Surface Reconstruction* documentations may prove useful.

See also the Conventions.

What's New?

The documentation will be available on the R13 SP2 level.

New Functionalities

Extracting Curves

Optimal CATIA PLM Usability for Shape Sculptor

Getting Started

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

The main tasks described in this section are:

This tutorial should take about 10 minutes to complete.

Entering the Workbench

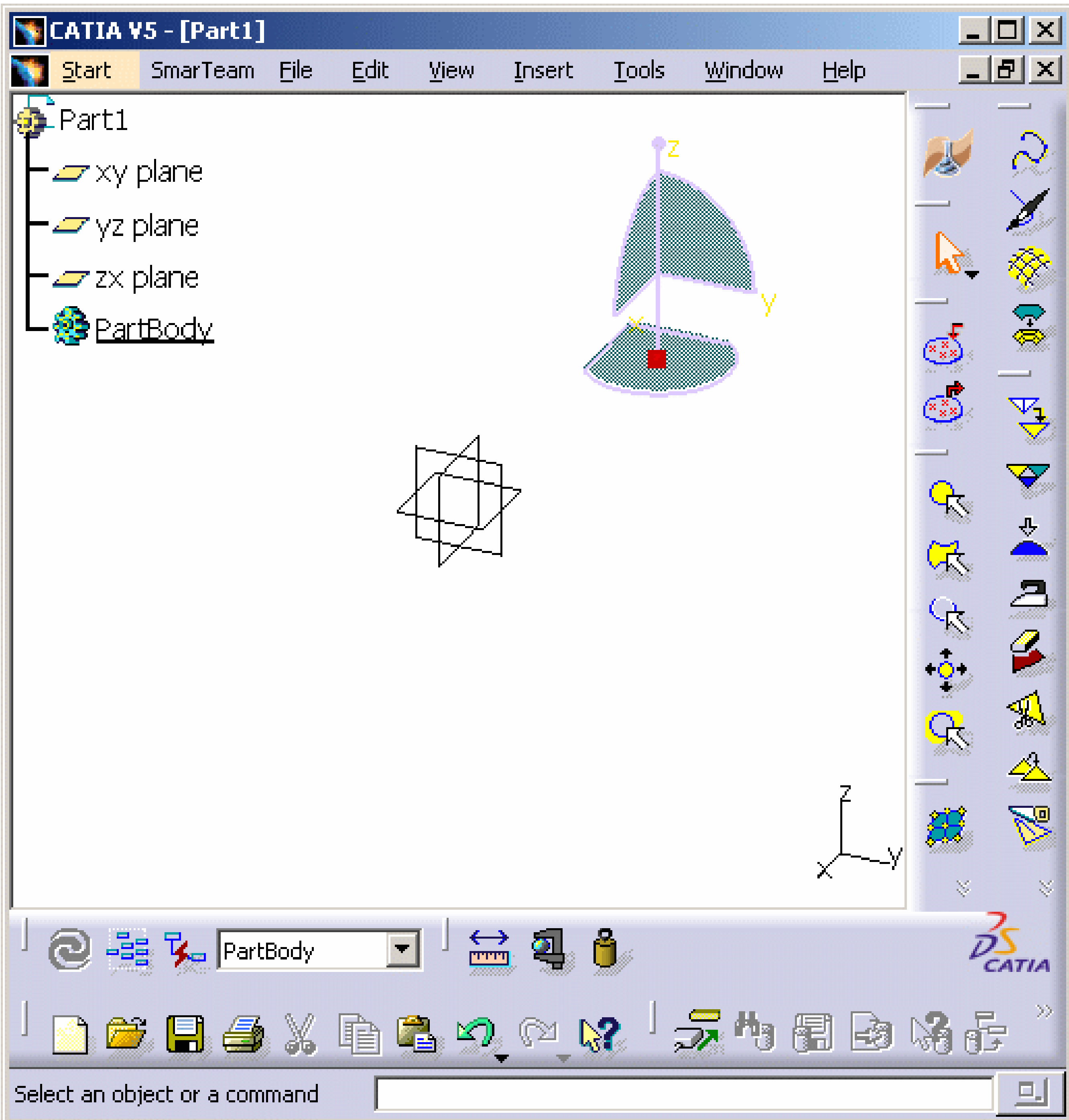
The first task will show you how to enter the Shape Sculptor workbench.

The only pre-requisite for this task is to have a current CATIA V5 session running.

1. Select **Shape Sculptor** from the **Start -> Shape** menu.

   The Shape Sculptor workbench is displayed and ready to use.

The workbench looks like this:



You may add the **Shape Sculptor** workbench to your Favorites, using the **Tools -> Customize** item. For more information, refer to the *Infrastructure User's Guide*.

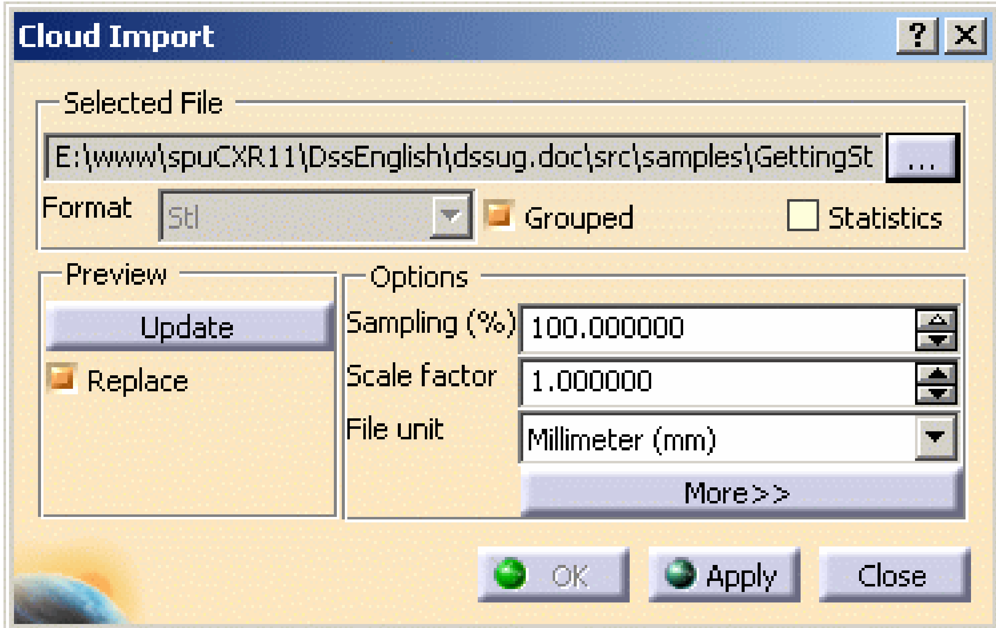
If you wish to use the whole screen space for the geometry, remove the specification tree clicking off the **View -> Specifications Visible** menu item or pressing F3.

Importing a Polygonal Mesh

This task shows how to import digit files describing a cloud of points (scanned or computed) or a polygon.

1. Click the **Import** icon.

   The **Cloud Import** dialog box is displayed.



2. In the **Selected File** area, use the button **...** to browse your directories and select a file.

3. Click **Apply** and **OK** to finish the import of the polygonal mesh.

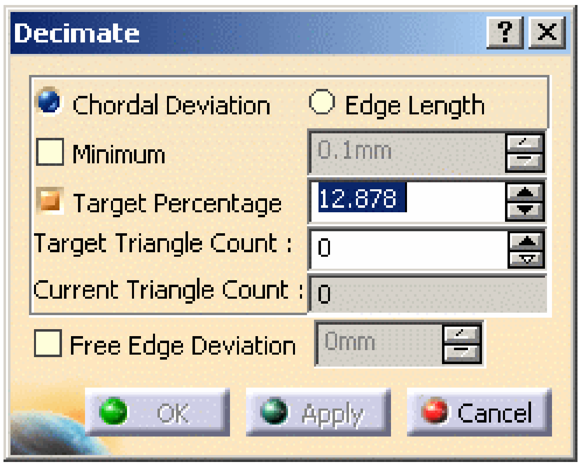
Decimating a Polygonal Mesh

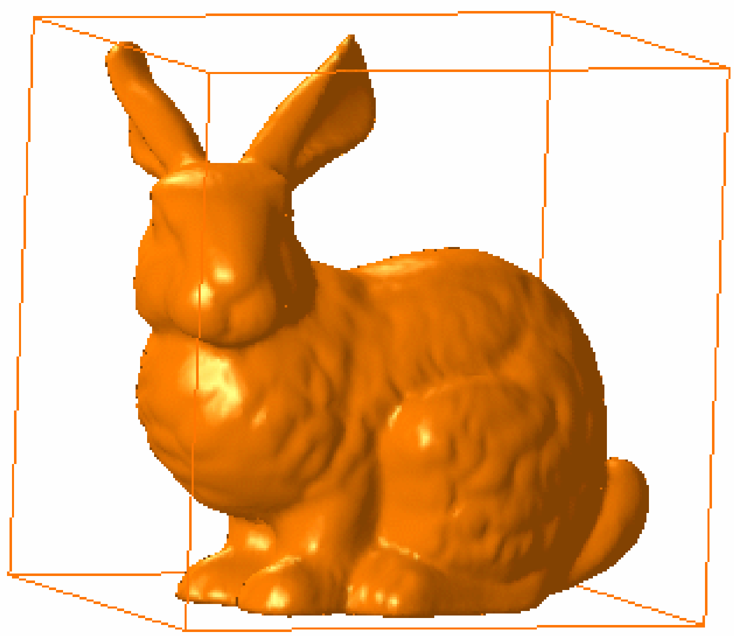
This task shows you how to decimate a polygonal mesh, in order to reduce the triangle count of a polygon.

Open the Bunny1.CATPart document.

1. Click the **Decimation** icon and select a polygonal mesh.

   The dialog box is displayed.



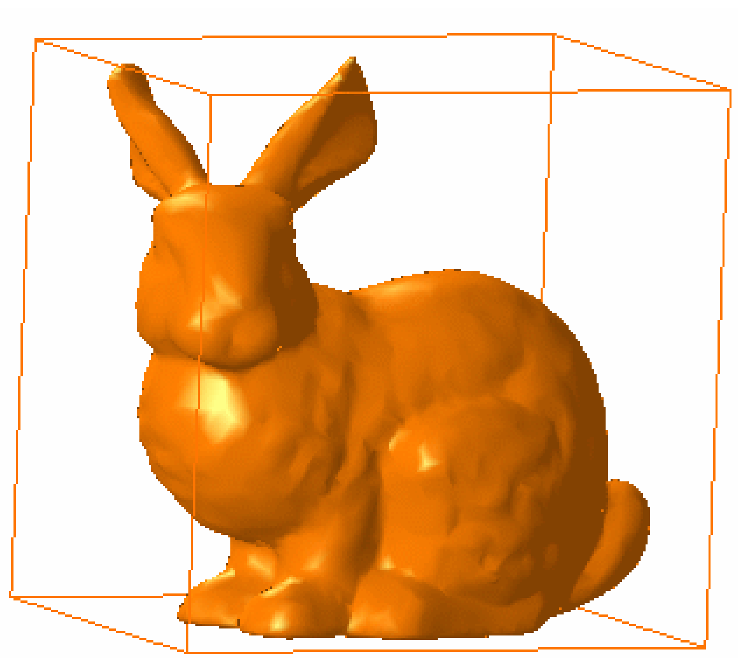


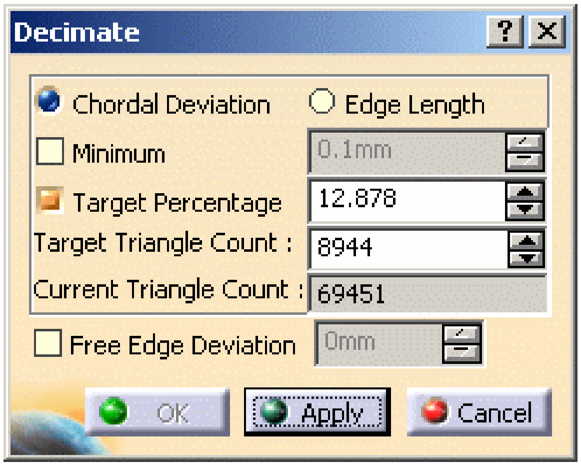
2. Check the type of decimation you want to apply:

- **Chordal Deviation**, or
- **Edge Length**

3. Click **Apply**.

   The decimation is computed. The counts of triangles are updated in the Current Triangle Count.





4. Either press OK to validate the decimation and exit the action,

or press Apply to start continue the decimation with the same values or new ones,

or press Cancel to exit the action and revert to the initial model

User Tasks

Input / Output

Importing Files

This task shows how to import digit files describing a cloud of points (scanned or computed) or a mesh.

Use the MultiImport1.cgo_ascii, MultiImport2.cgo_ascii, MultiImport3.cgo_ascii from the samples directory.

Available formats depend on the workbench you are working in.

Digitized Shape Editor

- Ascii free,
- Atos (the quality of the points can be taken into account),
- Cgo,
- Gom-3d (as points, scans, grids or meshes, the quality of the points can be taken into account),
- Hyscan,
- IGES (IGES Entities 116 are processed. If the cloud to import is made of Entities 116 only, the result is a cloud of points. Otherwise, the result is made of scans).
- Kreon
- Steinbichler (as points, grids or scans),
- Stl (bin or ascii, with creation of free edges and facets, if requested).

STL Rapid Prototyping

- STL files (bin or ascii, with creation of free edges and facets, if requested) (default option),
- Cgo,
- Ascii free,
- IGES (IGES Entities 116 are processed. If the cloud to import is made of Entities 116 only, the result is a cloud of points. Otherwise, the result is made of scans).

- In Cgo, Ascii and IGES formats, you can not process more than 10,000 points at each import, in one or several files, e.g. you can not import 4 files of 3,000 points each in one shot but you can import them separately.
- This limitation applies to the input files (before Sampling or resizing with the bounding box).
- If you try to import over 10,000 points in one shot, a fatal error panel is displayed: **Too many points for this configuration**.
  - If the **Grouped** option is active, no file is imported.
  - If the **Grouped** option is not active, files are imported as long as the sum of their points does not exceed 10,000 points.
- **Mesh Regeneration** is not available on those files.

Shape Sculptor

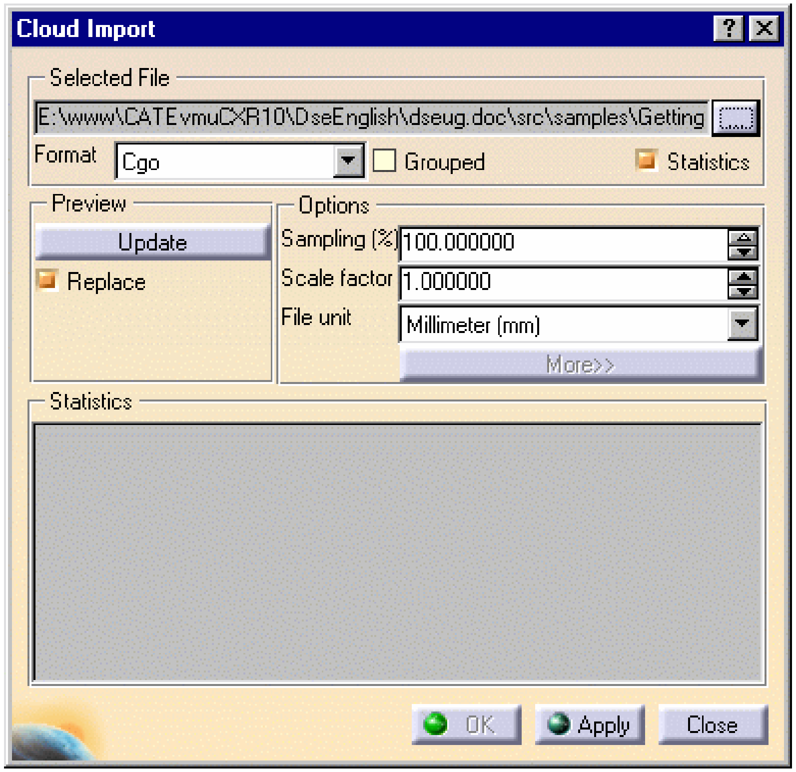
- STL files (bin or ascii, with creation of free edges and facets, if requested).

1. Click the **Import** icon . The **Cloud Import** dialog box is displayed.
2. In the **Format** field, select the file format.
3. In the **Selected File** area, use the button ... to browse your directories and select a file.
4. Check the box **Statistics** to display information about the model you are importing. If you want to import several files in one shot, please refer to the Grouped explanations.
5. In the **Options** field:

- Enter the **Sampling** percentage to apply;

  The sampling value determines the percentage of points or scans or grids that will be read from the digit file.

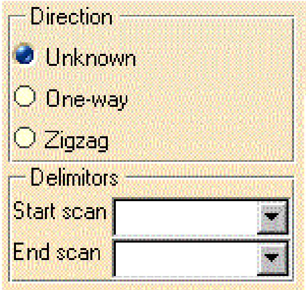
- Enter the **Scale factor** to apply to the model, as well as the **Unit** used in the file.



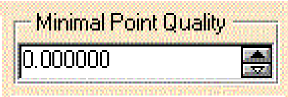
- If the extension of the file you have selected is consistent with the list proposed, the **Format** field is updated automatically. Otherwise, be careful to enter the correct format in that field.
- Once you have performed an import operation, V5 proposes the last entered file path and format as default. If you click on ..., the last entered directory is proposed as default.
- The **File unit** option is not relevant to the Steinbichler format, nor the **Sampling percentage** to the Stl format.

6. For some digit file formats, you may want to enter additional data that are displayed by clicking the button **More>>**

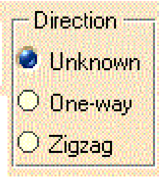
For Ascii:



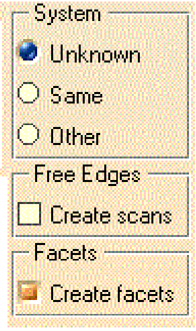
For Atos and Gom-3d:



For Iges:



For Stl:



Direction and **Delimitors** apply to scans. Enter these data whenever you know them.

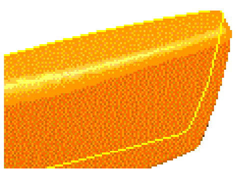
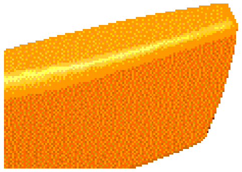
Minimal Point Quality is used to clean Atos file from invalid points. The quality value of a point lies between 0 and 255 (low to high). Choose a value to ignore points below that value:

- **Minimal Point Quality** value is 125:

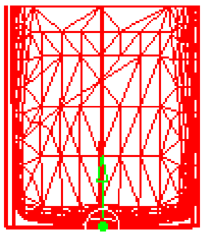
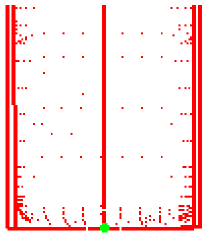
- **Minimal Point Quality** value is 75:

System applies to the operating system (Unix or Windows NT) used to generate the binary data: select **Same** if you know you are using the same operating system as the one used to generate the binary data, **Other** for the other way, **Unknown** if you have no indication.

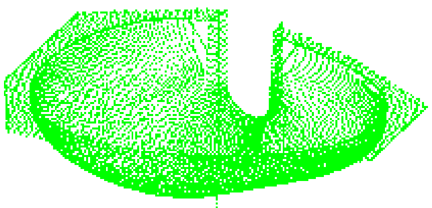
Free Edges is used to create or not the scans representing the free edges of a mesh:

 or 

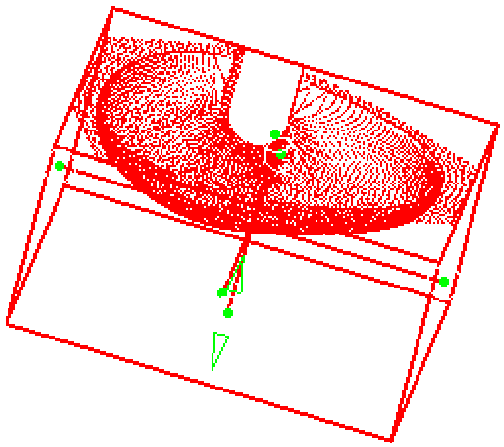
Facets is used to create or not the facets of the imported mesh:

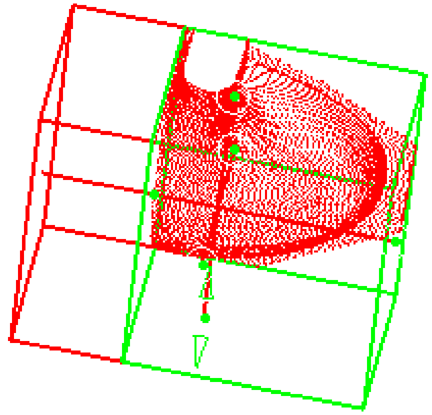
 or 

7. Click **Apply** to display the cloud of points or mesh:

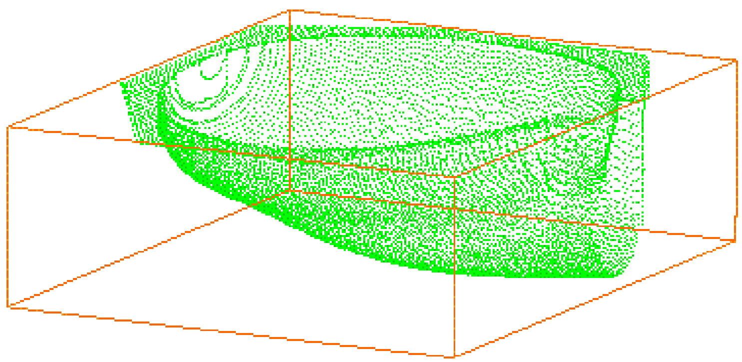


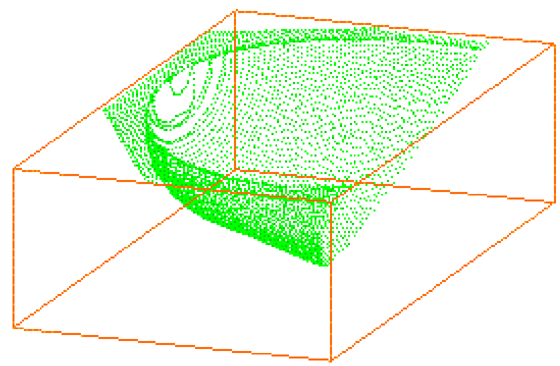
8. Push the button **Update** to display the bounding box of the cloud of points or mesh. Use the green arrow to resize it in order to import only a part of the cloud of points or mesh.





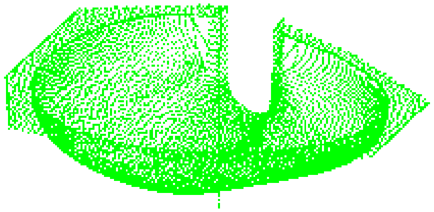
- The bounding box appears every time the cursor passes over a cloud of points or a mesh. Its size corresponds to that of the visible points.
- If a local axis system is set as current, the file will imported in this axis systems and not in the absolute axis system as previously. If no local axis system is set as current, the file will be imported in the absolute axis system.
- Moreover, if a local axis system is set as current, the axis system of the dynamic box used to select a portion of the imported file when the Update button is pushed is parallel to the local axis system axis.





The check box **Replace** is used to replace the current cloud of points or mesh by a new one.

9. Once you are satisfied with the preview, click **Apply** and **OK** to finish the import of the cloud of points or mesh.



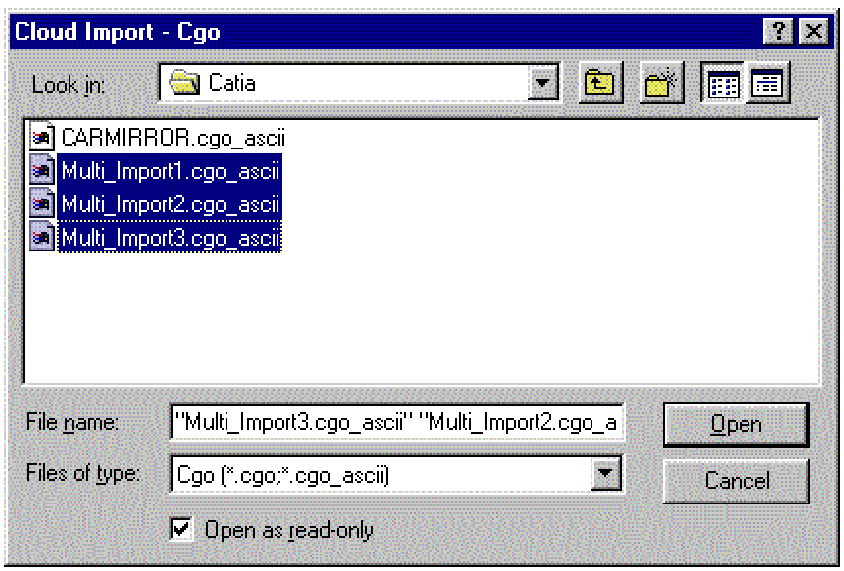
- The name of the element created in the specification tree is the name of the original file, without its extension.
- Undo and Redo are available.
- V5 memorizes the data of the last imported file and proposes them at the next import action.

Importing a Set of Files

1. Click the **Import** icon . The **Could Import** dialog box is displayed. The operating mode is the same as for one file.

The files to import must:

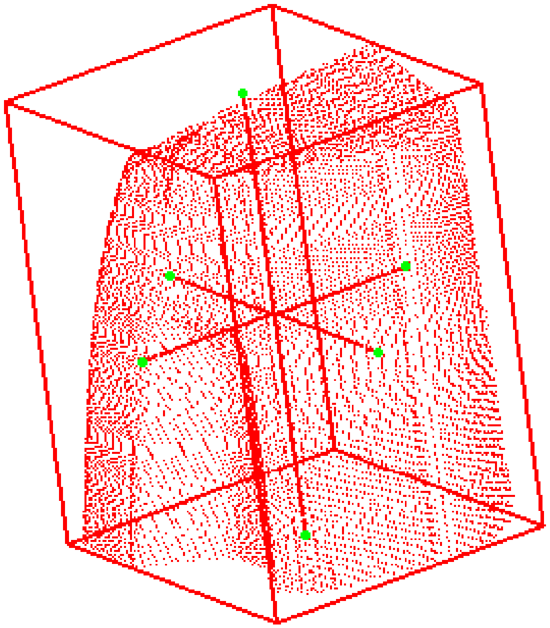
- have the same format,
- be located under the same directory.



The **Selected File** field looks like this: 

If you check the **Grouped** option (this is the default status):

All the files are imported as one single cloud of points instead of several:

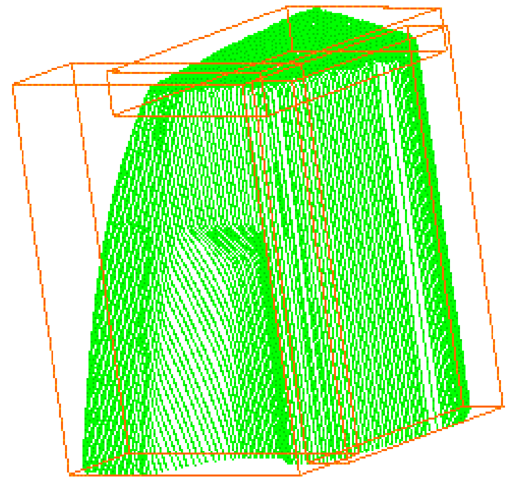


The three digit files have been imported together, resulting in one cloud of points or mesh.

One cloud of points Element **Cloud Import.x** is created in the specification tree, with the icon of the **Import** command.

If you do not check the **Grouped** option:

The files are imported separately.



The three digit files have been imported separately, resulting in three clouds of points.

One cloud of points element is created in the specification for each input file, with the icon of the Import command and the name of the input file (**MultiImport1.1, MultiImport2.1, MultiImport3.1**) .

- You can also merge several clouds of points into one whenever necessary, using the **Merge Clouds** command.

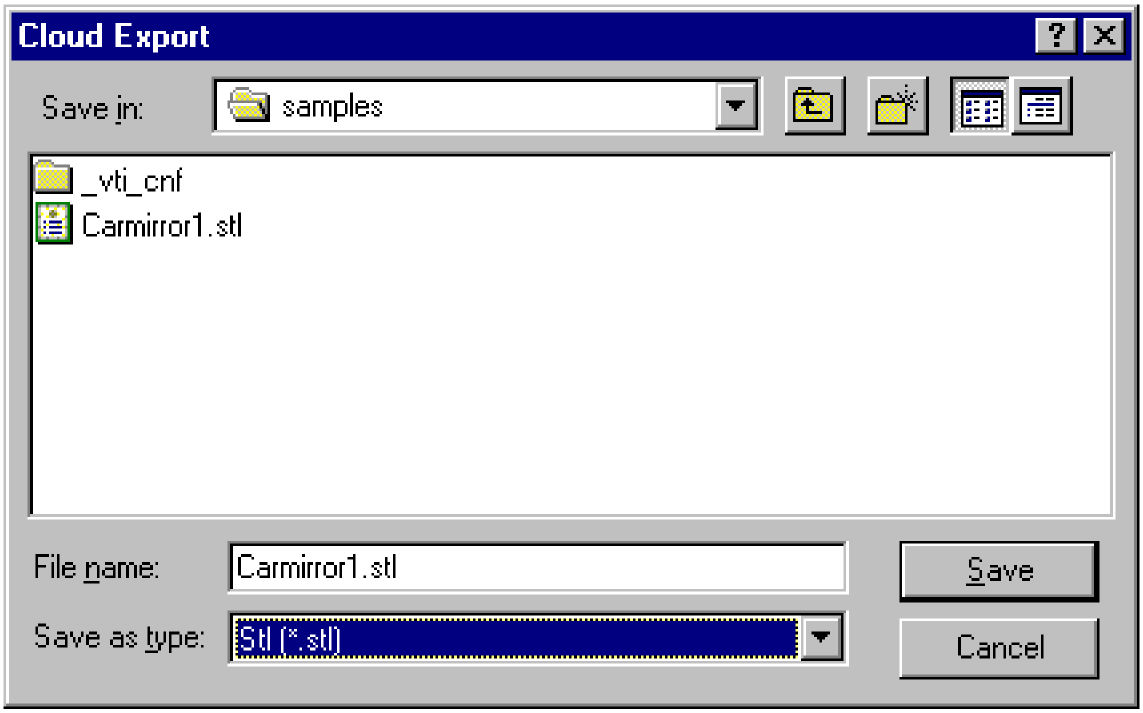
Exporting Polygons to STL

This task shows how to export a mesh to binary Stl format.

Open the Cloud.CATPart from the samples directory.

1. Select the Polygon.1 and then the export icon .

   The export dialog box is displayed.



2. One export format is available: **Stl.**
3. Browse your directories and enter the target directory and file name. Then click **Save**.

- The selection is exported with the current local axis system if any, with the absolute axis system otherwise.

- You can export only one element at a time.
- In STL Rapid Prototyping, only the Stl format is available.

Selecting Components

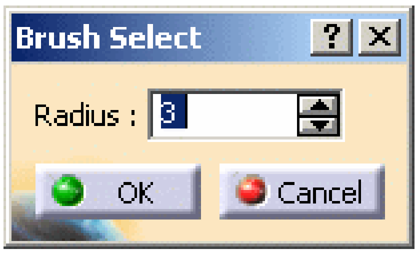
Selecting With a Brush

This task shows how to select triangles from one or multiple polygonal meshes using a brush.

Open the Bunny1.CATPart document.

1. Click the **Brush Select** icon .

The Brush Select dialog box displays.



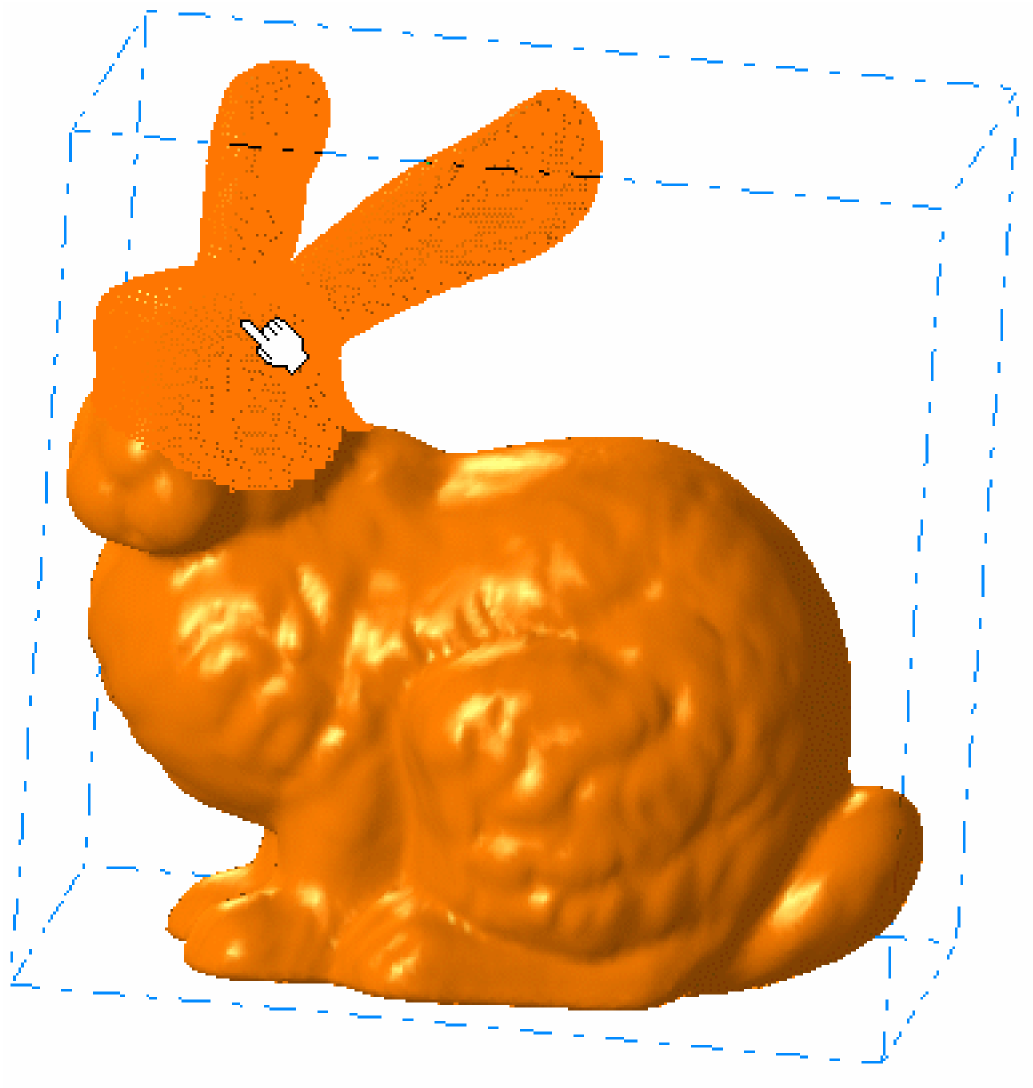
2. Define radius to increase or decrease the brush size.

3. Drag the brush over the polygonal mesh to select triangles.

4. Click OK.

You may modify the size of the brush during the selection, simply:

- use the **Ctrl** key to add triangles to the selection, or
- use the **Shift** key to remove triangles from the selection.



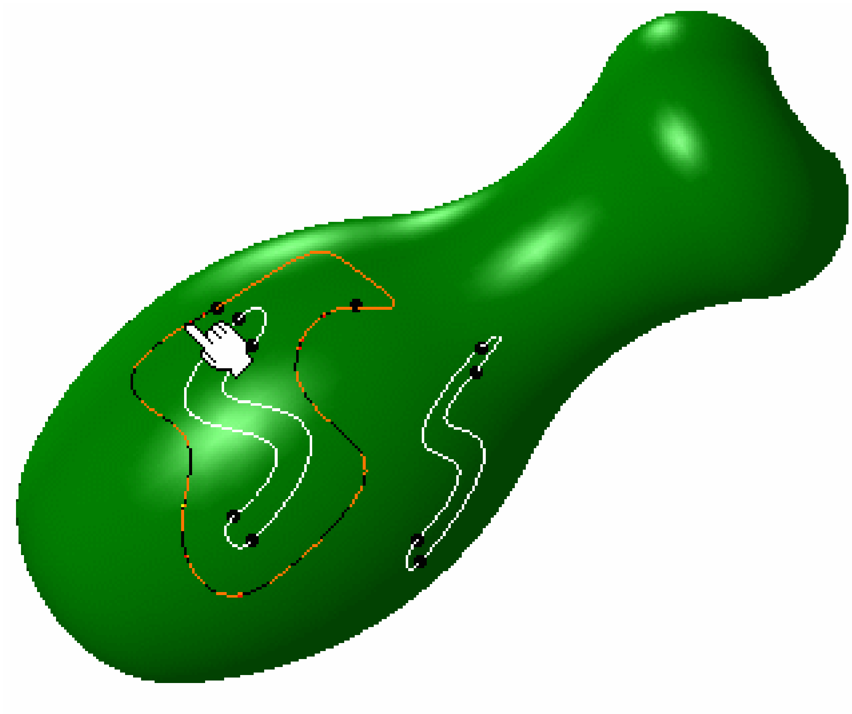
Selecting With Curves

This task shows how to select triangles from the polygonal mesh using curves.

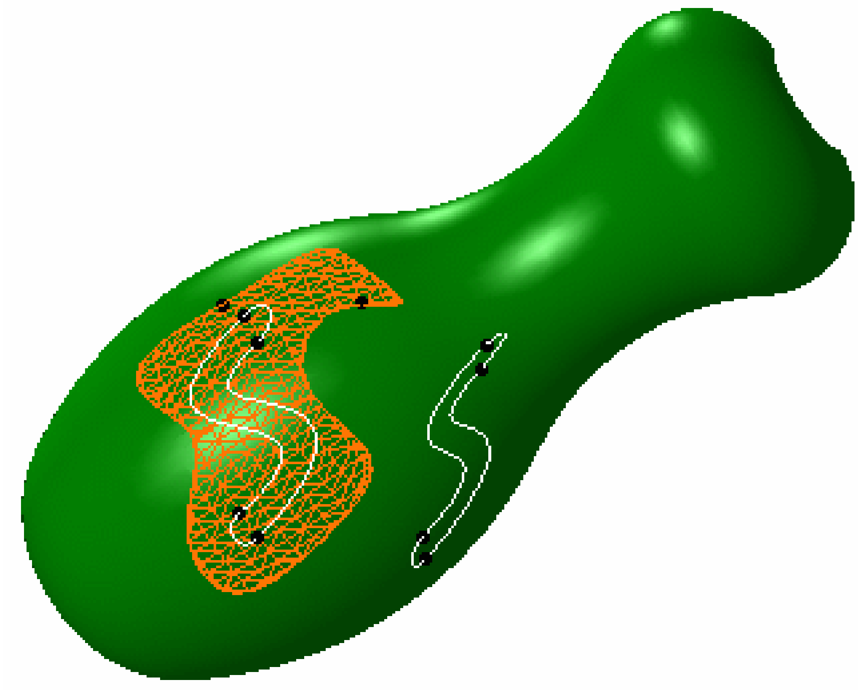
Open the Pin2.CATPart document.

1. Click the **Curve Select** icon .

2. Select the curves delimiting the area to select.



3. Click inside in this area to delimit it using the selected curves.

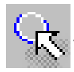


- Selecting several curves can be done holding the **Ctrl** key.
- Selecting a curve loop enables the selection of all triangles inside this loop.

Selecting Holes

This task shows how to select surfaces boundaries of a polygonal mesh.

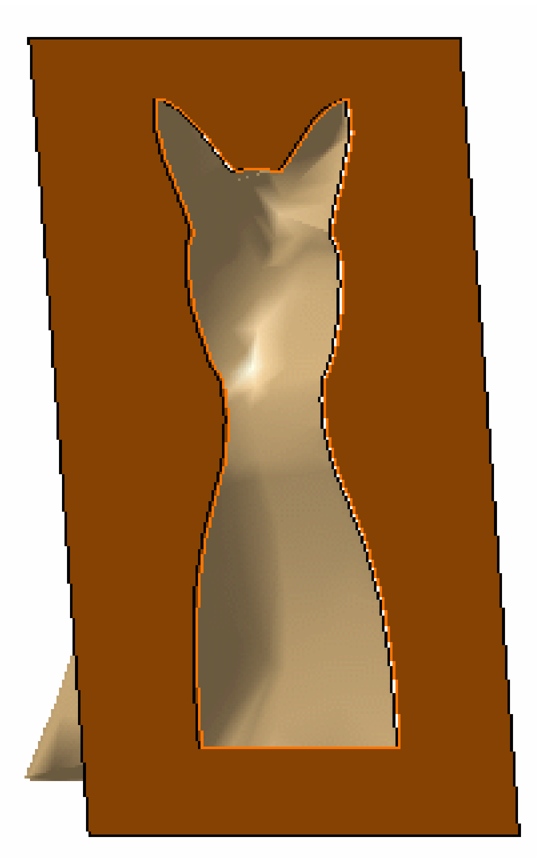
Open the Felix2.CATPart document.

1. Click the **Hole Select** icon .

Holes, that is the boundaries of the polygonal mesh, become visible.

2. Move the mouse onto a hole and select it.

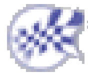
The hole is highlighted and put in the selection list.



- If you hold the **Ctrl** key while selecting the holes, the latter are appended to the selection.

  Otherwise the selection is automatically cleared before a new one is made.

- Once holes are selected, you can activate any other command.
  For instance, you can use the Stitching command to stitch the polygonal mesh along the selected holes.



Selecting With Flood

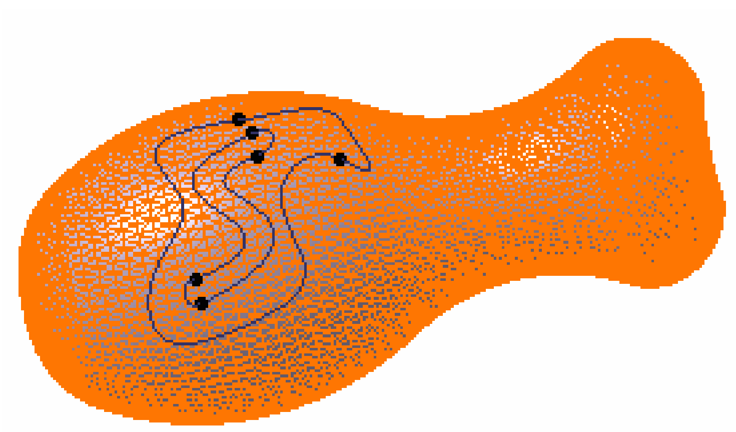
This task shows how to quickly select triangles that are topologically connected to the selected triangle.

Open the Pin1.CATPart document.

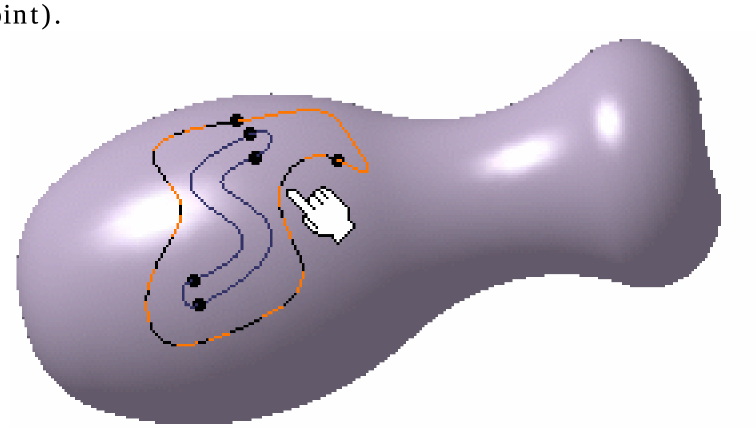
1. Click the **Flood Select** icon .

2. Click on a triangle of the polygonal mesh.

All triangles that can be reached from this initial triangle are selected.



Selecting a curve enables to extend the selection to all the connected curves (continuous in point).

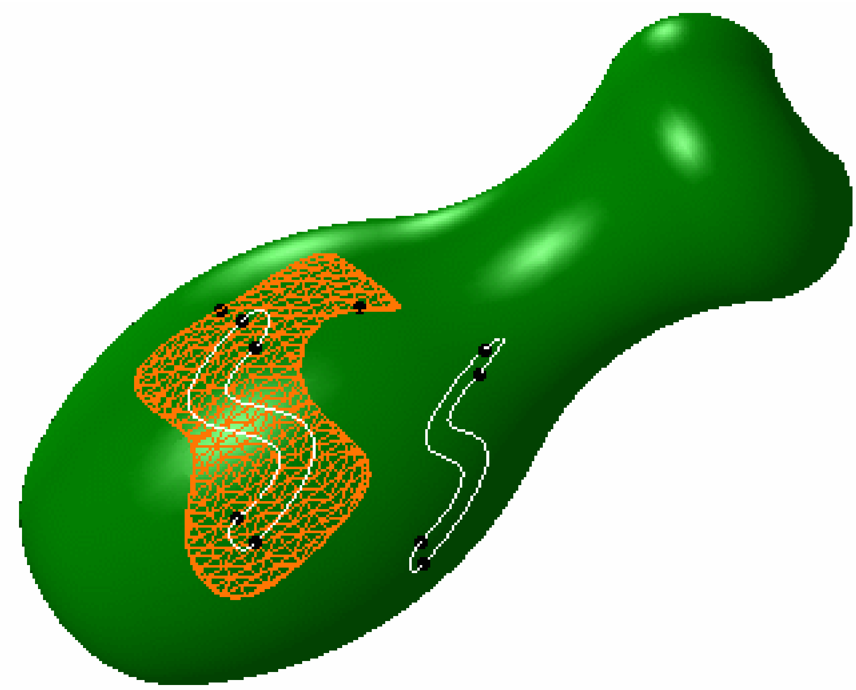


Inverting the Selection

This task shows how to reverse the selection of triangles.

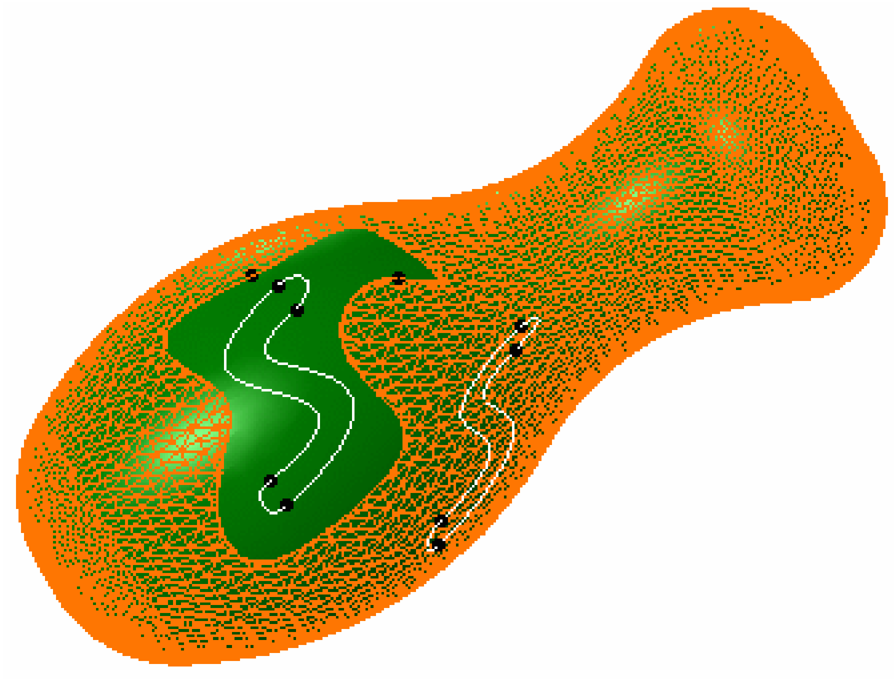
Open the Pin2.CATPart document.

1. Select triangles using curves as shown in the Selecting Using Curves chapter.



2. Click the **Invert Select** icon .

The selection of triangles is inverted from the current selection.



Creating

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.

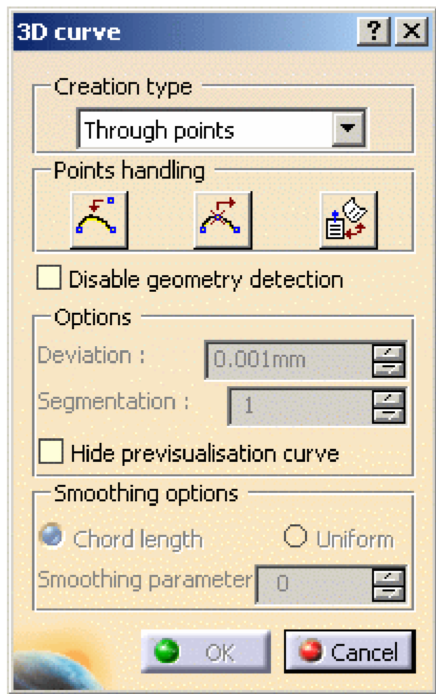
- Selecting 3D points
- Editing
- Keeping a point
- Imposing a Tangency Constraint
- Imposing a Curvature Constraint
- Setting as Arc Limit
- Removing a Point
- Constraining a Point

Open a new .CATPart document.

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

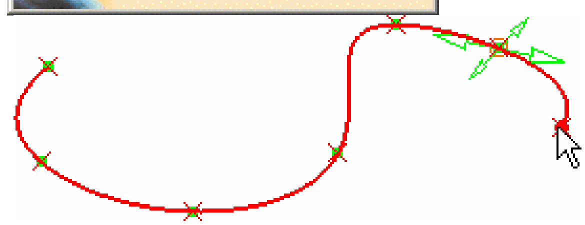
   The 3D curve dialog box is displayed.

2. Choose the curve creation type.

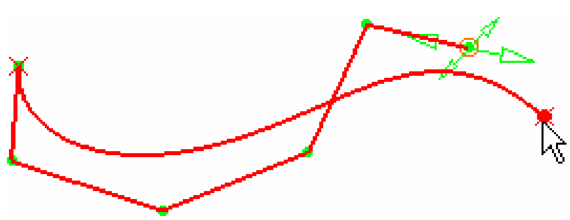


Whatever the chosen type, the curve is previewed in dotted line as you move the pointer.

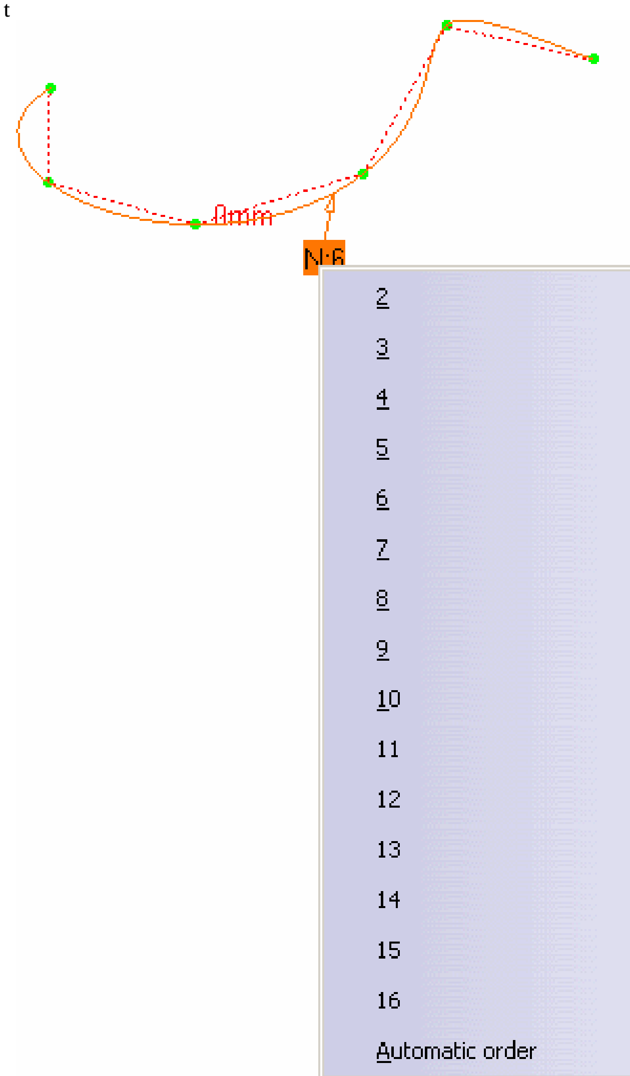
- **Through points**: the resulting curve is a multi-arc curve passing through each selected point.



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

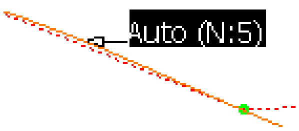


- **Near points**: the resulting curve is a single-arc, with a set degree and smoothed through the selected points.



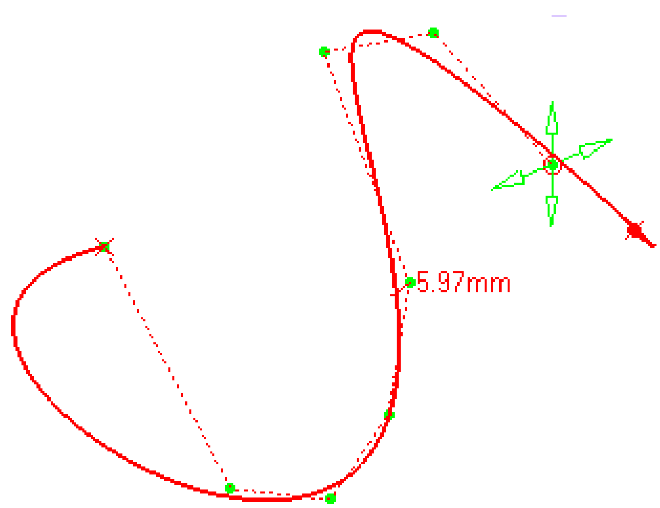
You can edit the order by right-clicking the displayed text (displayed using the U, V Orders icon from the FreeStyle Dashboard or the **Order** option from **Tools** -> **Options** menu, **Shape** -> **FreeStyle** -> **General** tab), and choosing a new order value.

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.

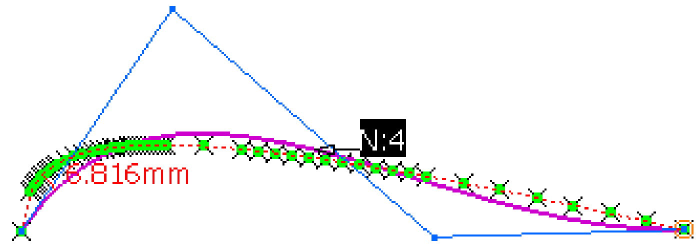
The result is a set degree through the selected points.



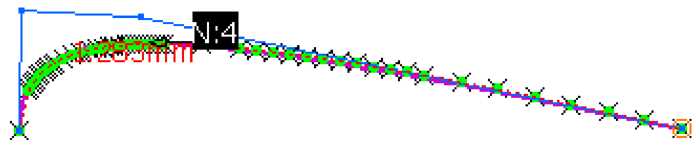
- The **Segmentation** option enables the user to set the maximum number of arc limits. These arcs are construction points and are inserted into the curve automatically.
The minimum value is set to 1.

- Smoothing options are now available to parameterize the curve:

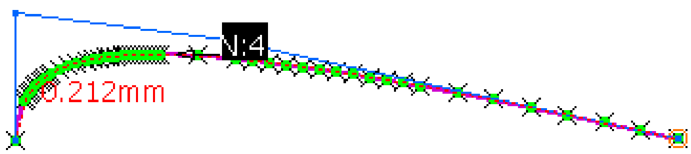
- **Chord Length** (default parameterization)
  Smoothing parameter = 0



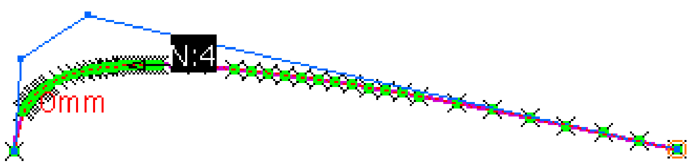
- **Uniform**
  Smoothing parameter = 0



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



Smoothing parameter = 50



Smoothing parameter = 130

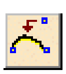
Deviation, **Segmentation**, and **Smoothing** options are only available for the Near Points creation type.

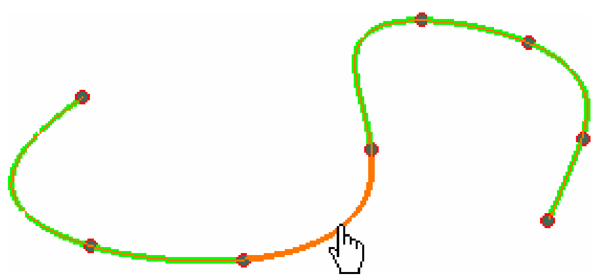
3. Move the pointer over a point.

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.

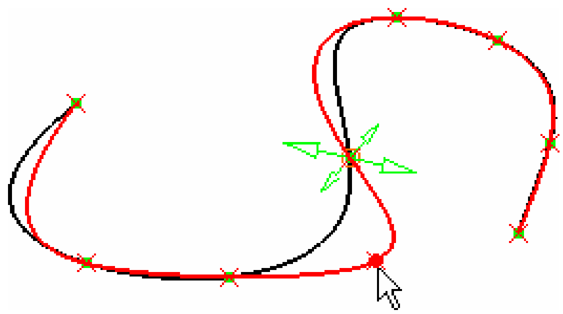
Right-click on the manipulator to display the contextual menu. From then on you can choose the **Edit** item to display the Tuner dialog box and enter space coordinates for the selected point, or choose the **Impose Tangency** item to set a tangency constraint on the curve at this point.

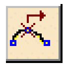
4. Click the **Insert a point** icon  within the dialog box.
   The curve freezes.



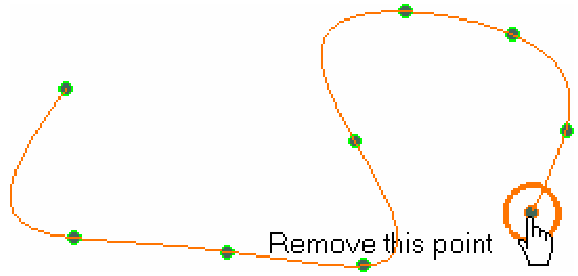
5. Click the segment, between two existing points where you wish to add a new point and click the point location.

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



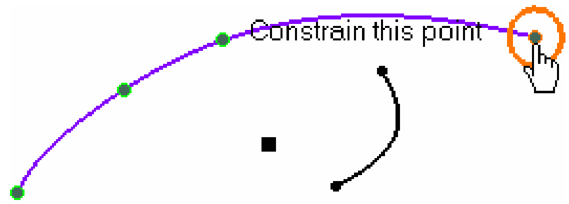
6. Click the **Remove a point** icon  within the dialog box, and select one of the existing points.

   The curve is recomputed immediately without the selected point.

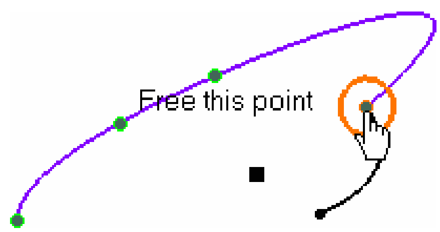


7. Click the **Free or constrain a point** icon  within the dialog box, then select the point.

- 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.



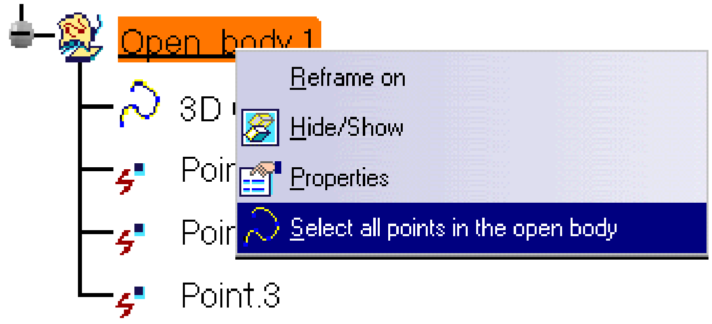
You can snap a point onto a surface using the **Free or constrain a point** icon. The point will be lying onto the surface, but not constrained. It can be moved using the manipulators.

8. Click OK to create the curve.

   A 3DCurve.xxx appears in the specification tree.

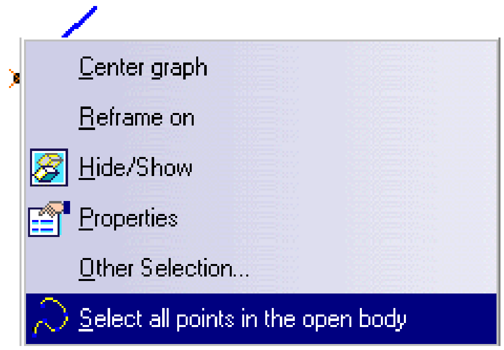
- Use the F5 key to move the manipulators into a different plane of the compass. See Managing the Compass.
- You cannot add a point past the end points. To do this, you need to add a point before the end point, move the new point where the end point lies, then move the end point to a new location.
- 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.

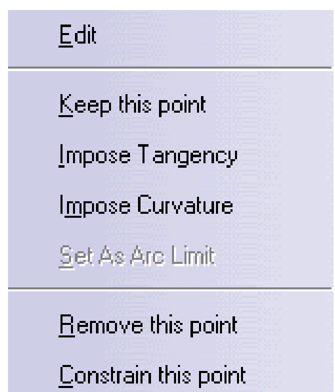
The Select all points contextual menu is available within the 3D curve action only, i.e. it is not available when you select the scan before entering the action but appears when the 3D Curve dialog box is open.



- 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.

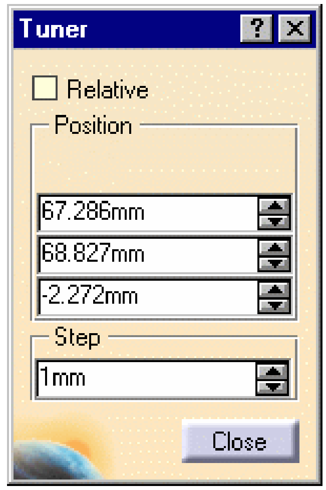


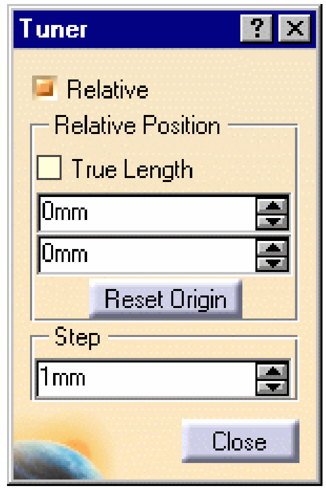
According to the creation type, the following options are available:

	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.

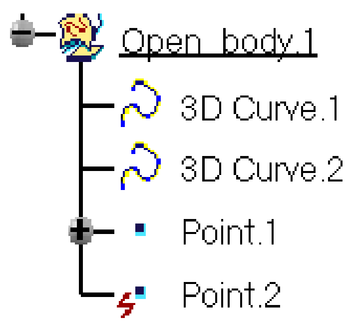


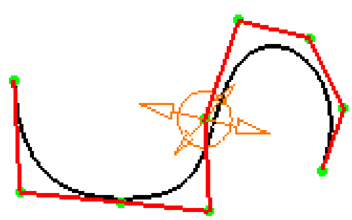


The **Relative** check box enables you to redefine the tangency relative position, and its vector's step.
The **Reset Origin** button allows you to reset the origin of the relative position.

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.

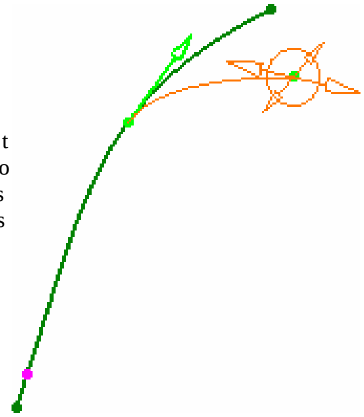




Imposing a Tangency Constraint

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.

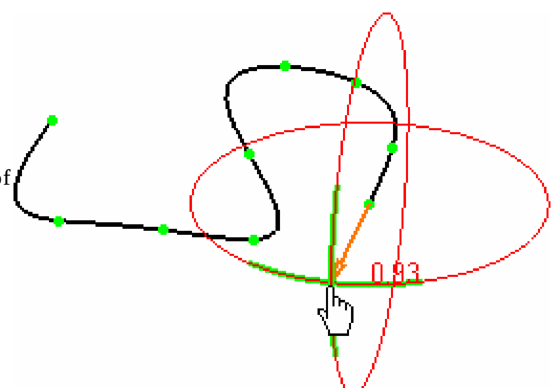
Here is how to do it:

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:

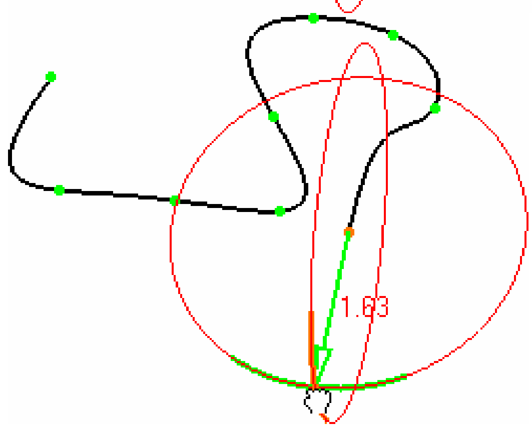
- two arrows representing the normed directions (vectors) of the tangency
- circles representing manipulators for this vector



You can also modify the tangency constraint by:

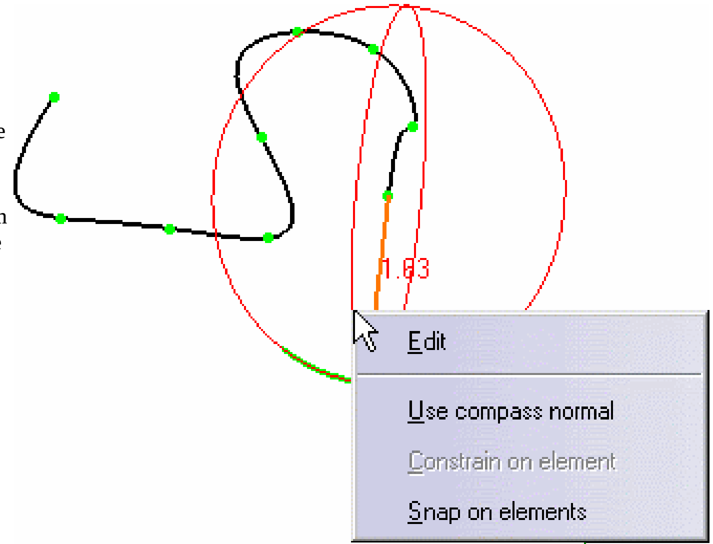
- pulling the arrow
- gliding the circles
- double-clicking the arrow to invert the tangency direction

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



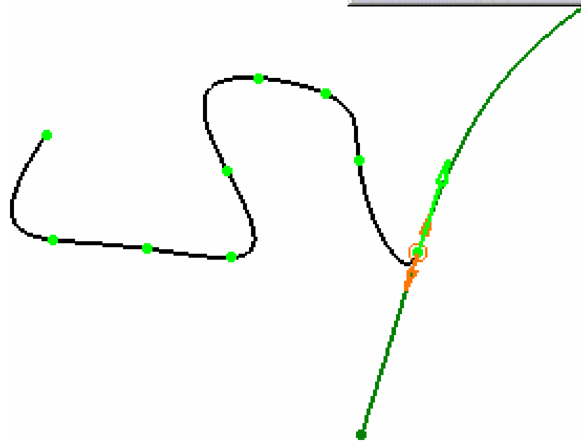
Right-clicking on any of the manipulators, you can also choose to define the constraint according to an external element:

- **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.



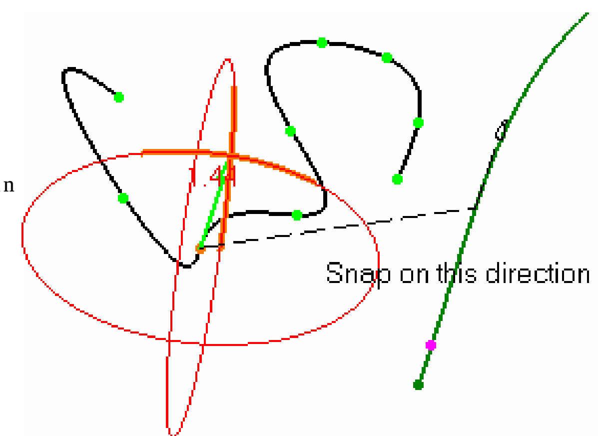
P2

- **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.

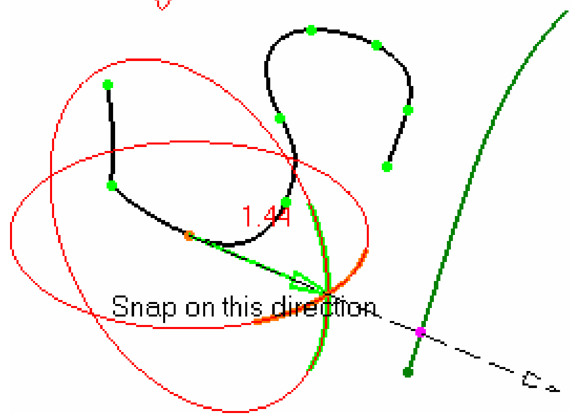


By default, when the tangency vector is constrained onto another curve, its initial direction is retained.

- **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.



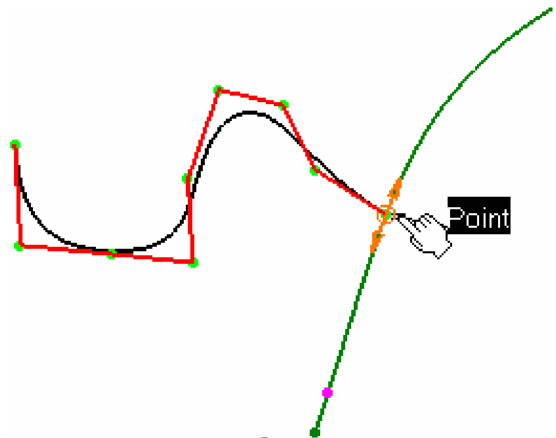
- When snapping on an element, use the Control (Ctrl) key to obtain an exact snap, taking into account both the detected element and the vector's norm.
- Use the Shift key as a shortcut to activate/de-activate the **Snap on elements option when** passing the pointer over geometric elements.

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.

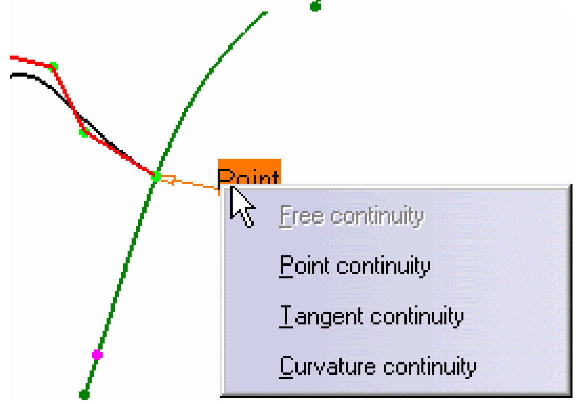
Control Points Curve Constraint

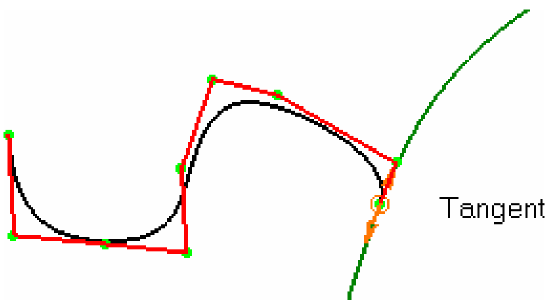
Even though you cannot impose a tangency constraint on a curve created in **Control points** mode, you can constrain its end points on another curve, as described in step 7 above.

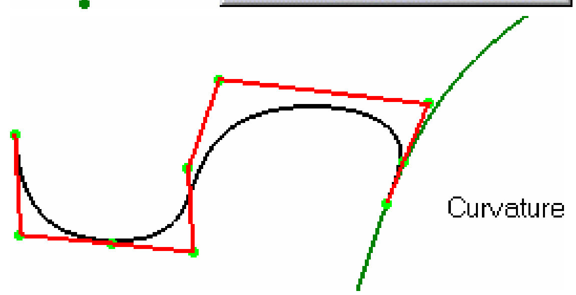
When setting a constraint on an end point, a text is displayed indicating the type of continuity between the two curves.



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

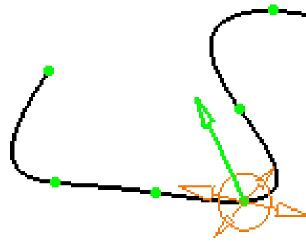
This means that these second and third points will be modified if you move the constrained point along the constraining element, using the manipulators. However, you cannot constrain these points, because they are considered as already constrained. If you try to do so, a warning message is displayed. 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.

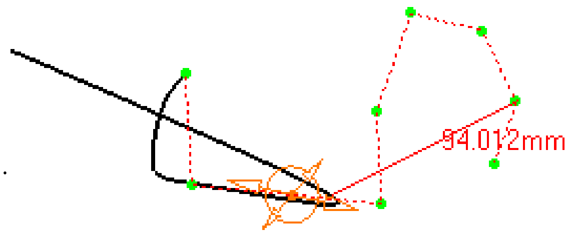


- To impose a curvature continuity, you must ensure that a tangency continuity already exists.
- This option is only available for the **Through points** and **Near Points** creation type.

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.



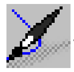
You can use standard shortcuts (Ctrl and Shift keys) to select, multi-select, and de-select any combination of control points on these curves.

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 Paint Curves

This task shows you how to create and draw curves onto the polygon that entirely lie on it.

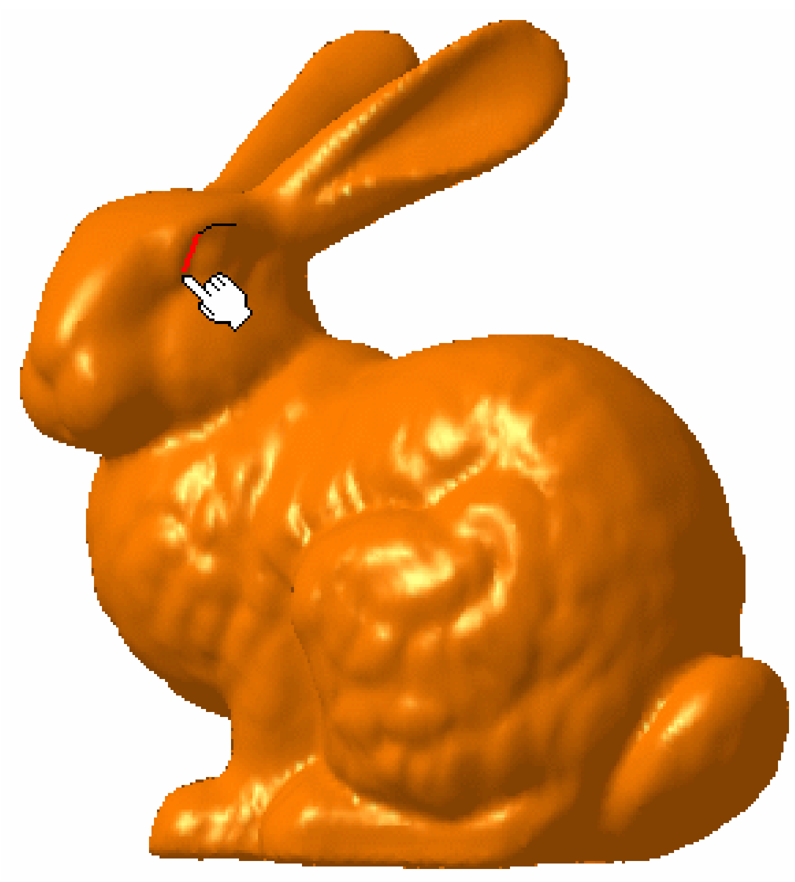
Open the Bunny1.CATPart document.

1. Click the **Paint Curve** icon .

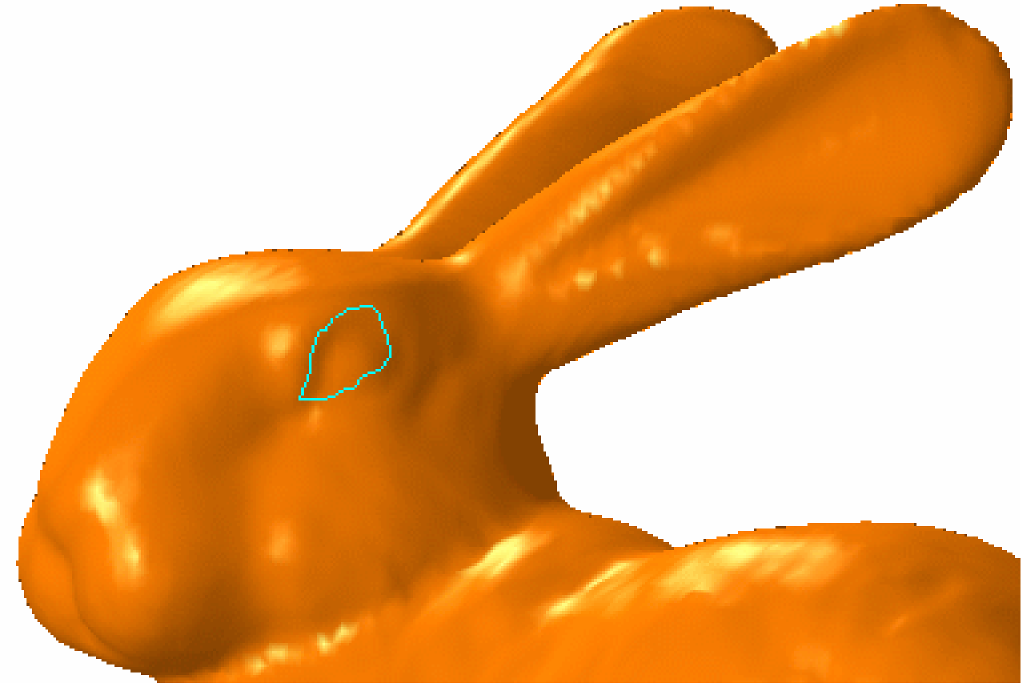
2. Using the mouse, select the first point on the polygon.

To paint the curve, you can either click point by point (subsequently create new segments) or drag the mouse.

3. Double-click on the polygon to end the painting and exit the command.

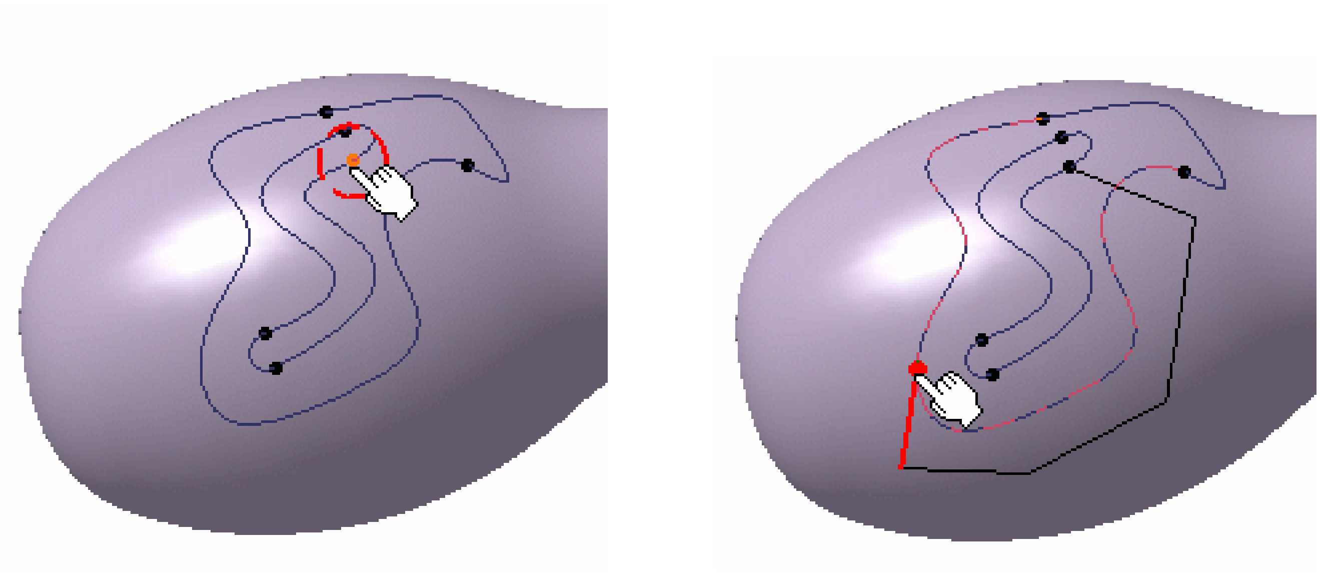


The curve (identified as Mesh Curve.xxx) is added to the specification tree.



- Hold the **Ctrl** button to complete the loop, and therefore exit the command.

- You can now paint a curve using an existing curve that lies on the polymesh, either by:
    - clicking a vertex to start or continue the paint curve, or
    - snapping onto the curve and selecting a point on this curve.



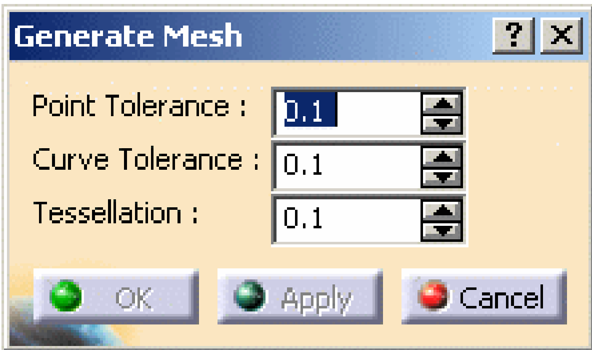
Generating Meshes

This task shows you how to generate a polygonal mesh from curves and selected triangles.

Open the Vase2.CATPart document.

1. Click the **Generate Mesh** icon .

   The Generate Mesh dialog box displays.



2. Using the **Ctrl** key, select the curves and triangles (if necessary).

3. Define the tolerances:

- Point Tolerance: tolerance for the fitting points
- Curve Tolerance: tolerance for the fitting curves
- Tessellation: resolution of the generated mesh

4. Click Apply to preview the polygonal mesh.

5. Click OK to generate it.

6. Perform the operation again with the other set of curves.

The generated mesh passes through the selected curves and vertices of the selected triangles.

The mesh (identified as Mesh Generation.xxx) is added to the specification tree.

Tessellating

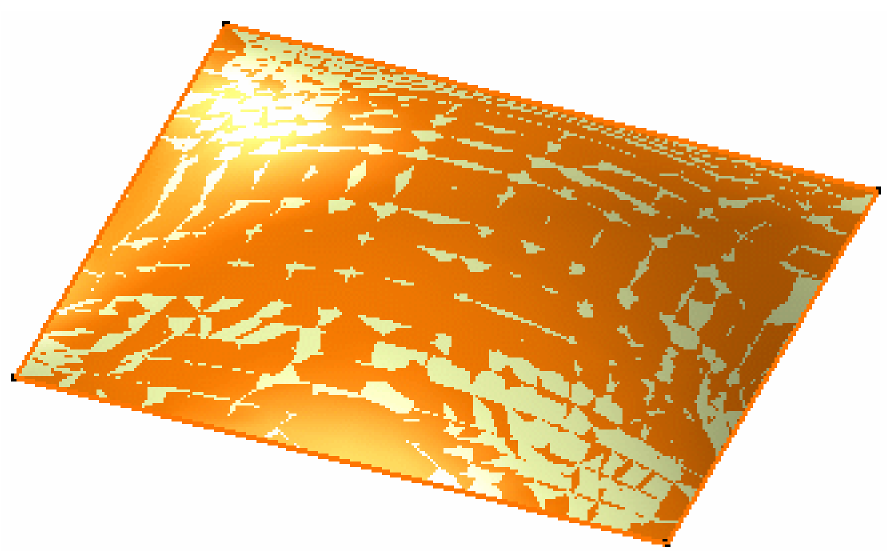
This task shows how to tessellate a surface and convert it into a polygonal mesh.

Open the Tessellate1.CATPart document.

1. Click the **Tessellate** icon .

   The Tessellate dialog box displays.

2. Select the surface(s) to be tessellated.
3. Define the Tolerance parameter to control the tolerance to which the surface(s) can be tessellated.
4. Click Apply.



The tessellation feature (identified as Tessellation.xxx) is added to the specification tree.

Editing

Decimating Meshes

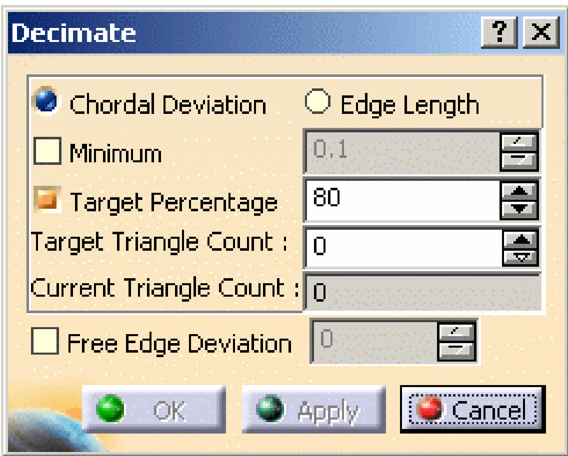
This task will show you how to decimate a mesh.

Decimation is a command reducing the triangle count of a mesh for a quicker execution of commands. It also reduces the memory requirements for the model. Many large meshes can be represented accurately with less triangles.

Decimation can be performed on the entire region or a selected region of the mesh.

Open the SmoothMesh01.CATPart from the samples directory

1. Click the **Decimation** icon and select a mesh. The dialog box is displayed:



2. Check the type of decimation you want to apply:

- by **Chordal Deviation** if you want to preserve the shape of your model, even in areas with a high curvature,
- by **Edge Length** if you want to remove triangles with tiny edges and obtain a more uniform mesh. However this may result in a loss of accuracy in areas with a high curvature.

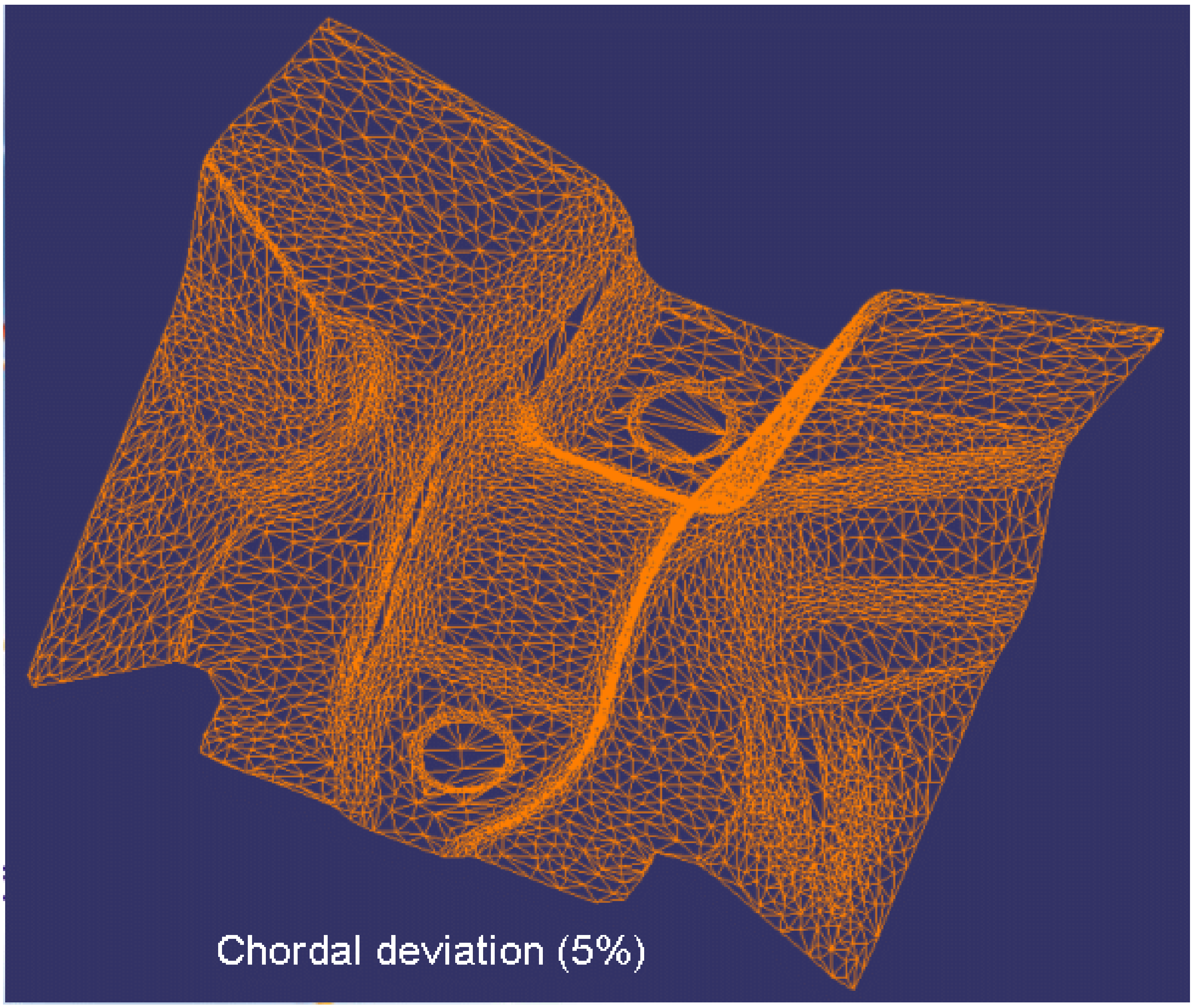
3. Then, decide how you want the decimation to process:

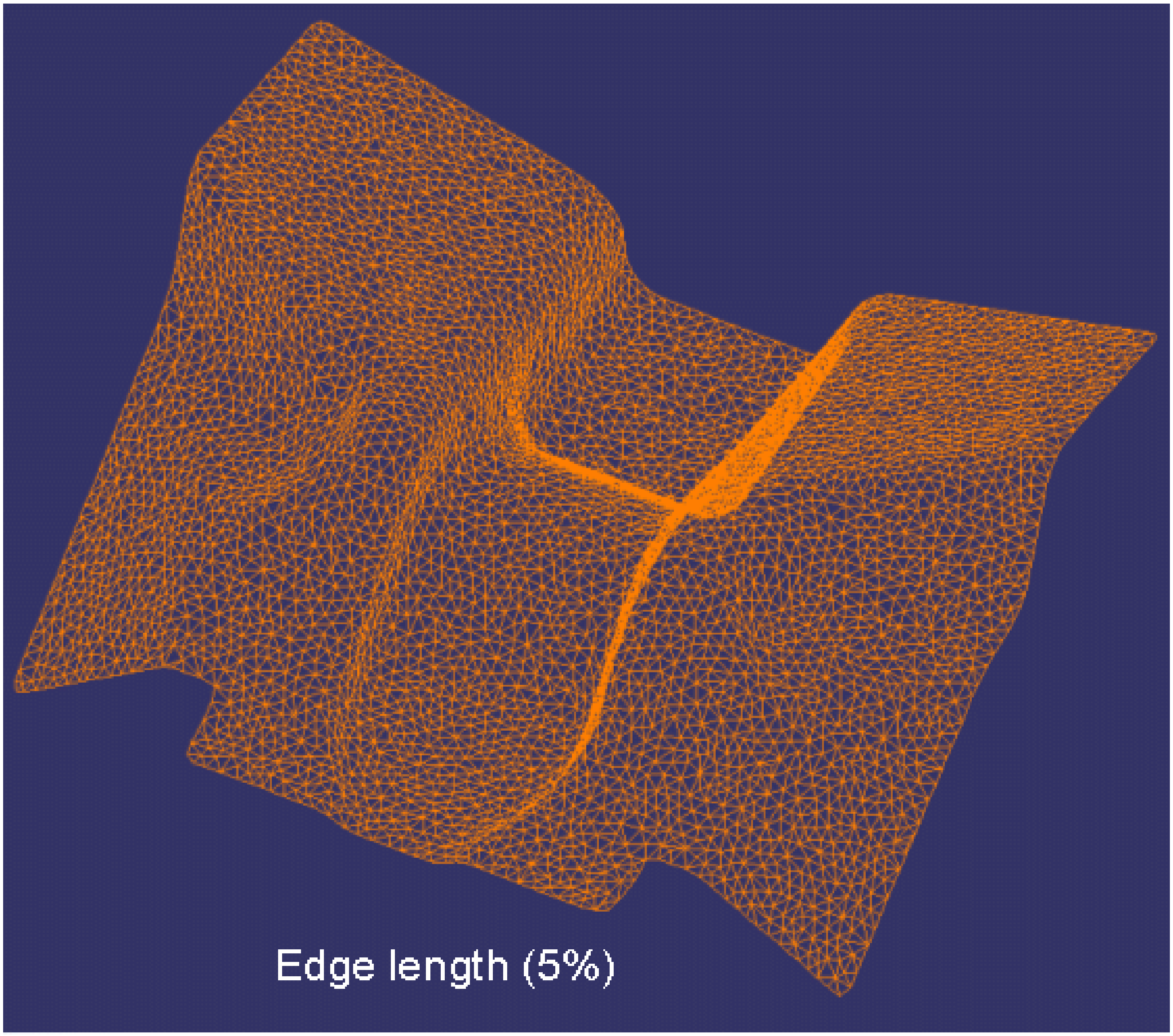
Check **Minimum** if you want to apply a Minimum criterion:

- For a decimation by **Chordal Deviation**, it is the chordal deviation that should not be exceeded during decimation. Decimation stops when the chordal deviation limit has been reached.
- For a decimation by **Edge Length**, it is the minimal length the edges of triangles should have. The decimation stops when all triangle edges are longer than the value set. However, it may be impossible to collapse some edges that will remain below this value.

Check **Target Percentage** if you want to obtain a given final number or percentage of triangles. Enter either the percentage value or the **Target Triangle Count**. Those fields are linked to each other and updated simultaneously.

Current Triangle Count indicates the current number of triangles, either of the original model when you enter the action, or of the result model when you have clicked Apply.





4. You may need to control the decimation of free edges, when a rectangular shape sees its corners cut off after decimation. You can avoid this by checking **Free Edge Deviation**. This activates the maximum allowable deviation that can occur for vertices on the boundary. The resulting decimated boundary will not be at a distance greater than this parameter from the original boundary.

Refining

This task shows how to refine a polygonal mesh.

Refinement is a command increasing the details of a polygonal mesh by splitting its triangles. It can be performed on the entire mesh or a selected region of the mesh.

Refinement preserves the selection: each selected triangle will be removed from the mesh and replaced with two triangles. The two new triangles lie in the same plane as the original triangle. Additional triangles besides those selected may be refined and these will not be placed in the selection after refinement.

Open the Refine1.CATPart document.

1. Click the **Refine** icon .

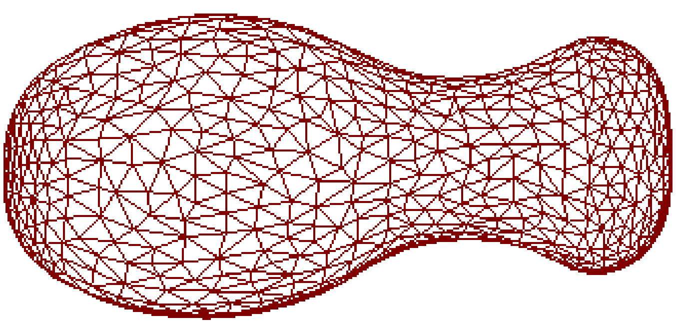
   The Refine dialog box displays.

2. Select the polygonal mesh.

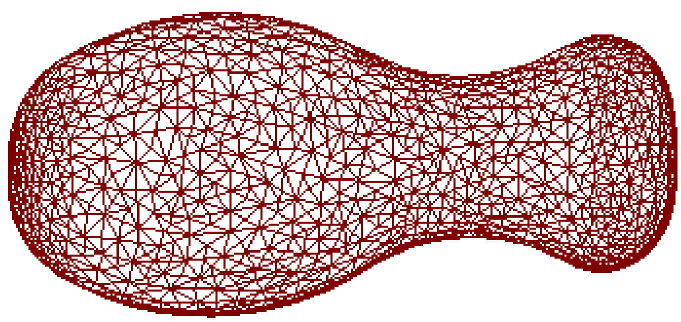
   You can use the **Wireframe** mode from the View toolbar to display the number of triangles.

3. Click **Apply** to preview the refinement.

4. Click OK to exit the command.



Before refinement



After refinement

- You can click **Apply** in the dialog box as many times as desired to perform a more precise refinement.
- You can perform several selections of triangles while in the command.

Smoothing

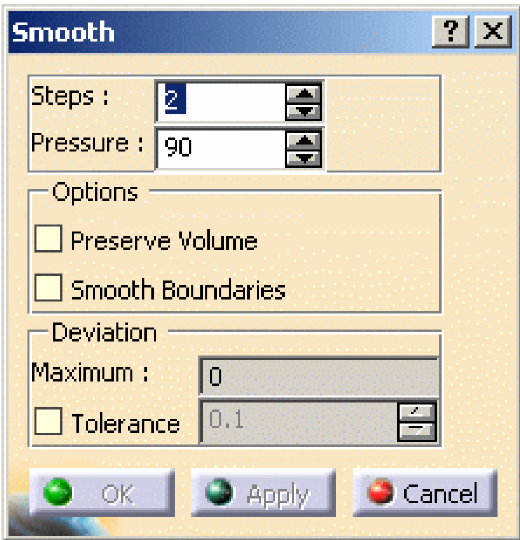
This task shows how to smooth the vertices of a polygonal mesh.

Smoothing can be performed on the entire mesh or a selected region of the mesh.

Open the Bunny1.CATPart document.

1. Click the **Smooth** icon .

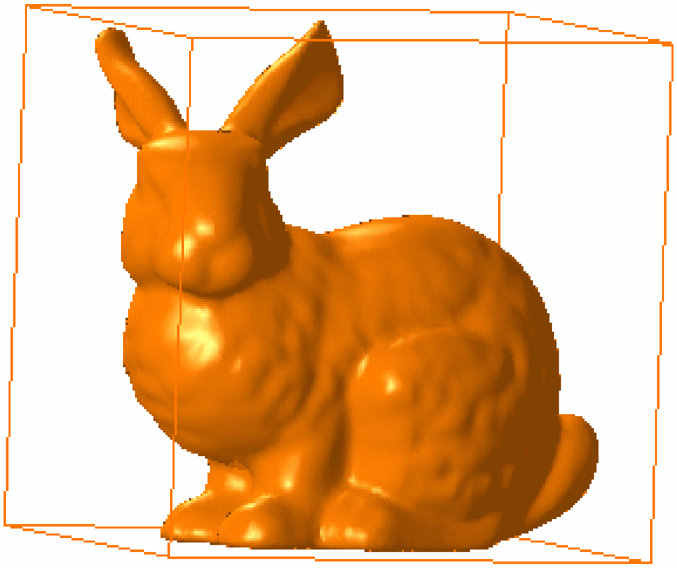
   The Smooth dialog box displays.



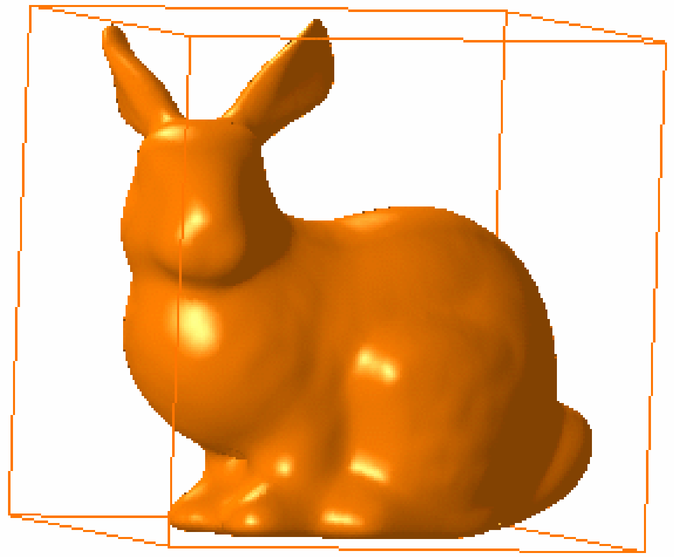
2. Select the polygon.
3. Define the number of **Steps** of the smoothing between 1 and 100.
4. Define the **Pressure** value between 0 and 100.

   A value of 0 means there is no smoothing, and a value of 100 corresponds to the maximum smoothing.
5. Click **Apply** to preview the smoothing.

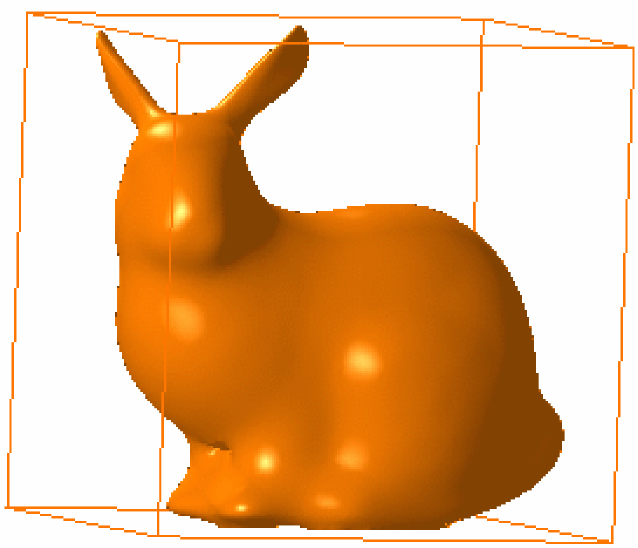
   The Maximum deviation is displayed.
6. Click OK.



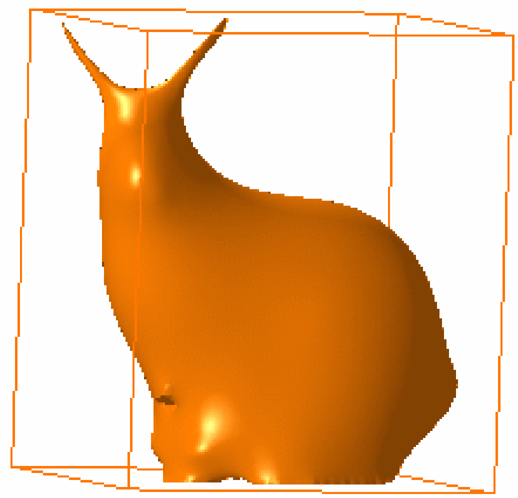
The example above shows a smoothing using the following values:
Steps = 1
Pressure = 10



The example above shows a smoothing using the following values:
Steps = 1
Pressure = 90



The example above shows a smoothing using the following values:
Steps = 40
Pressure = 10



The example above shows a smoothing using the following values:
Steps = 40
Pressure = 90

- The **Preserve Volume** option enables to keep the volume of the polygonal mesh after the smoothing (as you can see in the fourth example, the volume was not kept).
- The **Smooth Boundaries** option enables to smooth the boundaries (free edges) of the polygonal mesh.
- The **Tolerance** value enables to check if any of the vertices moved by a distance greater than the user specified tolerance. If the tolerance is violated, the vertices are restored to their original position and the smoothing is not performed. If successful, the Maximum Deviation of the vertices is displayed.

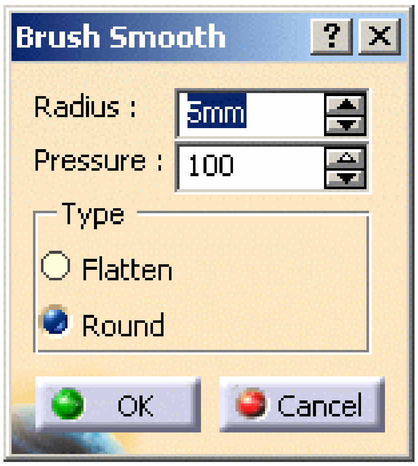
Smoothing With a Brush

This task shows how to locally smooth the polygonal mesh using a brush.

Open the Bunny1.CATPart document.

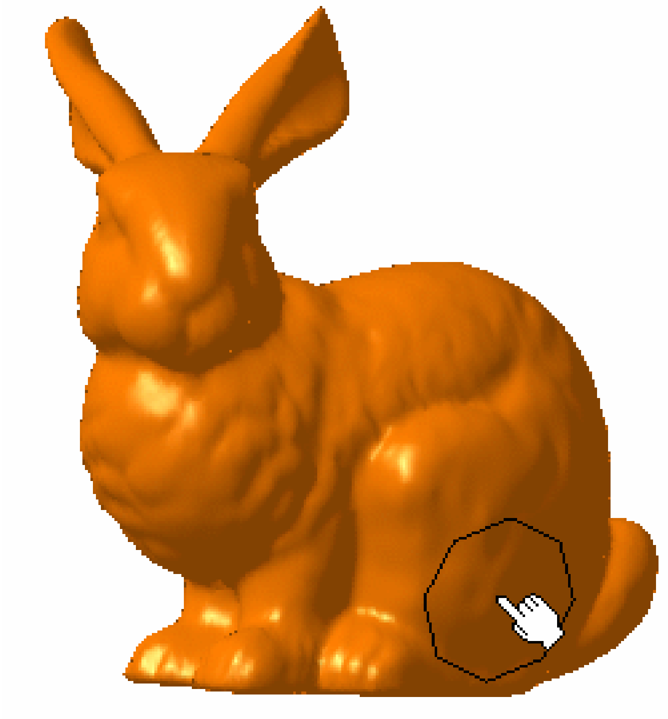
1. Click the **Brush Smooth** icon.

The Brush Smooth dialog box displays.



2. Define the **Radius** value corresponding to the size of the brush.
3. Define the **Pressure** value between 0 and 100.

   A value of 0 means there is no smoothing, and a value of 100 corresponds to the maximum smoothing.



4. Choose the smoothing type:

   - **Flatten**: the mesh tends to become flatter as it is being smoothed.

   - **Round**: the mesh is smoothed while keeping the rounded shapes.

5. Click **Apply** to preview the smoothing.

6. Click OK.

Flatten

Round

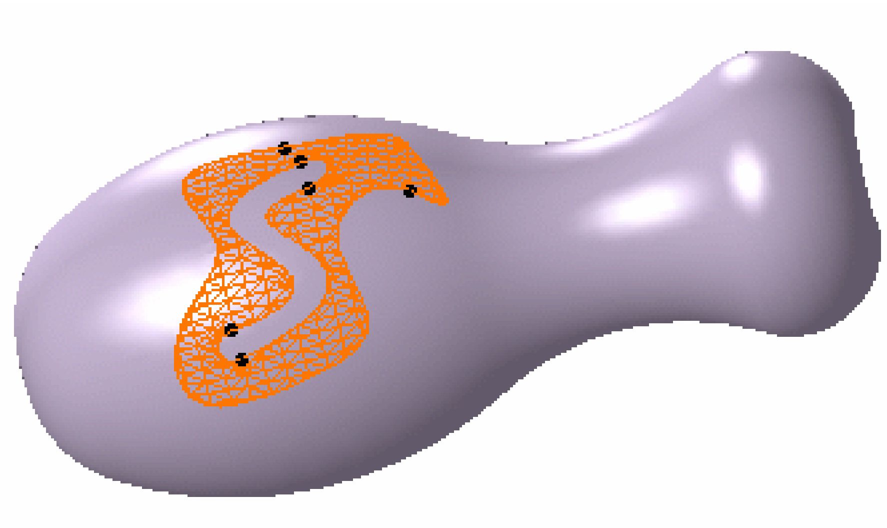
Deleting Triangles

This task shows how to delete selected triangles of a polygonal mesh.

Open the Pin1.CATPart document.

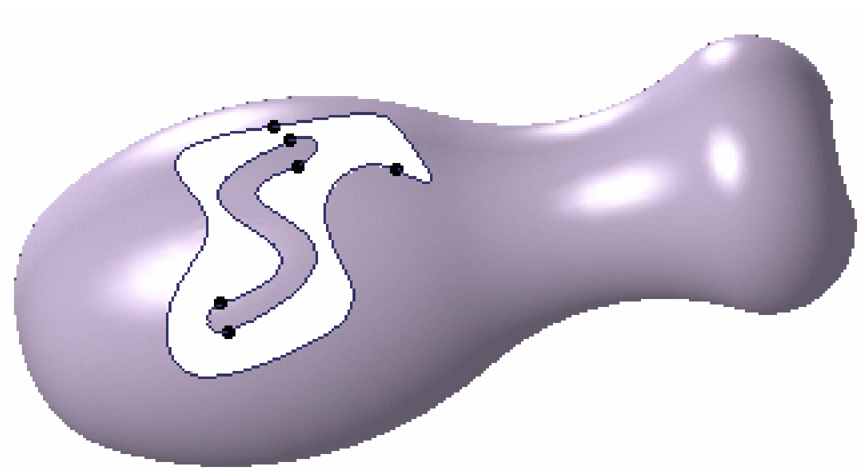
1. Select the triangles to delete using one of the selection components.

   In our example, we chose the Curve Select component.



2. Click the **Delete triangles** icon .

   The selected triangles are deleted from the polygonal mesh.



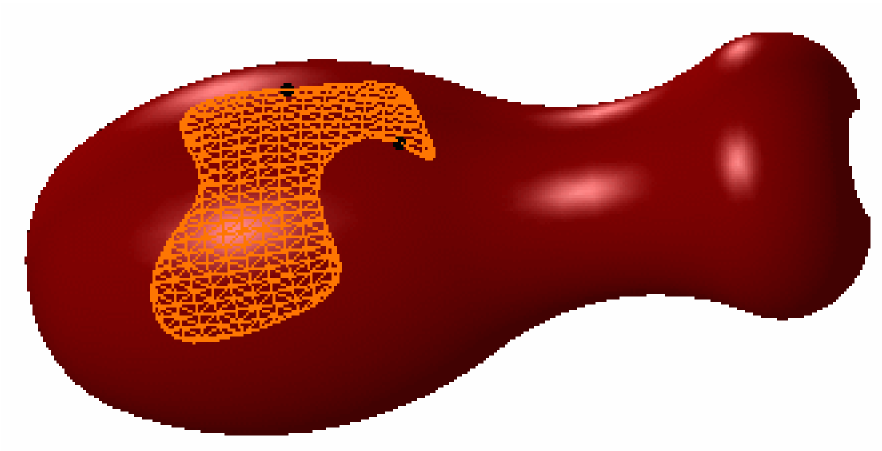
Extracting Triangles

This task shows how to create a mesh by extracting a selection of triangles from a polygonal mesh.

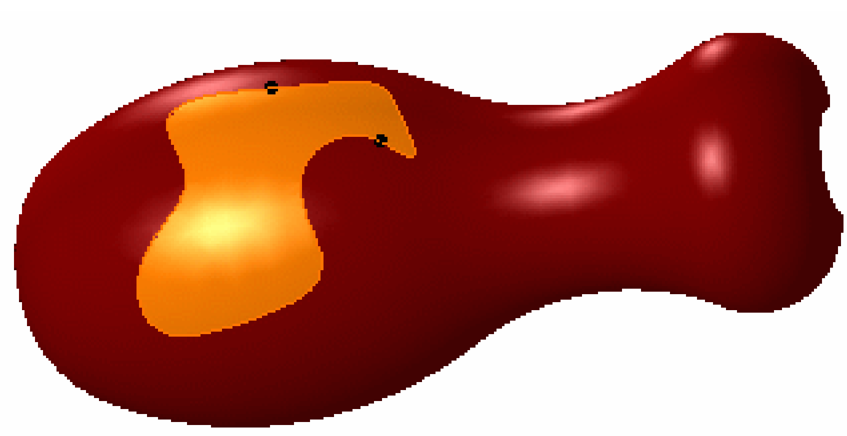
Open the Pin3.CATPart document.

1. Select the triangles to delete using one of the selection components.

   In our example, we chose the Curve Select component.



2. Click the **Extract triangles** icon .



A mesh is created with the selected triangles: they are extracted and deleted from the original mesh and added to a new polygonal mesh.

This new mesh (identified as Polymesh.xxx) is added to the specification tree.

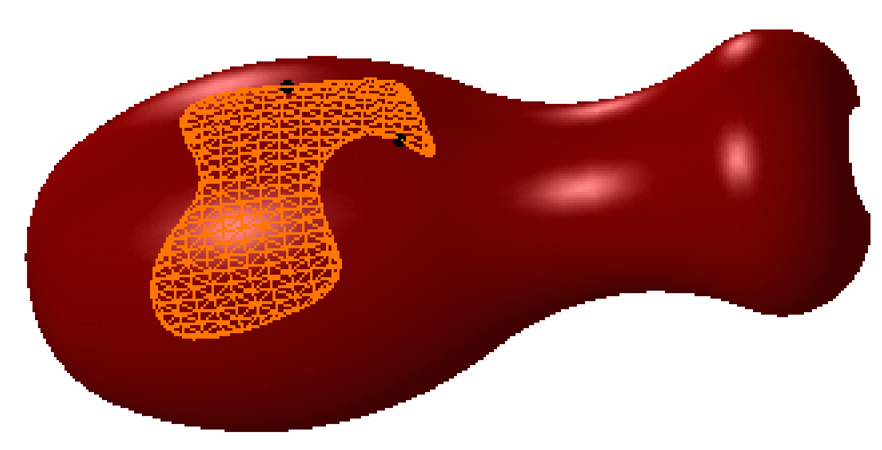
Copying Triangles

This task shows how to create a mesh by copying triangles from of a polygonal mesh.

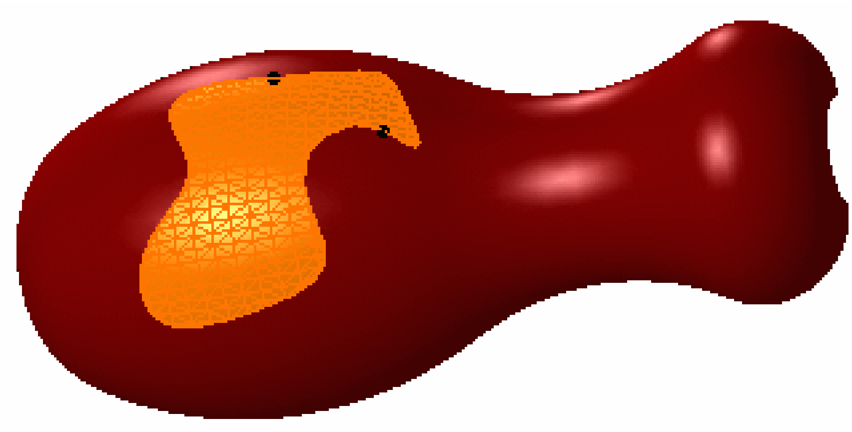
Open the Pin3.CATPart document.

1. Select the triangles to delete using one of the selection components.

   In our example, we chose the Curve Select component.



2. Click the **Copy triangles** icon .



A mesh is created with the selected triangles: they are copied from the original mesh and added to a new polygonal mesh.
This new mesh (identified as Polymesh.xxx) is added to the specification tree.

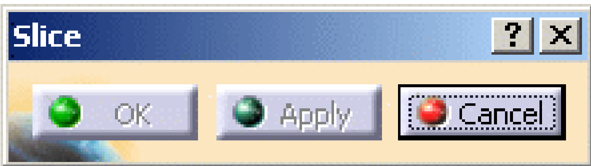
Slicing

This task shows how to slice a polygonal mesh using curves and scans.

Open the Pin3.CATPart document.

1. Click the **Slice** icon.

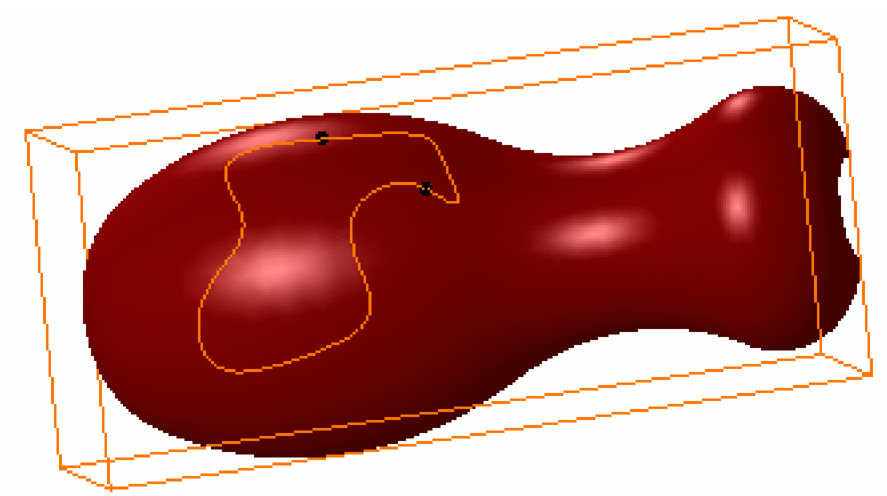
   The Slice dialog box displays.



2. Select a polygonal mesh.

3. Select one or more curves or scans.

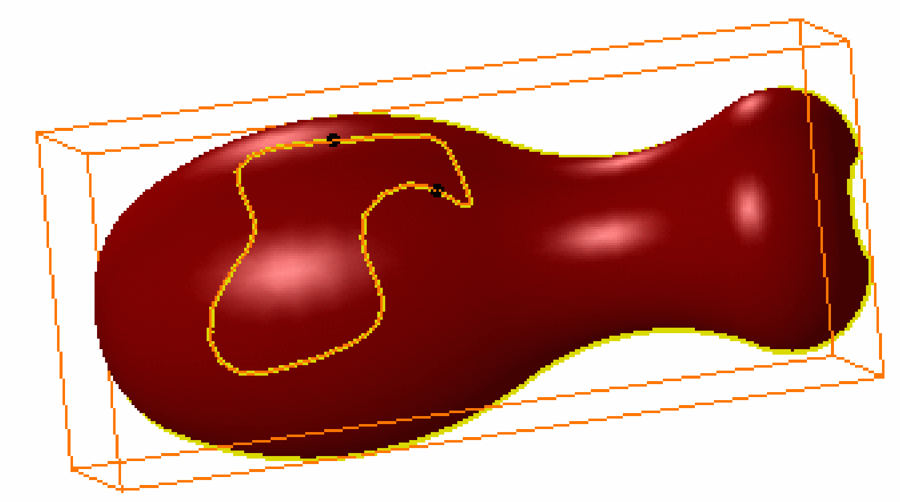
- The curves do not need to lie on the mesh: if they do not lie, they will be projected onto the mesh using a normal projection, before the slicing operation.



4. Click **Apply**.

All boundaries (free edges) are highlighted.

5. Click OK.



A hole is created in the polygonal mesh.

A refinement will be performed on the mesh in the area of the slicing entity (curve or scan).

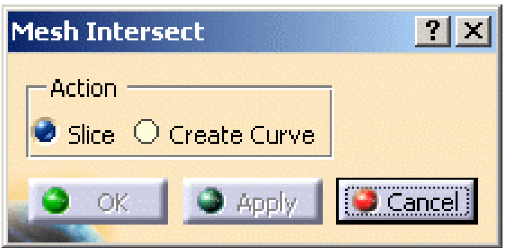
Intersecting Meshes

This task shows how to slice two polygonal meshes along their intersection.

Open the Intersect1.CATPart document.

1. Click the **Mesh Intersect** icon.

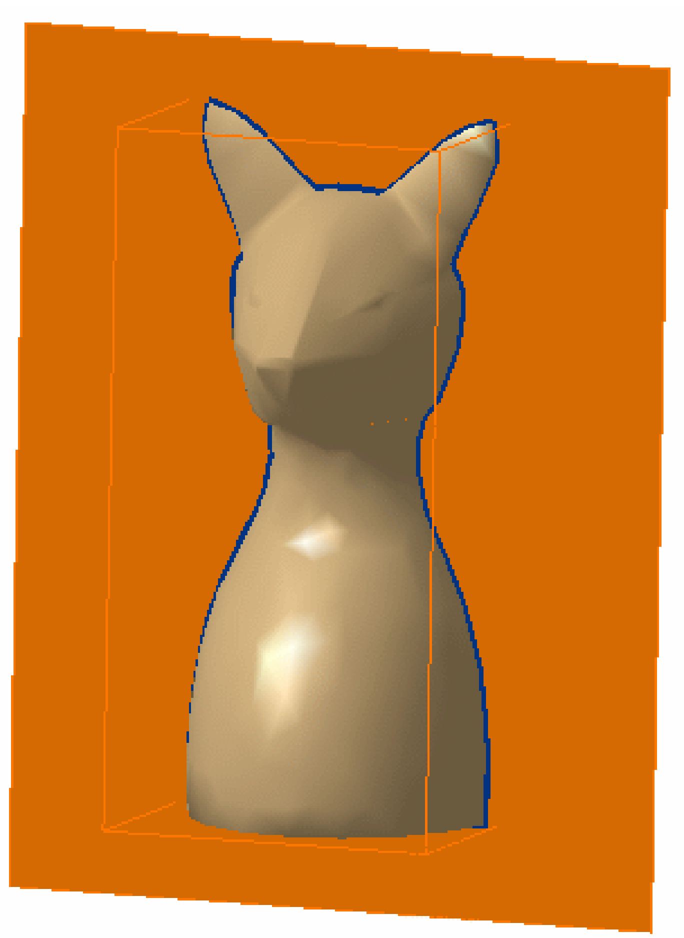
   The Mesh Intersect dialog box displays.



2. Select the first mesh.

3. Press the **Ctrl** key and select the second mesh.

Their intersection is highlighted in blue.

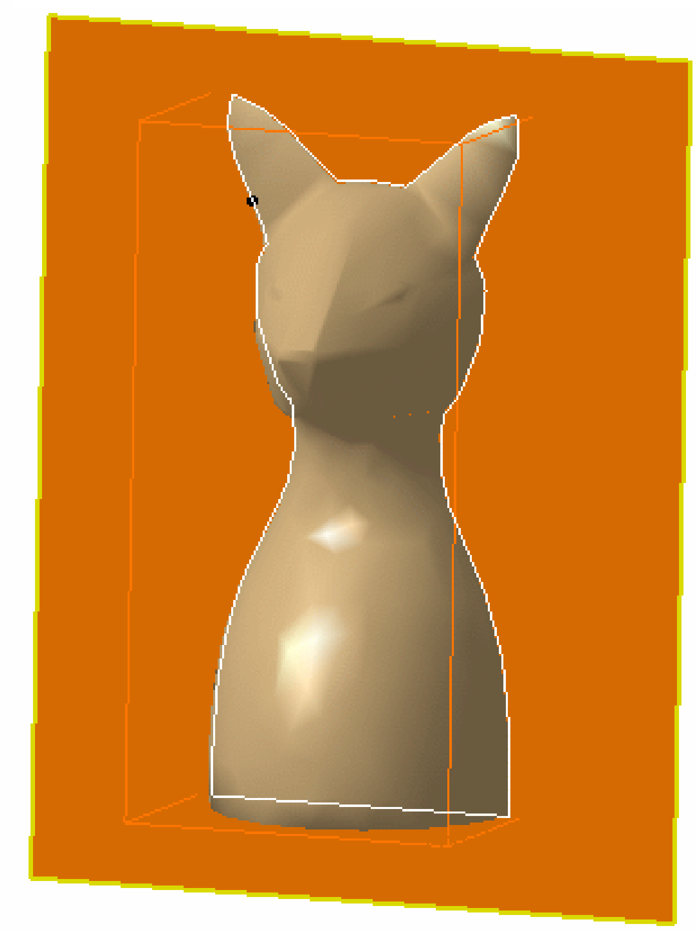


4. Select intersection type:

- **Slice**: meshes are sliced
- **Create Curve**: meshes are not sliced, however curves are created at each intersection. These curves appear in the specification tree as Intersection Curve.xxx.

5. Click **Apply** to preview the intersections.

Slice



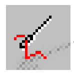
Create Curve

6. Click OK to exit the command.

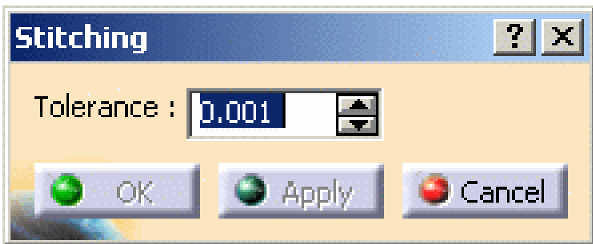
Stitching

This task shows how to stitch two or more polygonal meshes into one mesh, along their free boundaries.

Open the Felix2.CATPart document.

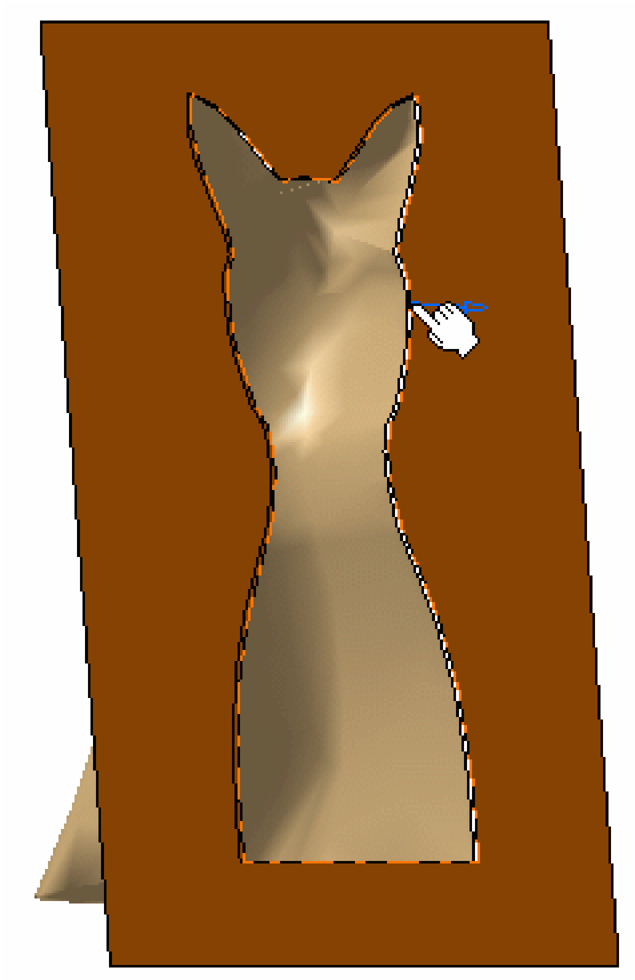
1. Slice the two polygonal meshes as shown in the previous task.
2. Click the **Stitch** icon .

   The Stitching dialog box displays.



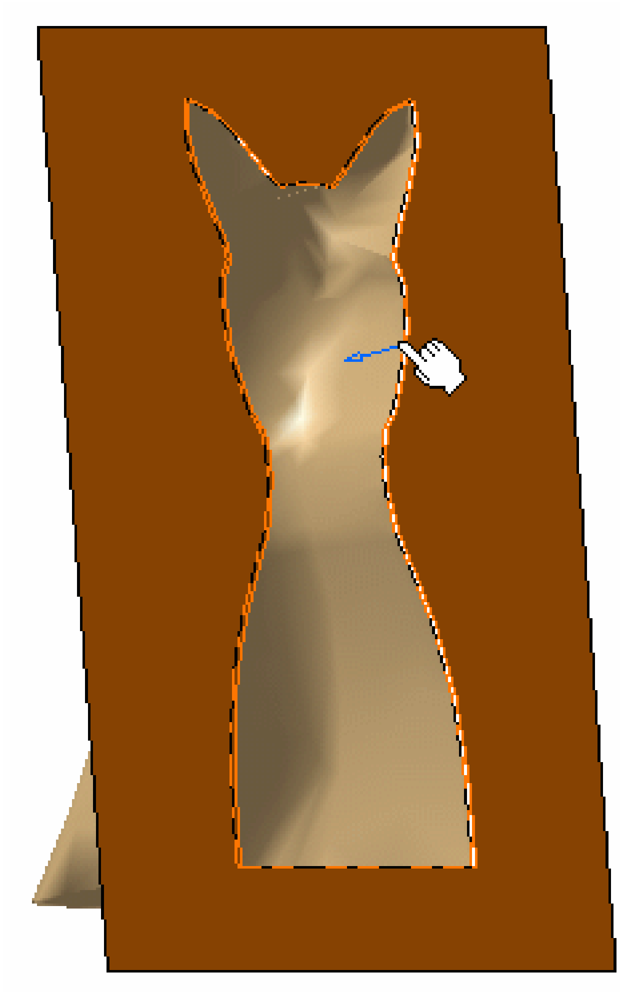
3. Select a hole along the boundary of the first mesh.

The blue arrow shows you which boundary is selected.



4. Holding the **Ctrl** key, select a hole along the boundary of the second mesh.

The blue arrow shows you which boundary is selected.



5. Adjust the tolerance using the spinners (in our scenario, it is set to 2.5 mm).

If a too low value is set, holes still remain.

6. Click **Apply** to preview the stitching.

Meshes are merged and stitched into one mesh.

Here the mesh is displayed as flat. Refer to Display Options and Graphic Properties for further information.

7. Click OK to exit the command.

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

Stitching cannot create non-manifold edges.

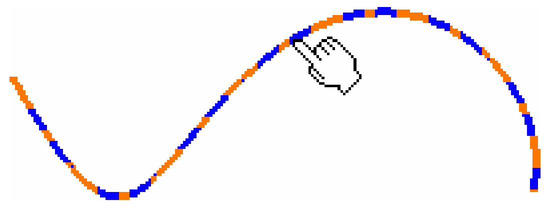
Modeling

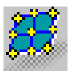
Editing Curves Using Control Points

This task explains how to modify a curve using its control points.

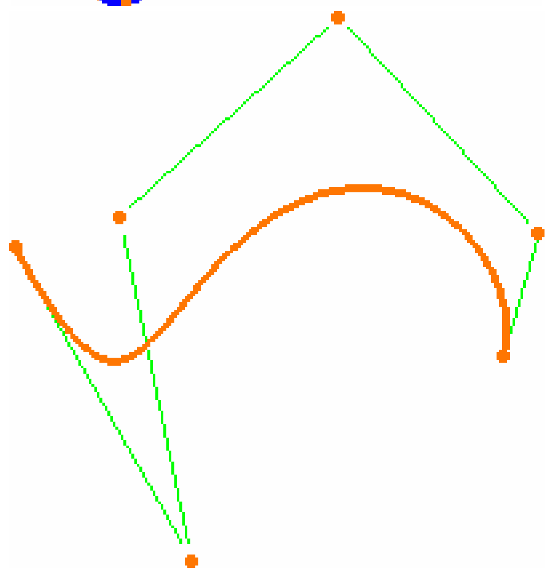
Open the EditControlPoints1.CATPart document.

1. Select the curve you wish to edit.



2. Click the **Control Points** icon .

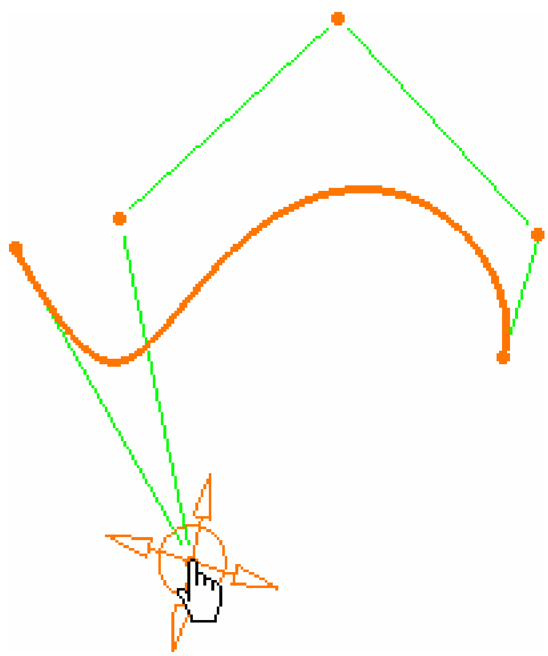
Control points and lines are displayed, along with the 3D compass and the curve degree.

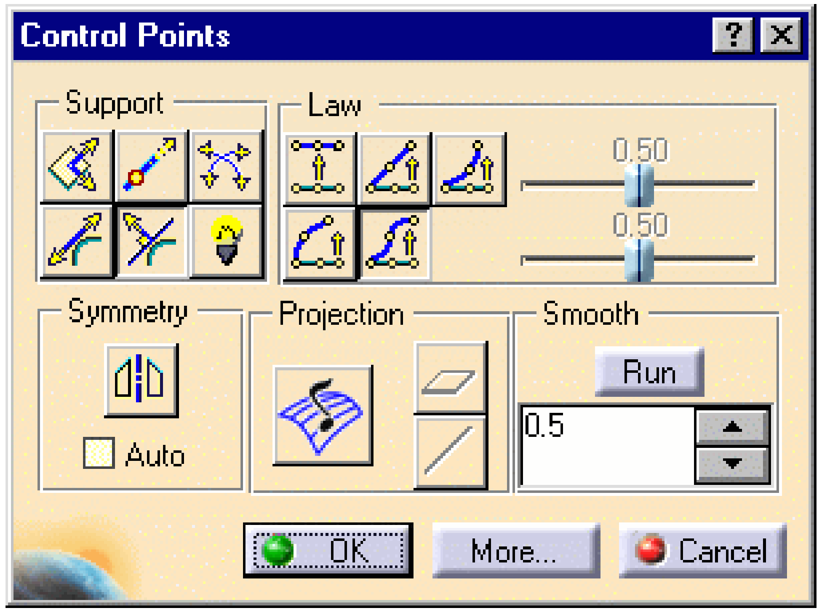


By default all control points and mesh lines are selected. Click a specific point to deform the surface at this point only, or select a set of points using the Ctrl-key or Shift-key while clicking (multi-selection capabilities). The same applies to mesh lines.

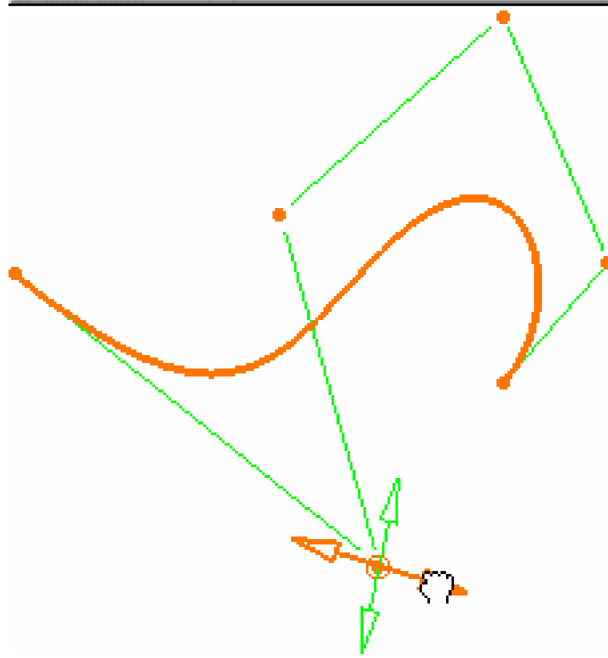
3. Move the pointer over a point or a line.

Arrows are displayed according to the **Support** and **Law** options active in the Control Points dialog box.

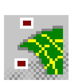




- **Support** defines the type of translation to be applied
- **Law** defines the type of deformation that is to be applied when several control points have been selected.

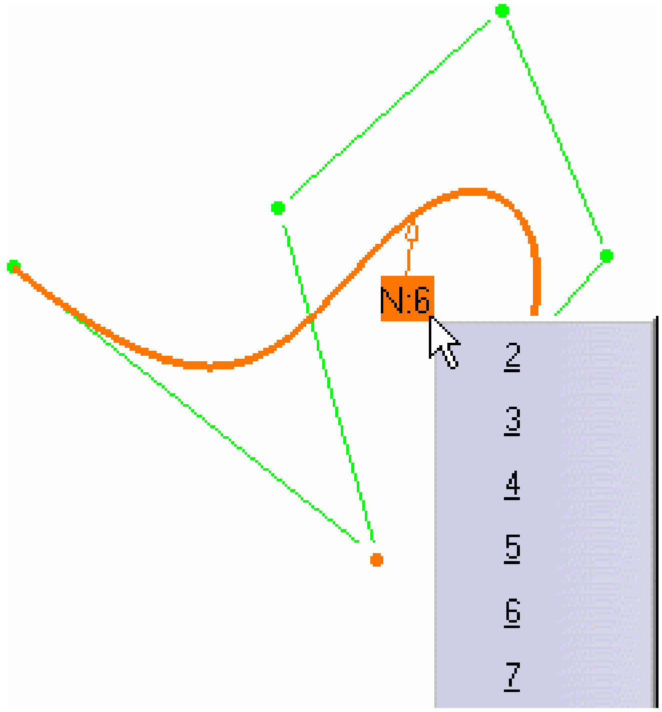


4. Pull on the arrow matching the direction in which you want to deform the curve.

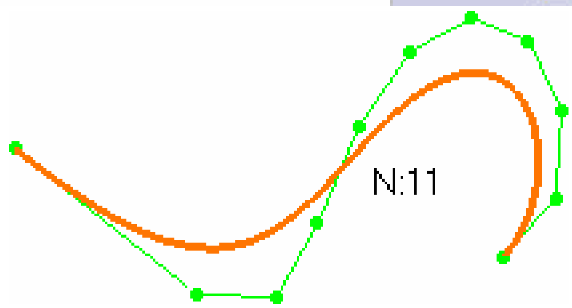
5. Use the Dashboard to display the order number directly on the curve, using the **U, V Orders**  icon.

The value is displayed on the curve.

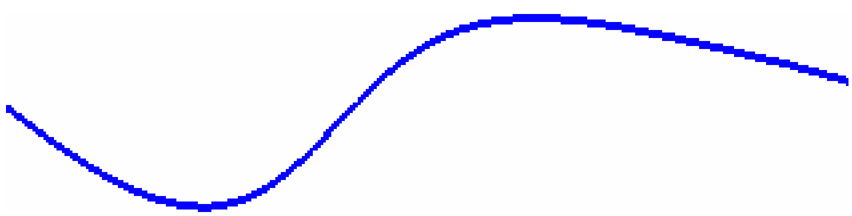
6. Use the contextual menu to choose the order number for the curve.



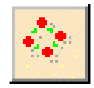
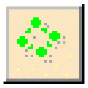
If you increase the order number to 11, the result is:



7. Click OK in the Control Points dialog box to validate the modifications.



- Use the Control Points dialog box to modify the curve according to certain predefined laws.
- To perform a quick analysis of the mesh line inflection, choose the **Inflections** checkbox.

- Further modification options are available from the **More...** button of the Control Points dialog box:
  - Freezing of the selected points (no other can be selected) and of the Support option
  - Dynamic display of the initial curve and of the delta as you pull on the control points.
  - Global selection/de-selection of control points using the **Select All**  and **De-Select All**  icons, without having to click the geometry.

Projecting Curves

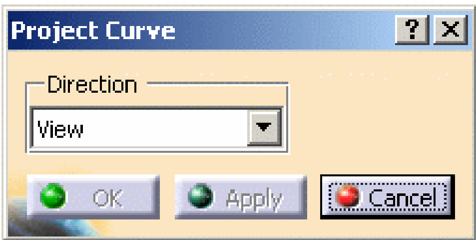
This task shows you how to project curves on surfaces.

Open the Project1.CATPart document.

1. Press the **Ctrl** key and select the curves you want to project.

2. Click the **Project Curve** icon .

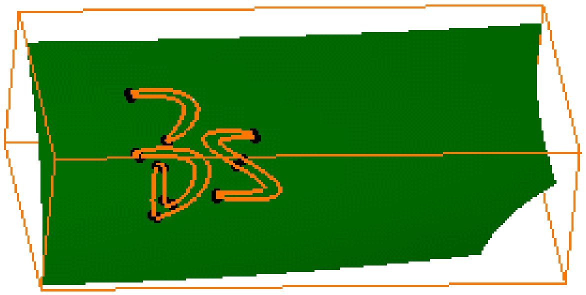
   The Project Curve dialog box displays.



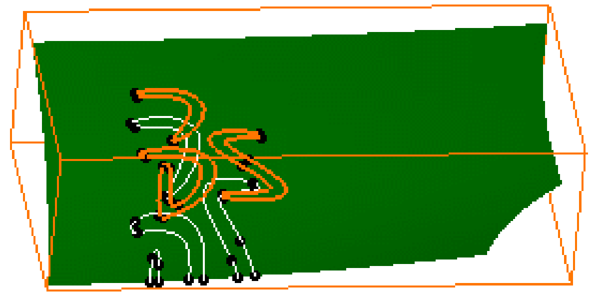
3. Choose the projection orientation from the Projection dialog box:

- **View**: projection according to the current view
- **Compass**: projection according to the compass
- **Normal**: projection normal to the surface

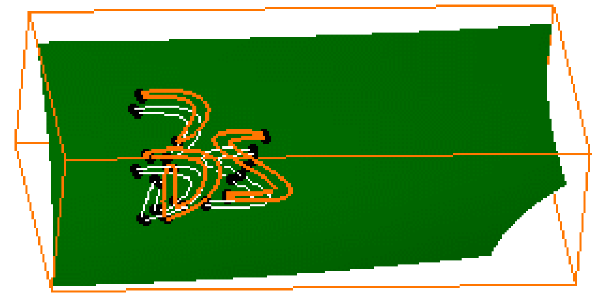
4. Press the **Ctrl** key and select the surface or set of surfaces on which the curve(s) should be projected.
5. Click Apply to preview the projected curves on the target surface.



Projection according to the current view



Projection according to the compass orientation



Projection according to the normal to the surface

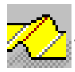
6. Click OK to create the projected curves.

Curves (identified as Curve.xxx) are added to the specification tree.

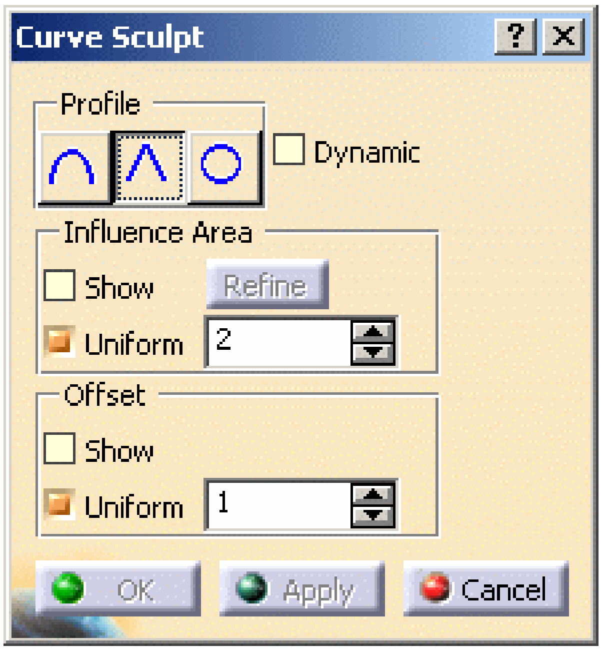
Sculpting Curves

This task shows you how to sculpt a polygonal mesh a curve or a set of connected curves.

Open the Pin4.CATPart document.

1. Click the **Curve Sculpt** icon .

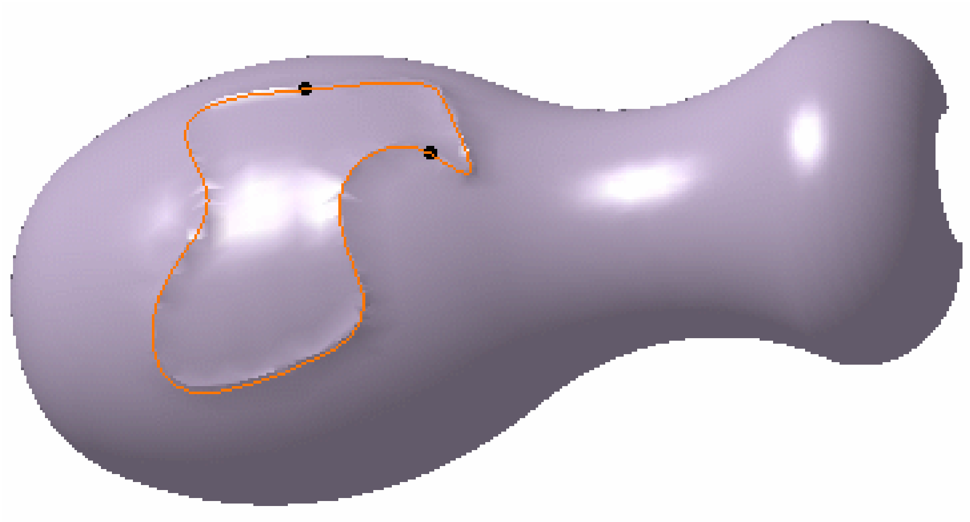
   The Curve Sculpt dialog box displays.



2. Holding the **Ctrl** key, select the curves to sculpt with.

3. Select the polygonal mesh.

4. Click **Apply**.



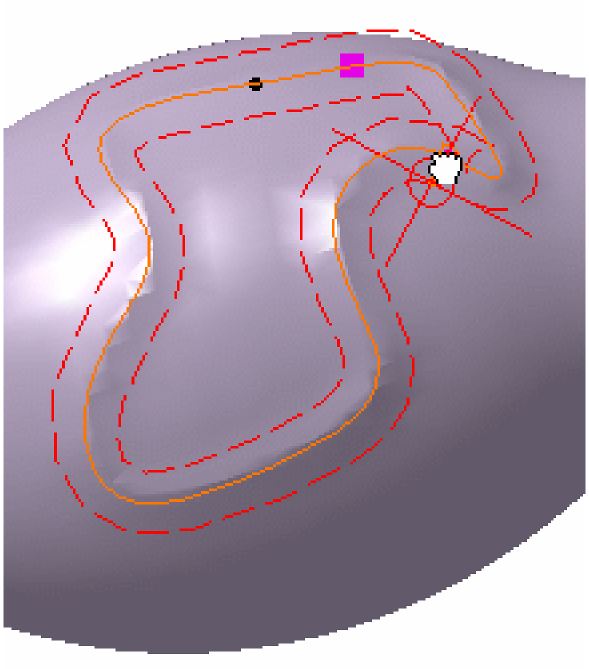
5. Define the profile:

- **Smooth**
- **Sharp** (used in our example)
- **Circular**

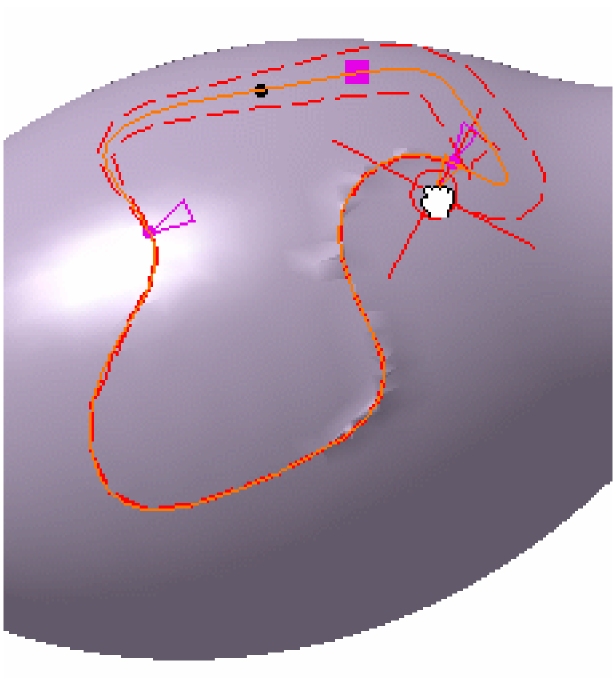
You can check the **Dynamic** button to automatically sculpt the mesh on the fly.

6. Define the influence area around the curves:

- Check the **Show** button to display the area that is affected by sculpting on the polygonal mesh. Use the manipulators to define the area (as shown in the picture)
- Click **Refine** to refine the sculpt by adding triangles
- Check the **Uniform** button and use the spinners if you wish to force a constant transition width along the curves.

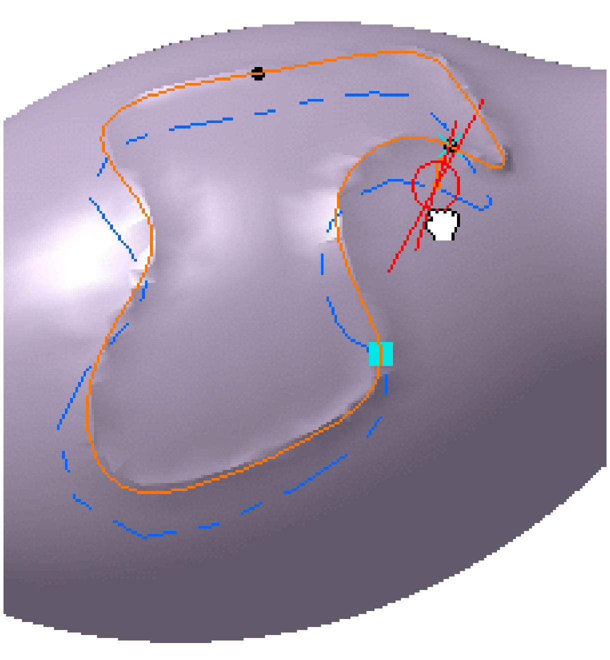


If you uncheck the **Uniform** button, you are able to define the influence area curve by curve.
Simply position the manipulator on the curve whose area you wish to modify and drag the manipulator to the desired transition width.

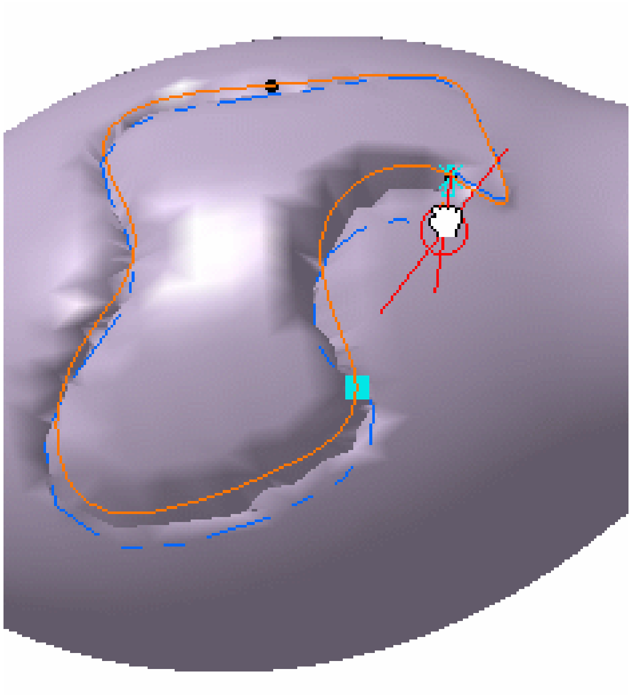


7. Define the offset:

- Check the **Show** button to display the change to the sculpting curve before applying it to the polygonal mesh. Use the manipulators to define the offset (as shown in the picture)
- Check the **Uniform** button and use the spinners if you wish to force a constant offset.

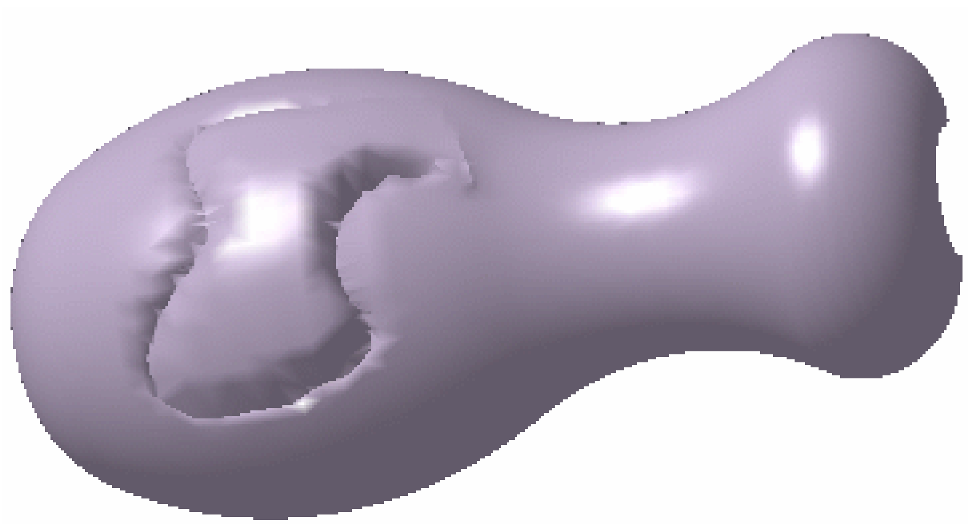


If you uncheck the **Uniform** button, you are able to define the offset curve by curve. Simply position the manipulator on the curve you wish to modify the offset and drag the manipulator to the desired offset.



8. Click OK to end the sculpt.

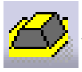
Here is the result:



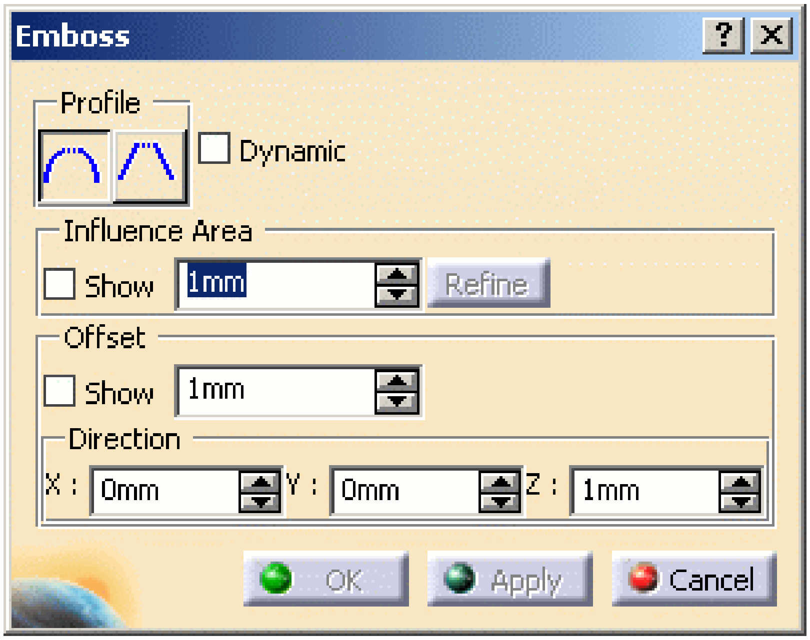
Embossing

This task shows how to emboss a polygonal mesh using curves.

Open the Pin4.CATPart document.

1. Click the **Emboss** icon .

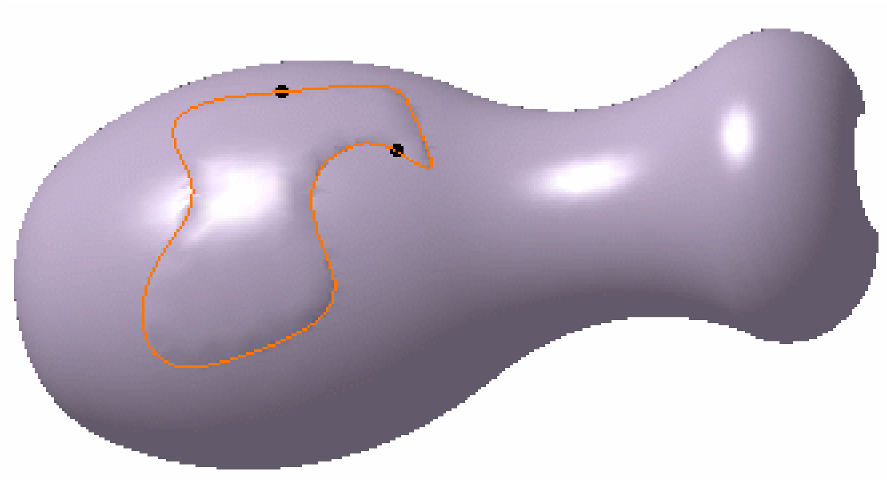
   The Emboss dialog box displays.



2. Holding the **Ctrl** key, select the curves to be used in embossing.

3. Select the mesh to emboss (you can select inside or outside the curves, in our example we selected outside)

4. Click **Apply**.



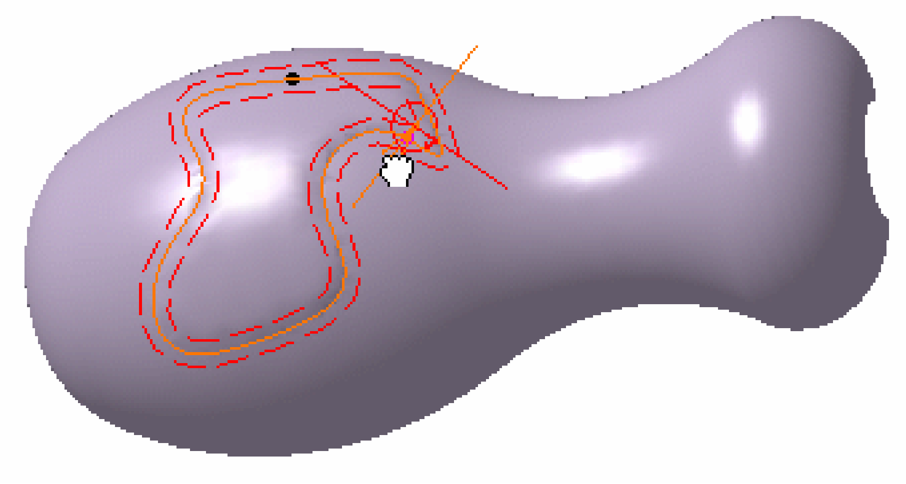
5. Define the profile:

- **Smooth** (used in our example)
- **Sharp**

You can check the **Dynamic** button to automatically sculpt the mesh on the fly.

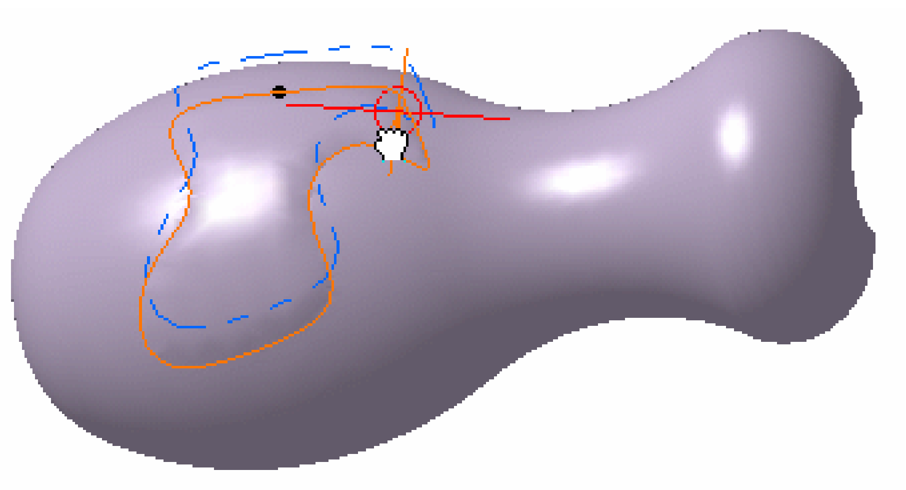
6. Define the influence area around the curves:

- Check the **Show** button to display the area that is affected by embossing on the polygonal mesh. Use the manipulators to define the area (as shown in the picture)
- Click **Refine** to refine the embossed area by adding triangles, therefore increasing the mesh resolution.
- Use the spinners to force a constant transition width along the curves.



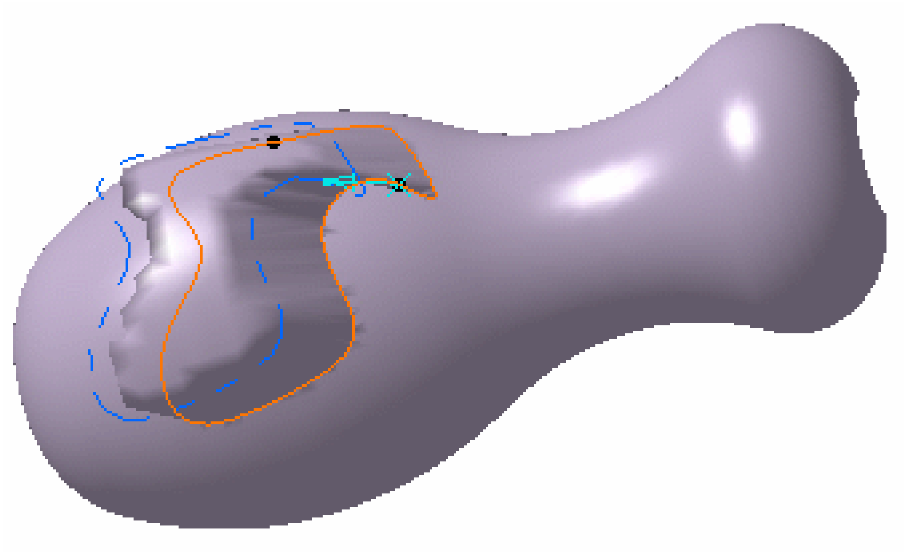
7. Define the offset:

- Check the **Show** button to display the change to the curves to be embossed before applying it to the polygonal mesh. Use the magenta arrow to define the offset (as shown in the picture)
- Use the spinners to force a constant offset.



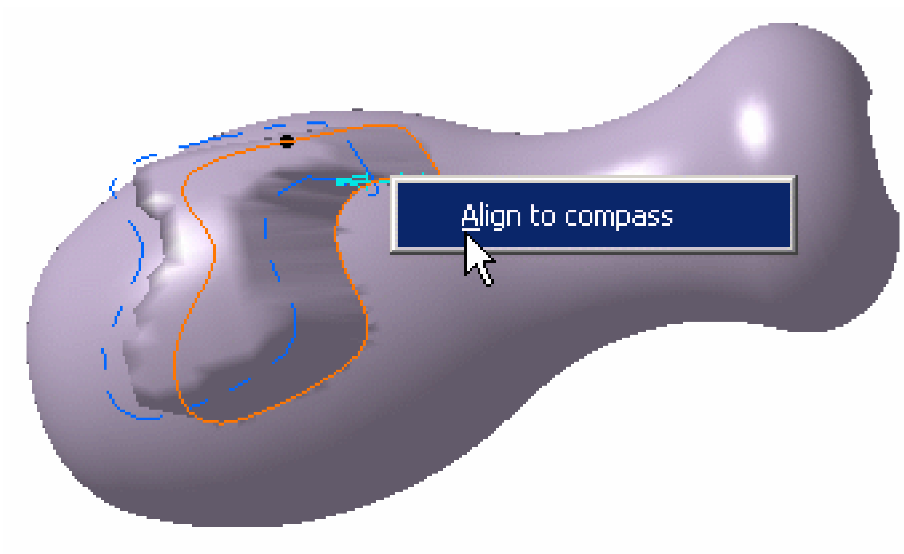
You can change the offset direction, using:

- either the spinners or the cyan arrow to define a value for X, Y, and Z. Here is an example with X= -50, Y=0, and Z=100.



- the compass: Adjust the compass to the desired direction by moving the compass on the selected mesh, then right-click the offset arrow and select **Align to compass** contextual menu.
  The offset direction is now aligned with the compass direction.

 Make sure the **Show** button is checked.



8. Click OK to exit the command.

Selecting multi-loops

It allows you to emboss a more complex shape on the mesh.

Open the EmbossMultiCurves1.CATPart document.

1. Click the **Emboss** icon .

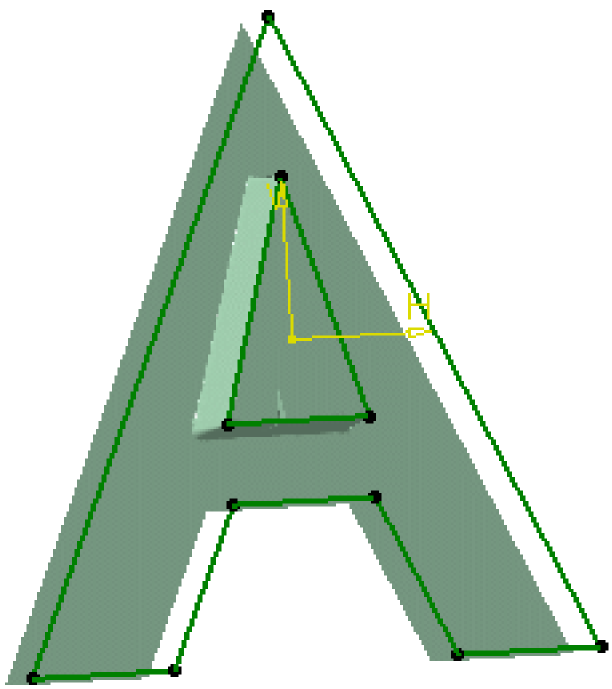
   The Emboss dialog box displays.

2. Holding the **Ctrl** key, successively select the sketches of the outer loop and the inner loop.

3. Select the mesh to emboss outside the inner loop and inside the outer loop.

4. Set the **Offset** as 10mm.

5. Click **Apply**.



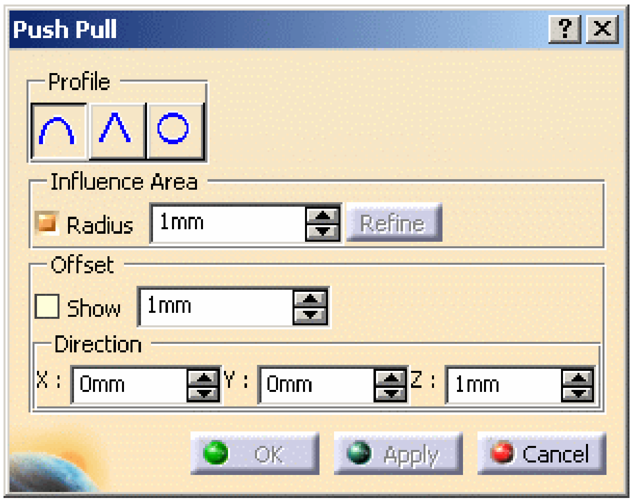
Pushing / Pulling a Mesh

This task shows how to locally modify a polygonal mesh by pushing or pulling a point on the mesh.

Open the PushPull1.CATPart document.

1. Click the **Push/Pull** icon .

The Push Pull dialog box displays.



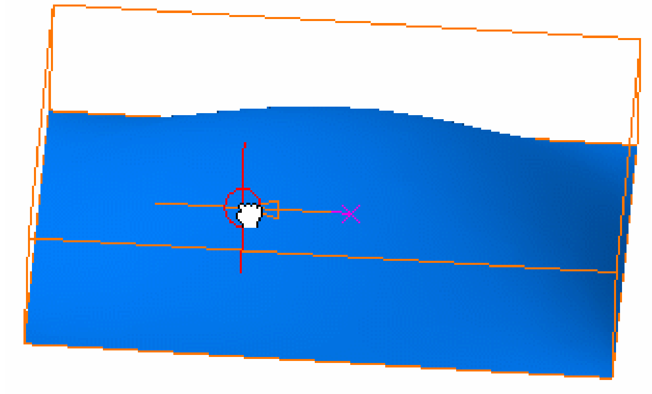
2. Define the profile of the sculpted area:
   - **Smooth**
   - **Sharp**
   - **Circular**

The sculpted area can be defined by:

- ## a point

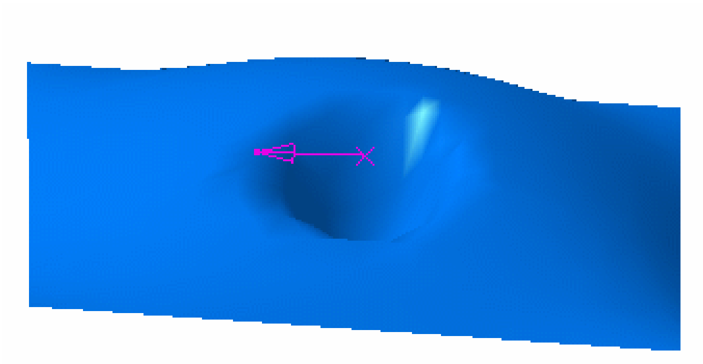
1. Select a point on the mesh.
2. Define the influence area around the point:

- Check the **Radius** button to define a constant influence area (that is the circle around the center point) and display the magenta arrow in the geometry.
- Use the spinners to define the push/pull magnitude.
- Use the arrow to define the push/pull direction.



3. Click **Apply** to preview the push/pull.

The Radius button is greyed out.



- one or more curves forming a closed loop
- one or more curves and a part of the mesh boundary
- the mesh boundary

 Make sure the **Radius** button is unchecked.

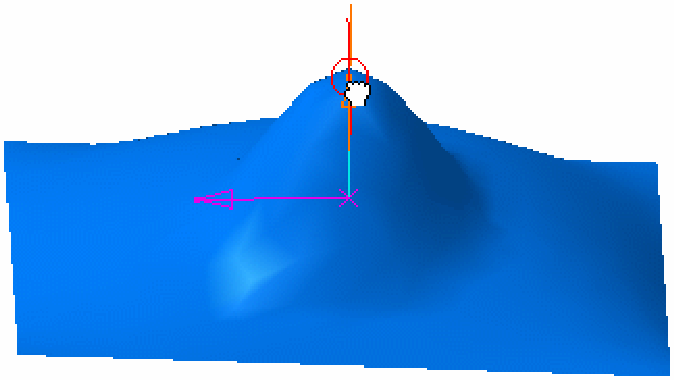
1. Select the curve(s) or mesh boundary (hold the **Ctrl** key if you selected several curves).

   In our scenario, we selected three curves.

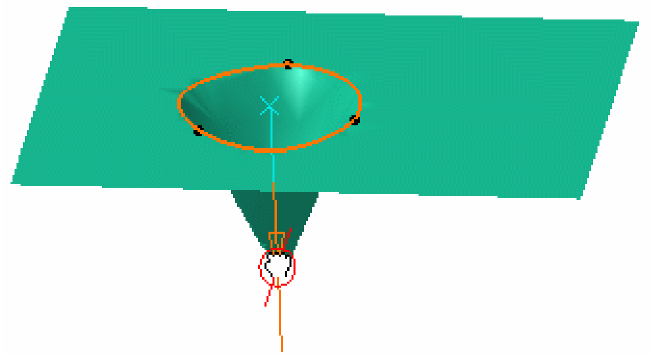
2. Click a point inside the curve(s) or boudary.
3. Click **Apply**.

3. Define the offset:

- Check the **Show** button to display the cyan arrow in the geometry.
- Use either the spinners or the arrow to define the offset.

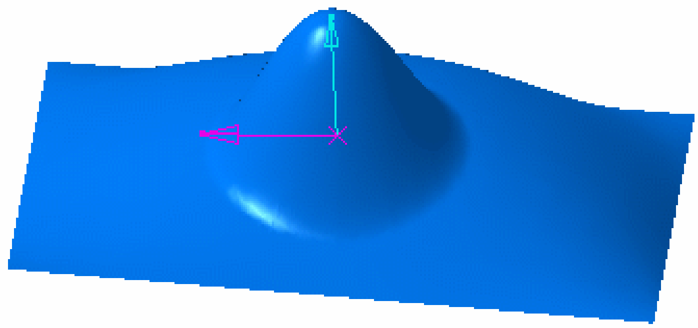


Point selection

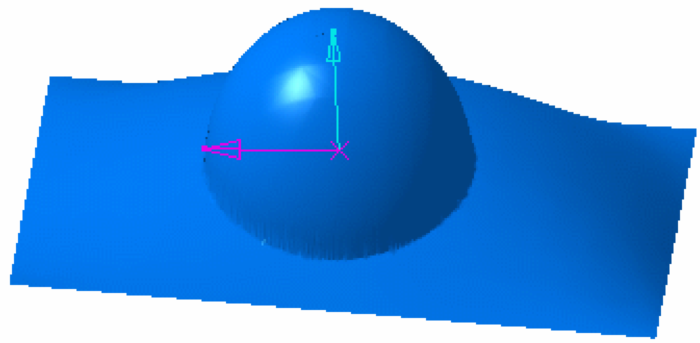


Three-curve selection

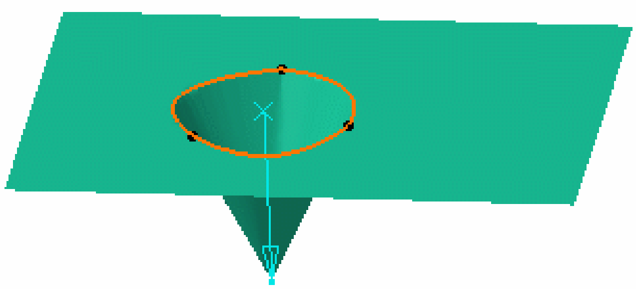
You can click **Refine** to refine the sculpt by adding triangles, therefore increasing the mesh resolution.



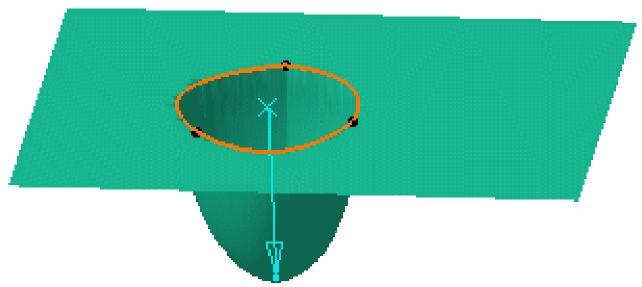
Point selection with Smooth profile



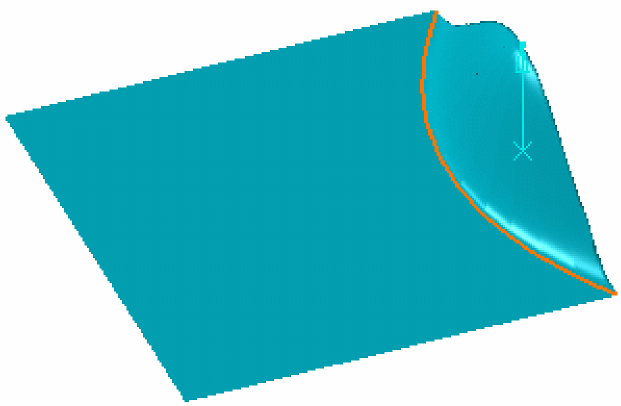
Point selection with Circular profile



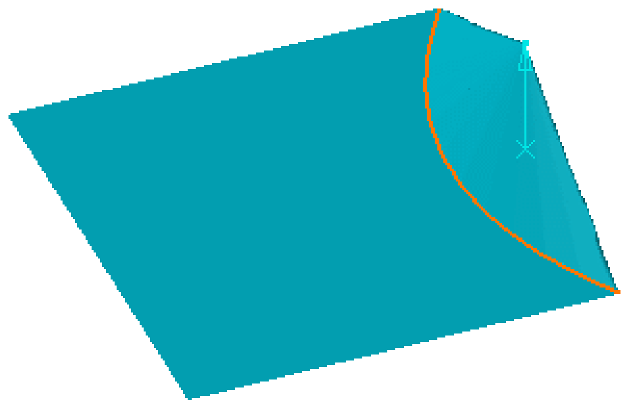
Three-curve selection with Sharp profile



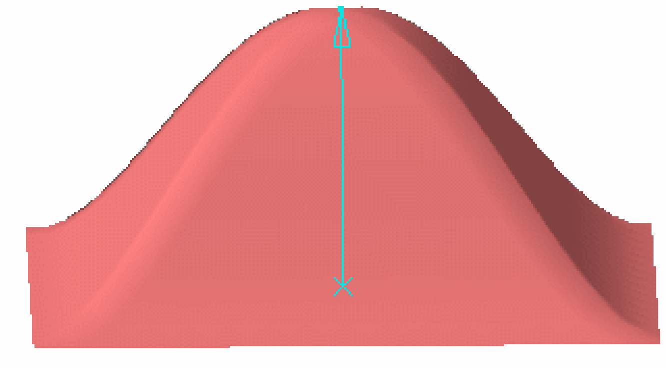
Three-curve selection with Circular profile



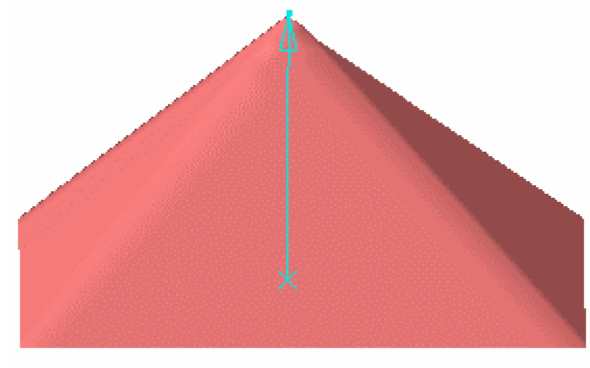
One curve & mesh boundary selection with Smooth profile



One curve & mesh boundary selection with Sharp profile



Mesh boundary selection with Smooth profile

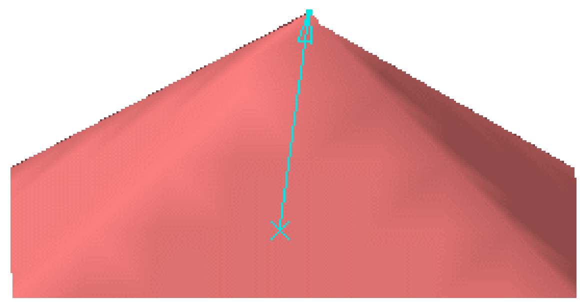


Mesh boundary selection with Sharp profile

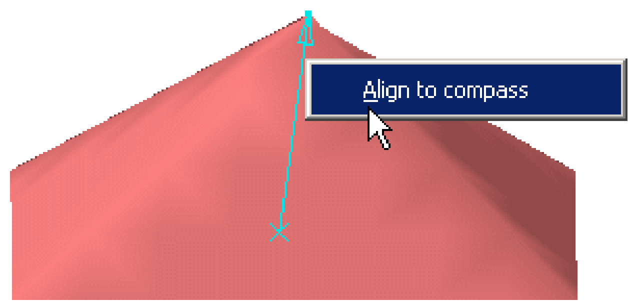
4. Click OK to exit the command.

You can define the offset direction using:

- either the spinners or the cyan arrow to define a value for X, Y, and Z.
  Here is an example with X= 10, Y=3, and Z=25.



- the compass: Adjust the compass to the desired direction by moving the compass on the selected mesh, then right-click the offset arrow and select **Align to compass** contextual menu.
  The offset direction is now aligned with the compass direction.



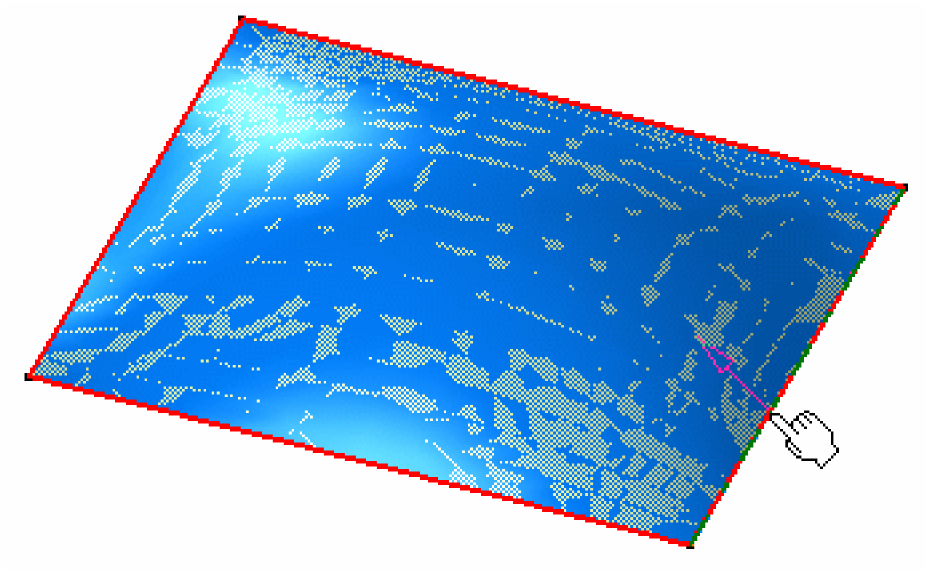
Sculpting With Surfaces

This task shows how to modify a portion of a polygonal mesh with a NURBS surface.

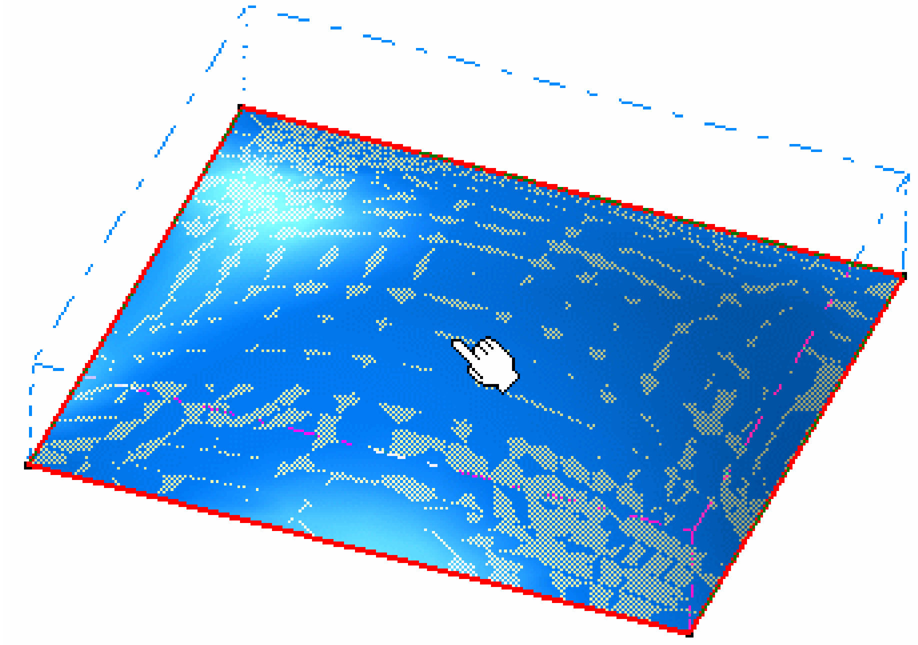
Open the SurfaceSculpt1.CATPart document.

1. Click the **Surface Sculpt** icon .

2. Press the **Ctrl** key and select four curves that form the closed contour of the region to be sculpted.



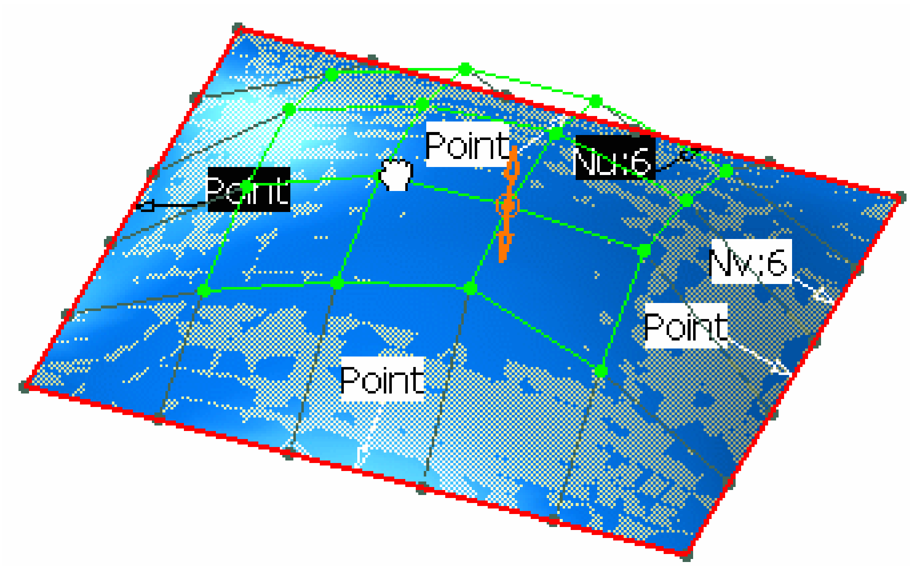
3. Click on the polygonal mesh inside the curves.



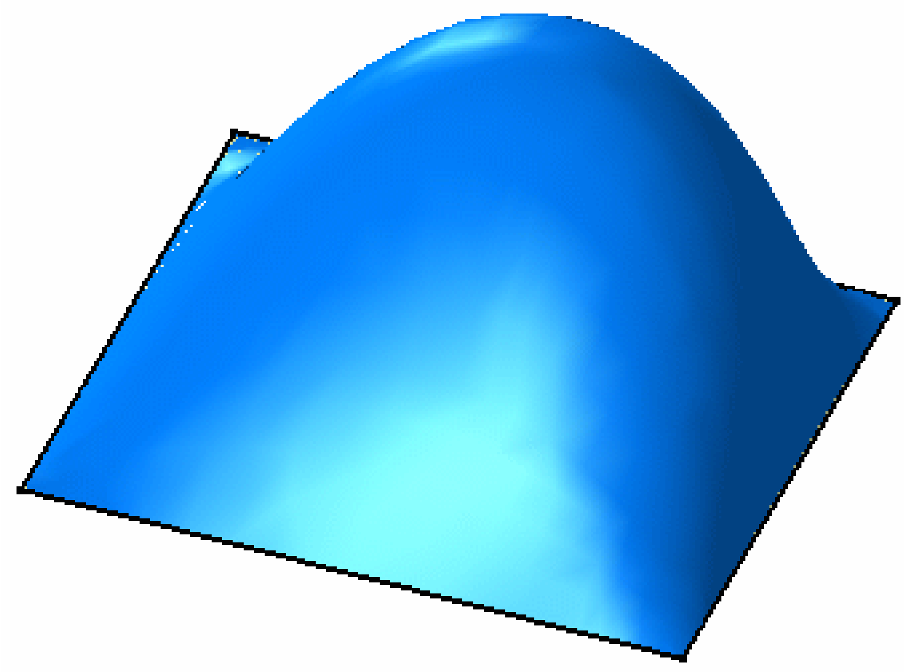
Control points are displayed as well as the Control Points dialog box.

4. Sculpt the polygonal mesh using the Control Points tool.

For further information, please refer to the Editing Surfaces Using Control Points chapter in the *FreeStyle Shaper, Optimizer, & Profiler* documentation.



5. Click OK in the Control Points dialog box.



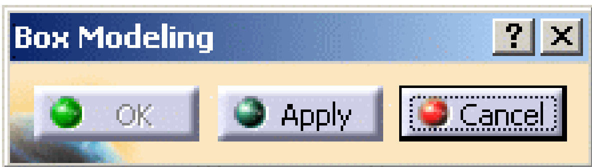
Modeling Using a Box

This task show to apply global deformation to one mesh using a box.

Open the Felix1.CATPart document.

1. Click the Box Modeling icon .

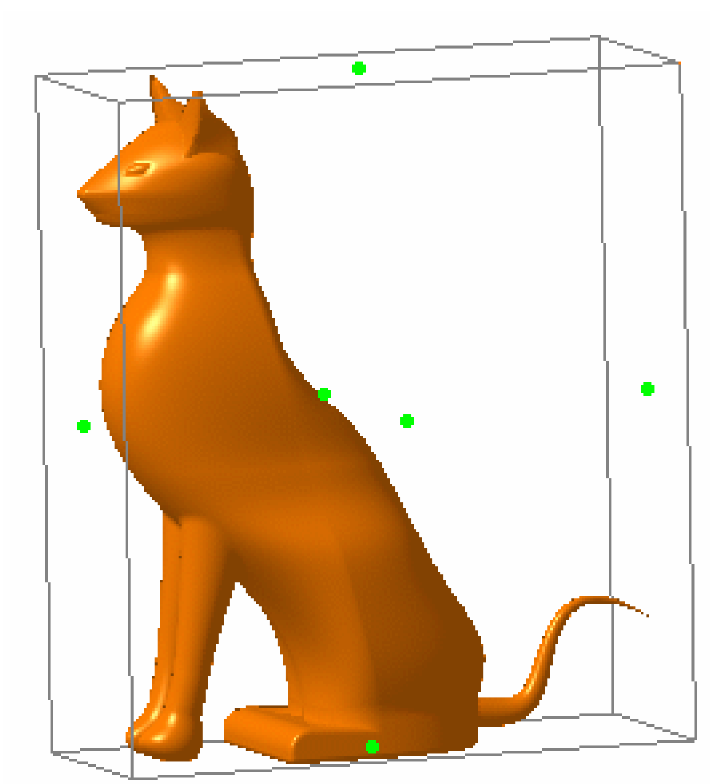
The Box Modeling dialog box displays.



2. Select the mesh to be deformed.

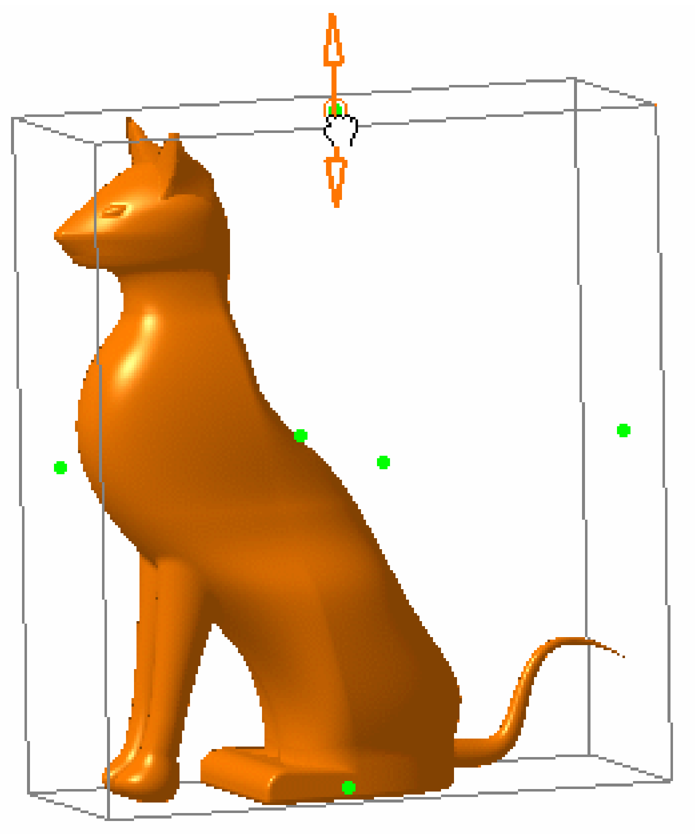
A box frames the mesh.

You can also select the mesh before entering the command.

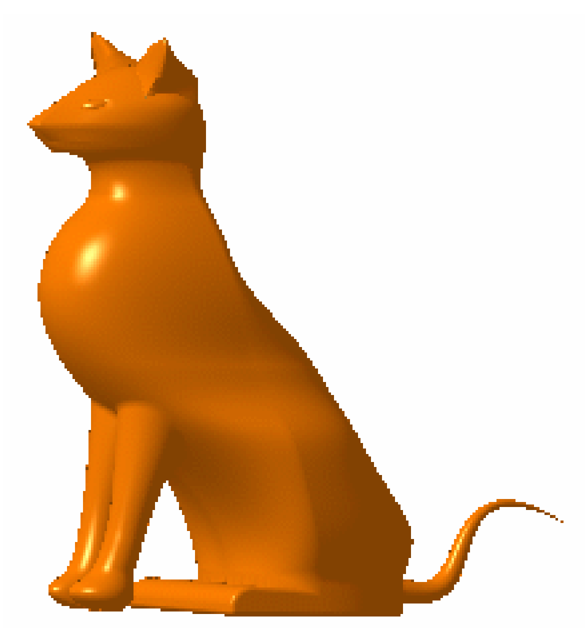


Dots are displayed in the middle of each face of the box.

3. Move the mouse close to any of the dots: two arrows display letting to stretch the model in any direction.



4. Click OK when you are satisfied with the modeling.



Analyzing

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.

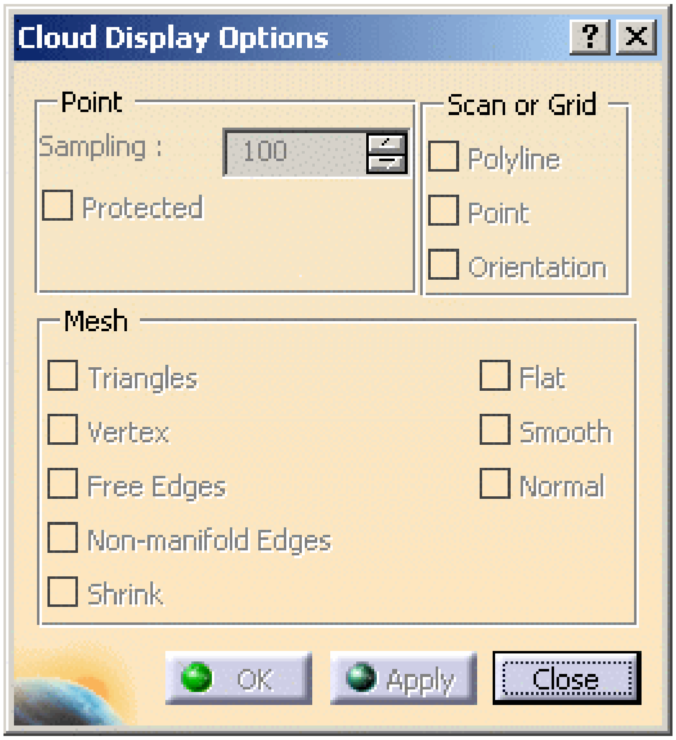
From the **Edit/Properties** menu, you can:

- 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.

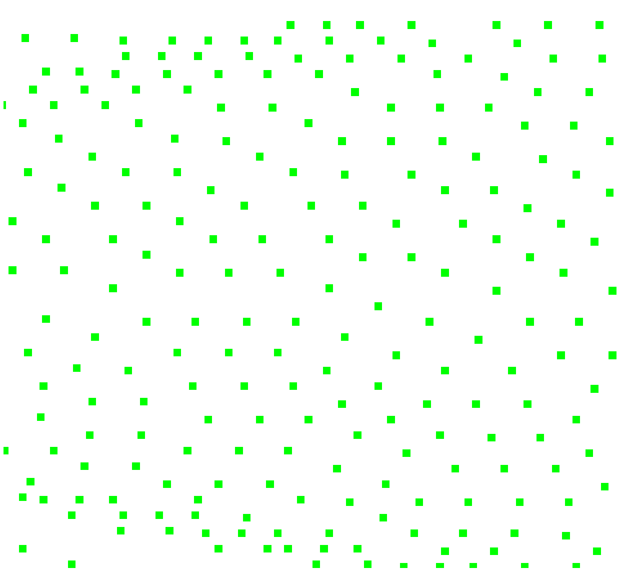


2. Select the cloud to modify. Display options are proposed according to the type of the cloud selected:

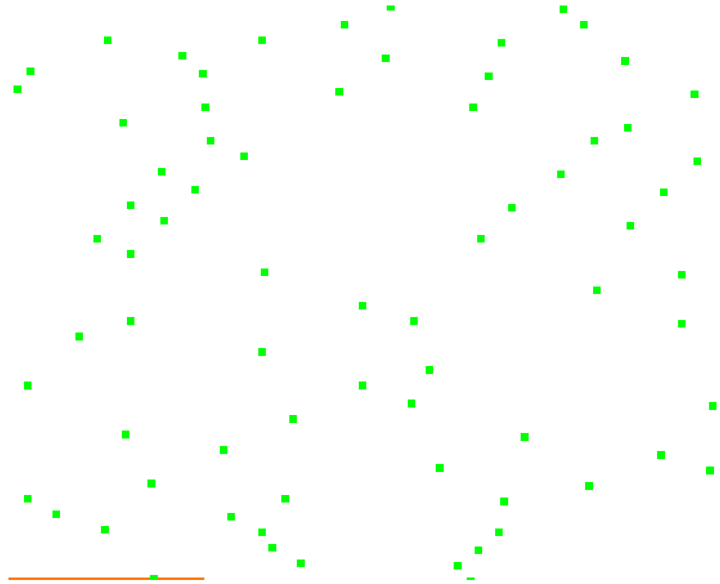
Following options are not yet available:

- 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.



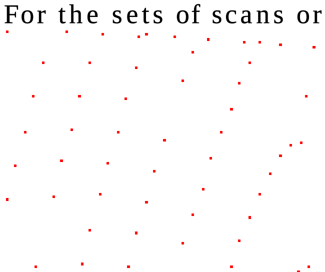
Sampling=100

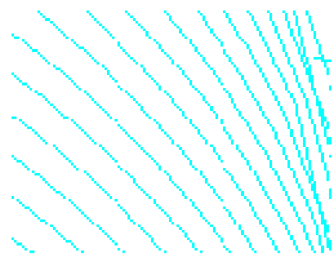


Sampling=25

The **Symbol** options are not available in that box, but in the Graphic Properties menu.

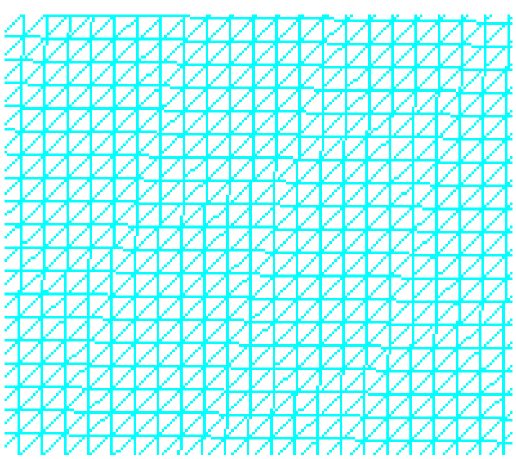
For the sets of scans or grids, you can display them as line of points or points or both:



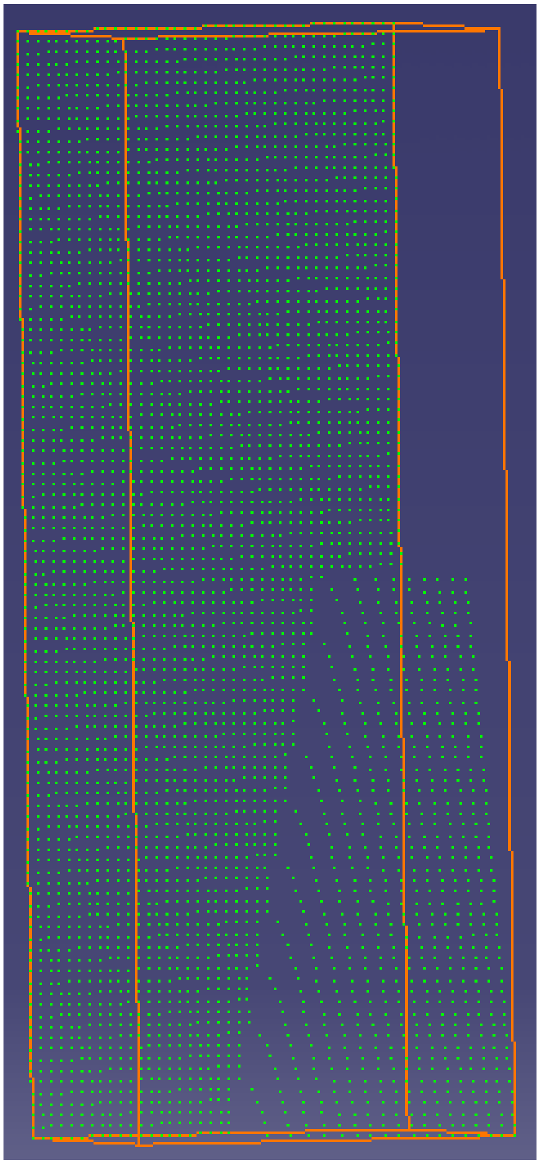
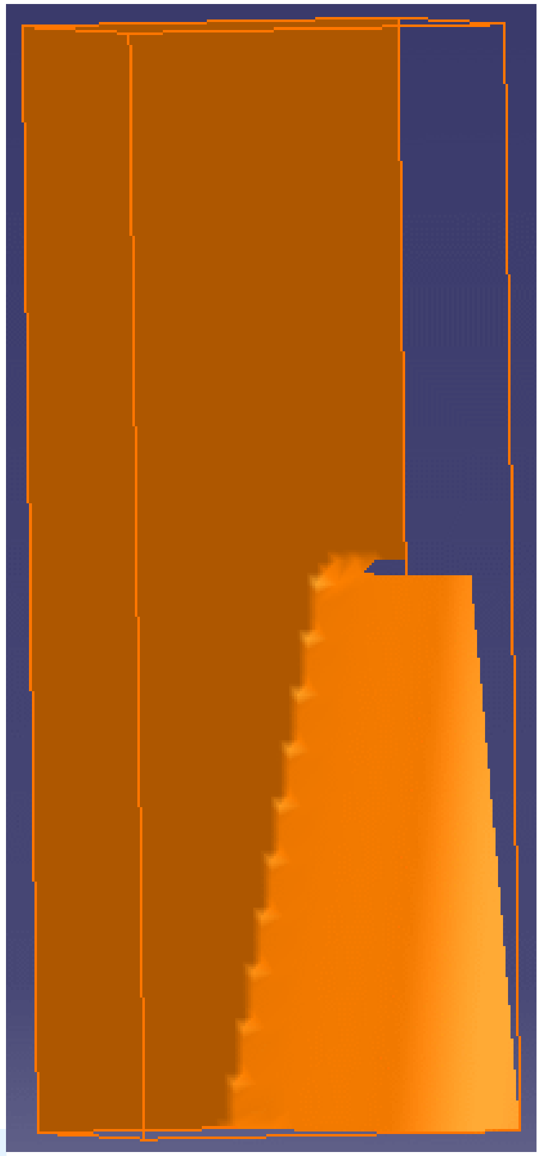


For the mesh, you can:

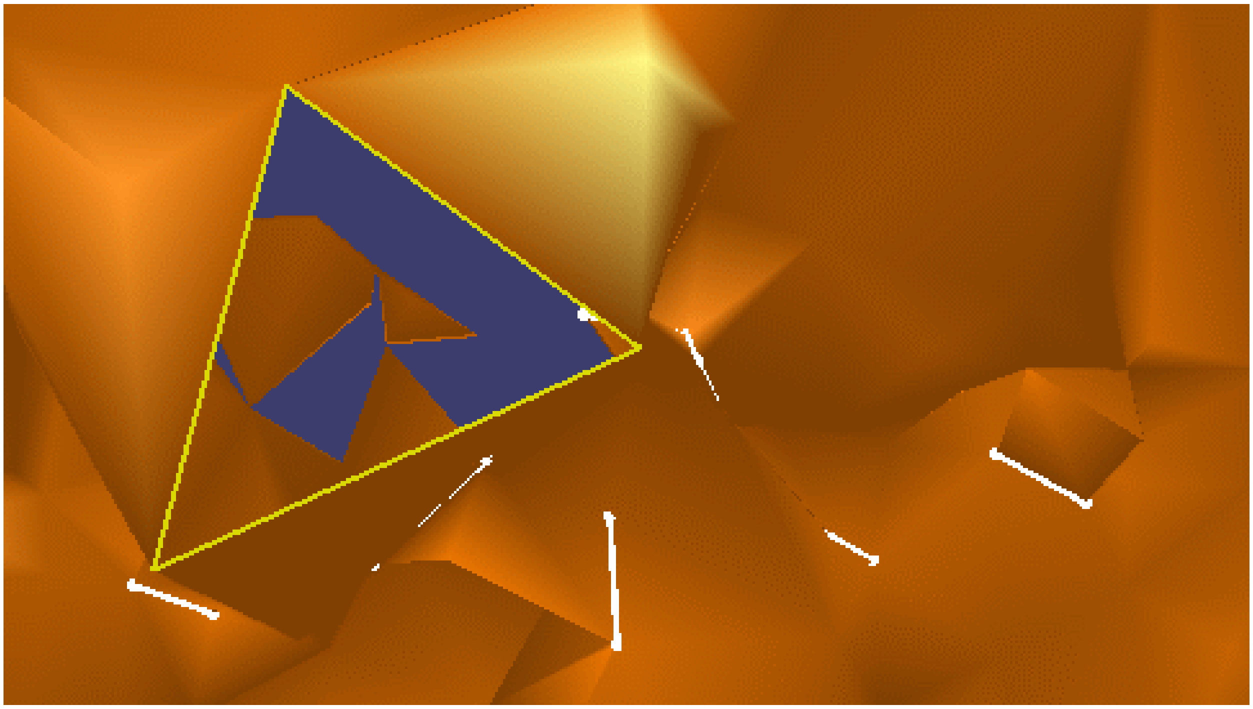
- display the triangles,



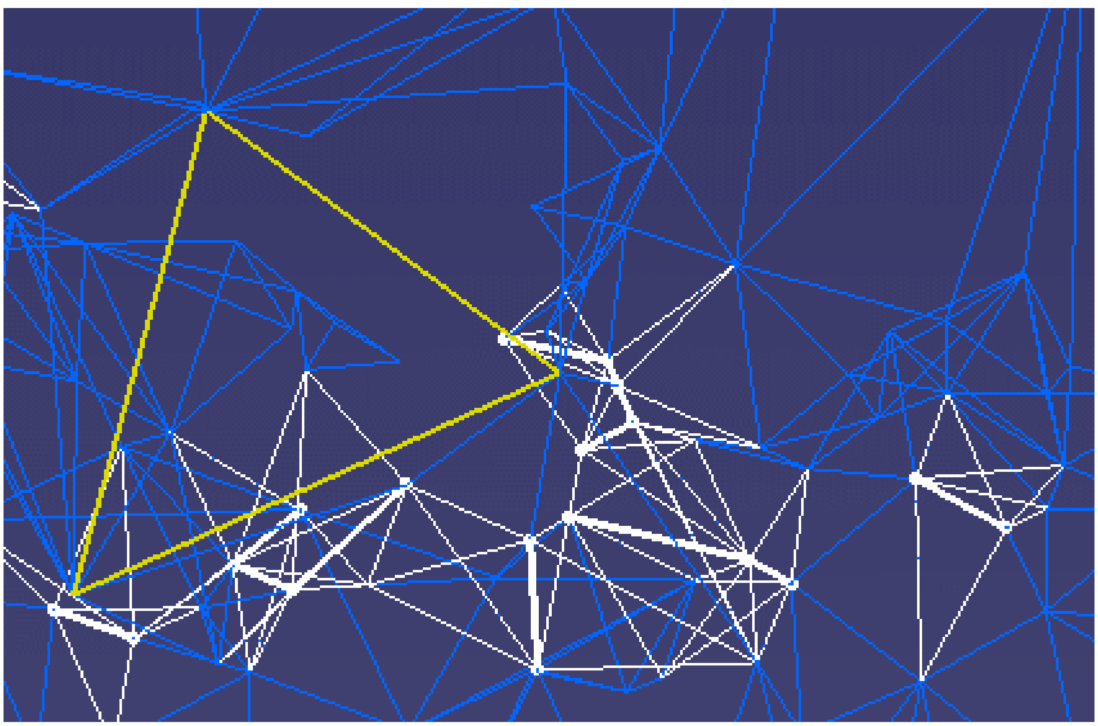
- visualize only the vertices for a lighter display (do not forget to de-activate the Smooth, Flat or Triangles options)

 instead of 

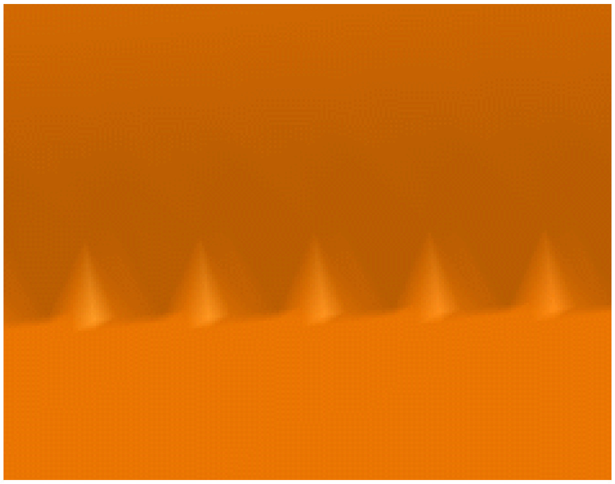
- the free edges in yellow,
- the non-manifold facets and their vertices in bold white lines.



- If you choose the display of triangles, the triangles accepting a non-manifold edge have their edges displayed as regular white lines.

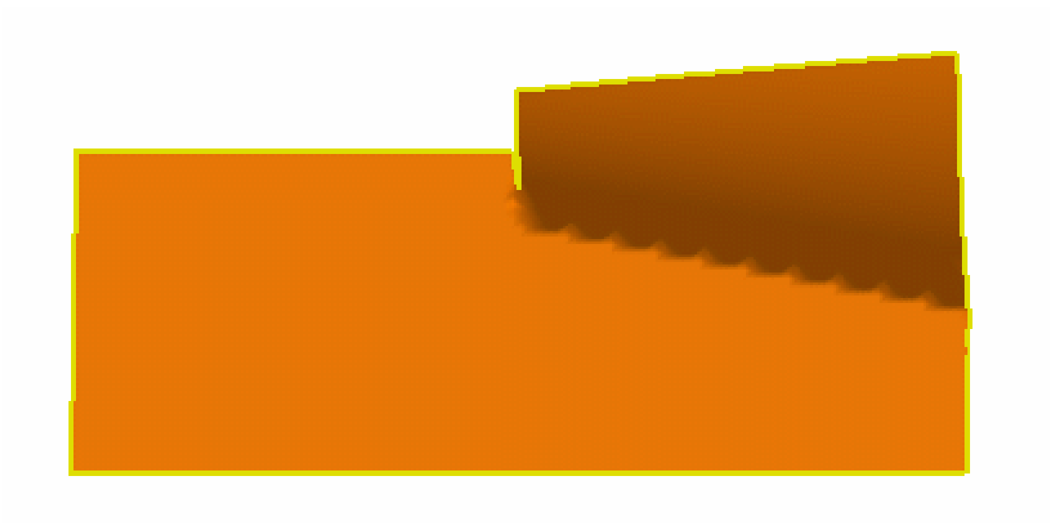


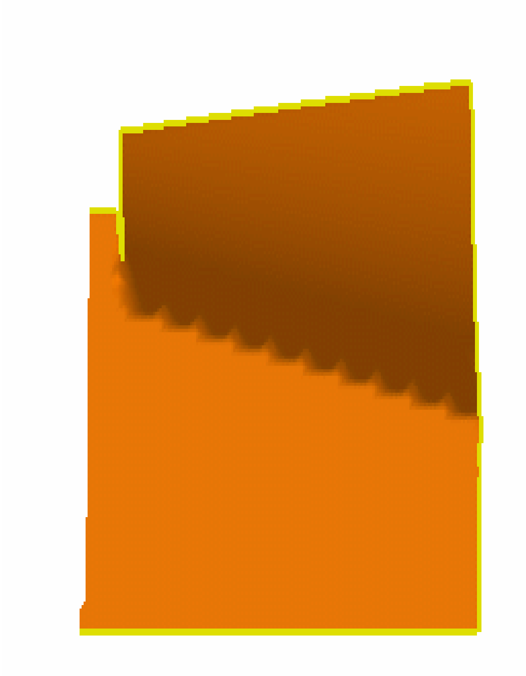
- display the mesh as a smooth or a flat mesh.

 or 

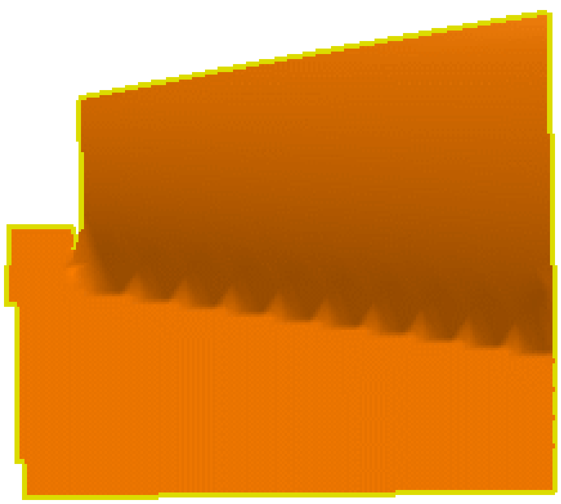
The free edges displayed are those of the complete cloud of points:

- 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.



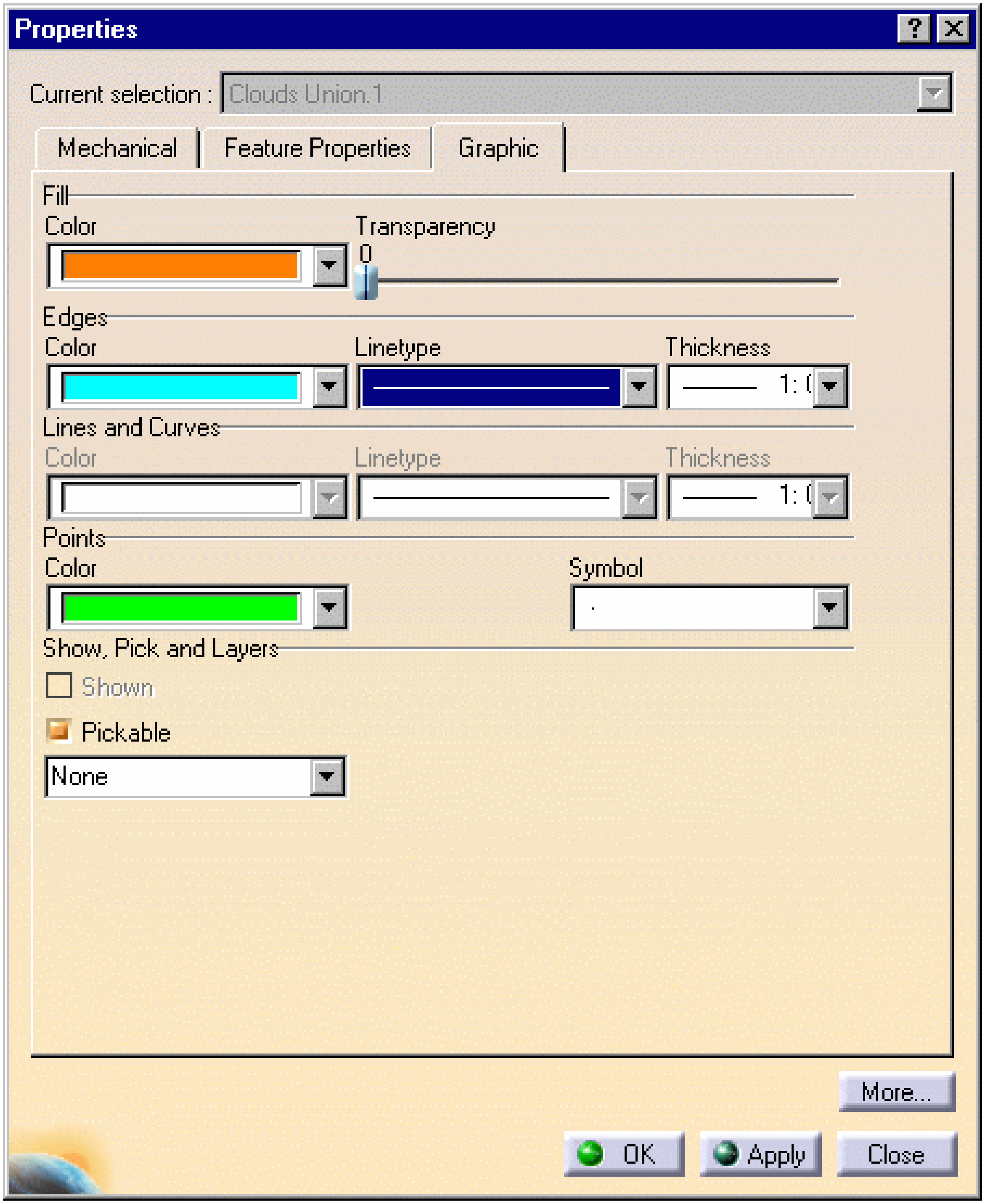
- If you move a cloud of points or a mesh, its graphic display options (not the graphic properties) are lost.
- The display options are not saved in the CATPart while the graphic properties are.

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.

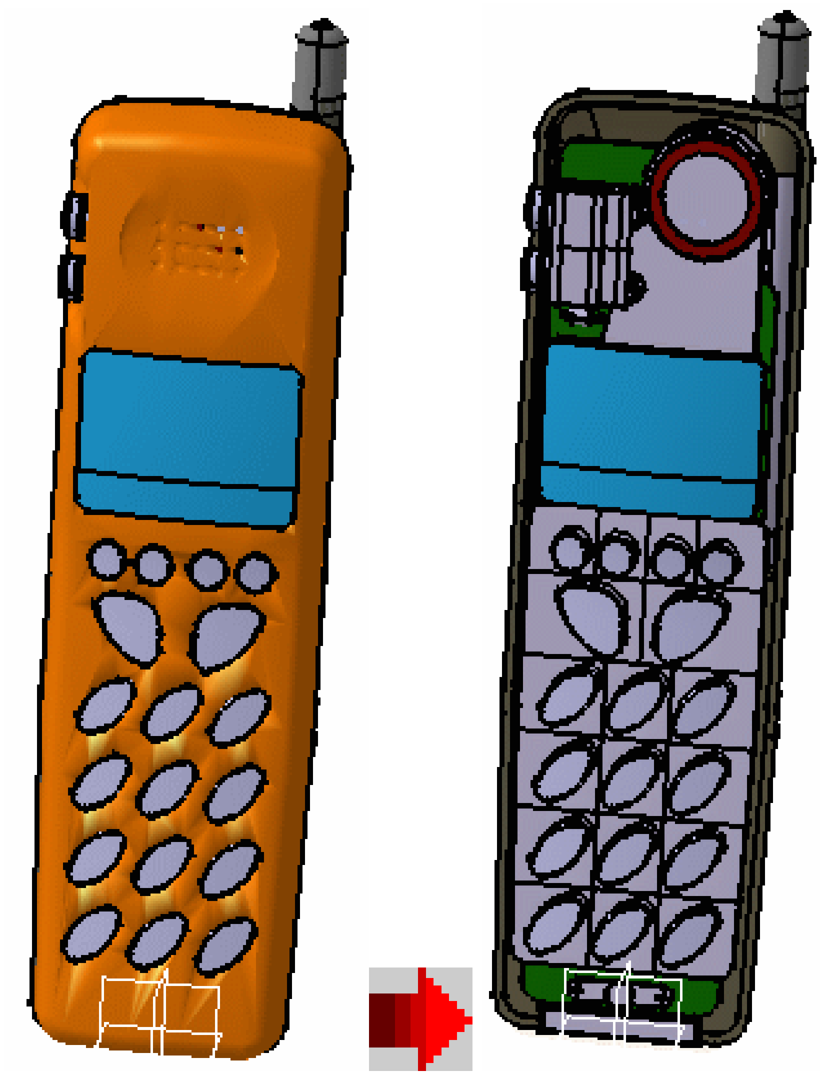
You can access this menu through **Edit / Properties**, or through the pop-up menu of the element, or display the **Graphic Properties** toolbar (**View / Toolbars / Graphic Properties**).



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:

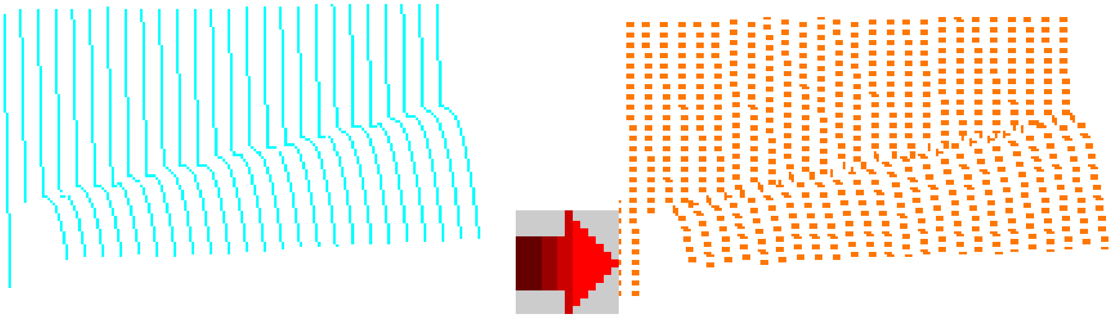


Please note that :

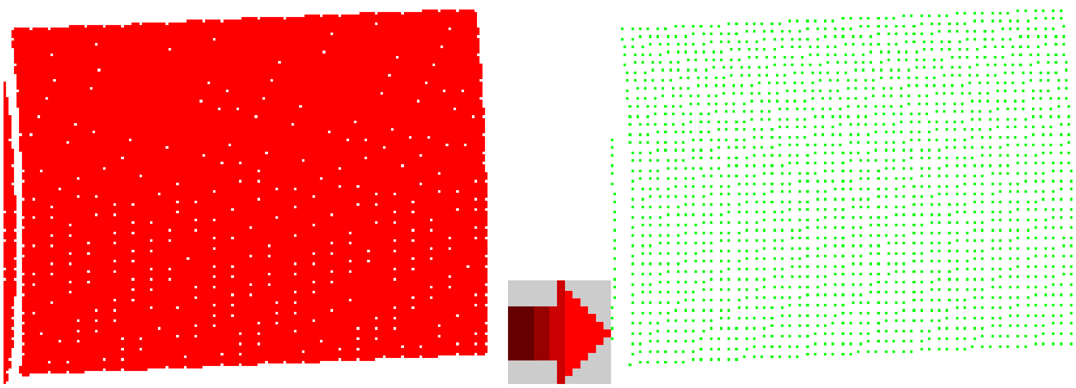
- the color of mesh free edges is yellow, and is not editable,
- the color of non-manifold edges is white, and is not editable,
- the default color of scans has changed to cyan.

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.

Analyzing Using Highlights

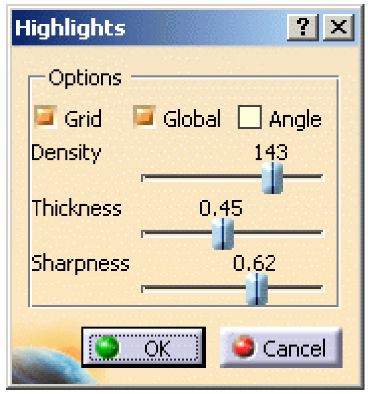
This task shows how to perform an analysis of the surfaces quality of shaded highlights.
It analyses the angle between surface normals and a predefined direction. Dependent on the angle, it sets the color at this surface position. All positions with the same angle between surface normal and predefined direction get the same shading color. If the angle is 90 degrees, the resulting stripe is the silhouette line related to the predefined direction.

Open the Highlights1.CATPart document.

Make sure the Shading and Material visualization options are active by selecting the **Customize View** icon in the View mode toolbar.

1. Click the **Highlight** icon .

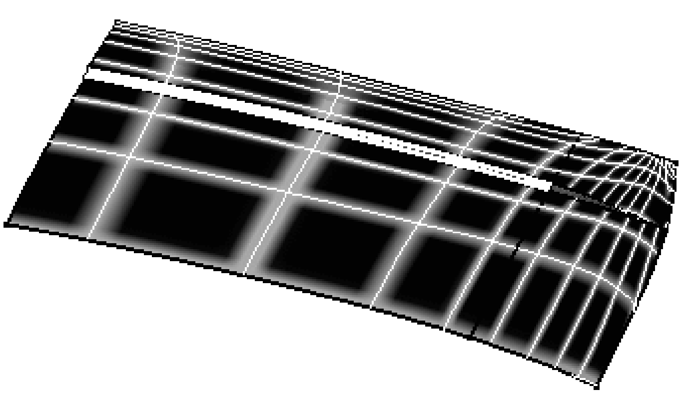
The Highlights dialog box displays.



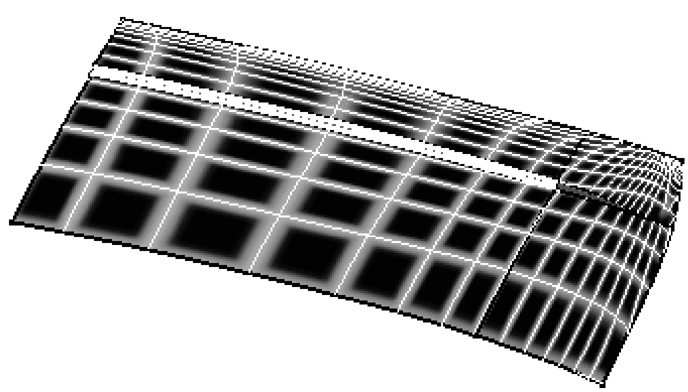
You do not need to select the part in the specification tree as the analysis works globally on the part.

2. Define the display options:

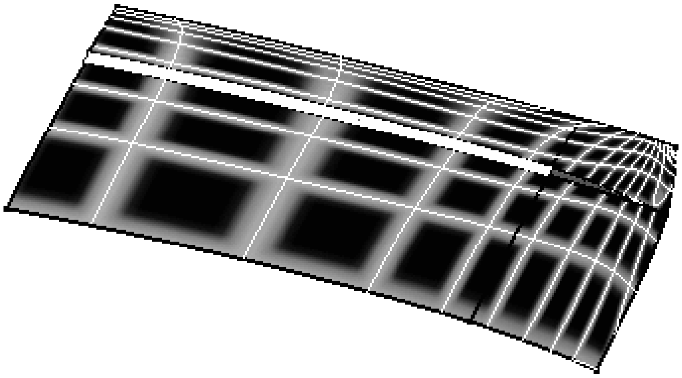
- **Grid**: two predefined directions define a grid of stripes (not necessarily perpendicular) on the surface. The plane tool u and v direction, are used as predefined directions in grid mode. If you uncheck the Grid options, it switches to the Stripes mode: the plane tool normal is the predefined direction.
- **Global**: the whole part is highlighted.
- **Angle**: highlights are distributed per angle.
- **Density**: defines the number of stripes or grid lines.
- **Thickness**: defines the thickness of stripes or grid lines.
- **Sharpness**: defines the sharpness of stripes or grid lines (soft or hard color transition).



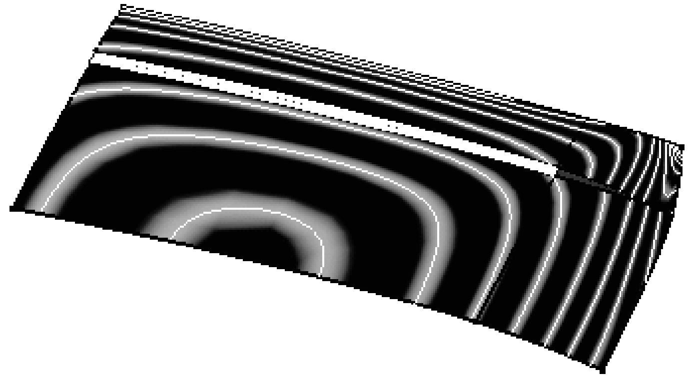
Here is an example the following values
Grid and Global
Density=50
Thickness=0.30
Sharpness=0.80



Here is an example the following values
Grid and Global
Density=80
Thickness=0.70
Sharpness=0.30

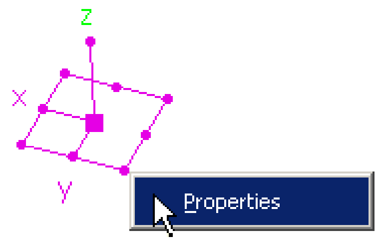


Here is an example the following values
Grid, Global, and Angle
Density=80
Thickness=0.70
Sharpness=0.30

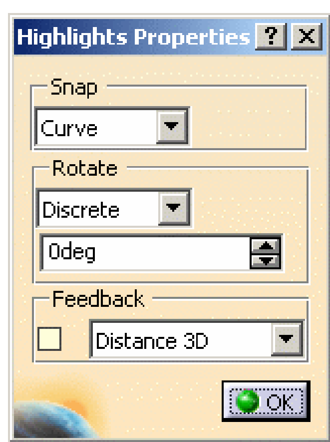


Here is an example the following values
Global and Angle
Density=80
Thickness=0.70
Sharpness=0.30

3. Right-click the plane tool to display the Highlights Properties.



- Define the snap mode to snap tool position and optionally orientation on geometry
  - Off: no snap mode
  - Vertex: the plane tool snaps on a vertex. Dependent on the vertex (surface corner, 3D curve endpoint), the direction is adapted to the surface normal or curve tangent
  - Curve: the plane tool snaps on a curve or surface edge. As a result in stripes mode the silhouette line of the plane tool normal crosses the edge at the plane tool origin
  - Surface: the plane tool snaps on a surface and adapts the surface normal. As a result in grid mode the silhouette line of the plane tool u direction and the silhouette line of the plane tool v direction are crossing at the plane tool origin



- Define the rotation mode of the 3D tool:
  - **Dynamic**: normal dynamic rotation mode
  - **Discrete**: the rotation of the tool snaps to certain relative grid angles.
  - **Grid**: the rotation of the tool snaps to certain absolute grid angles.
  - **Static**: the rotation is activated by a mouse click on the corresponding rotation handle of the tool. The grid value defines the specific rotation angle
- Define the rotation value by entering a value or using the spinners

- Define the numerical feedback of the 3D tool:
  - **Distance 3D**: direct distance between the start position of the translation and the current position
  - **Distance XYZ**: distance shown in x, y and z components of the current coordinate system
  - **Coordinates**: absolute position of the tool in the model coordinate system

4. Click OK to create the analysis.

For further information, please refer to the *CATIA Automotive Class A* documentation.

Interoperability with Wireframe

Creating Points

This task shows the various methods for creating points:

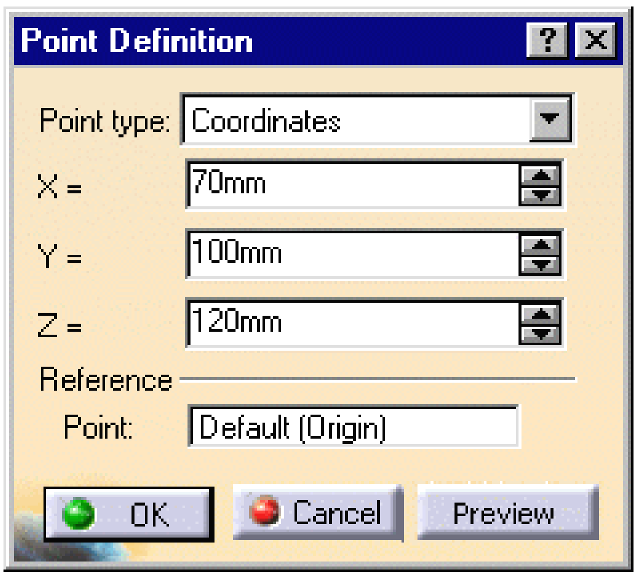
- by coordinates
- on a curve
- on a plane
- on a surface
- at a circle center
- tangent point on a curve
- between

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.



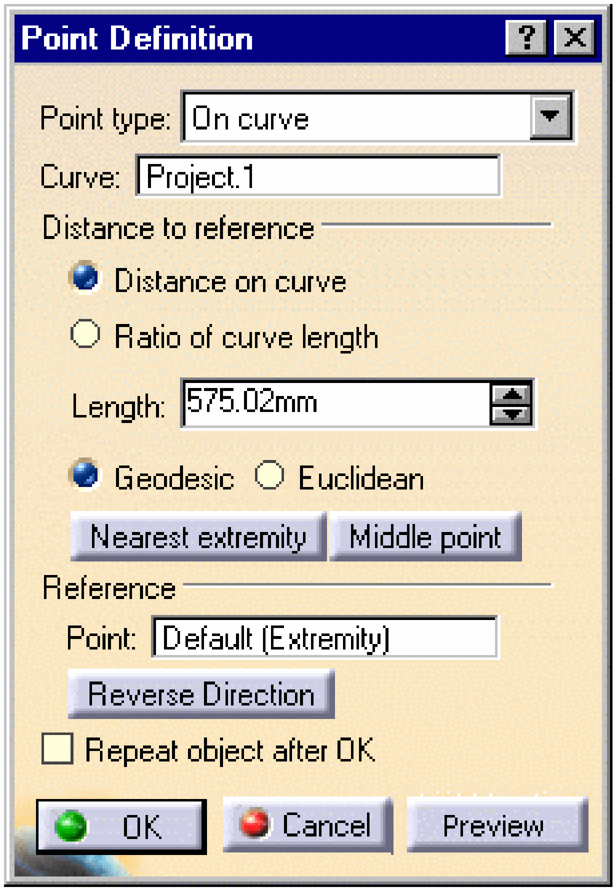
Coordinates

- 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.

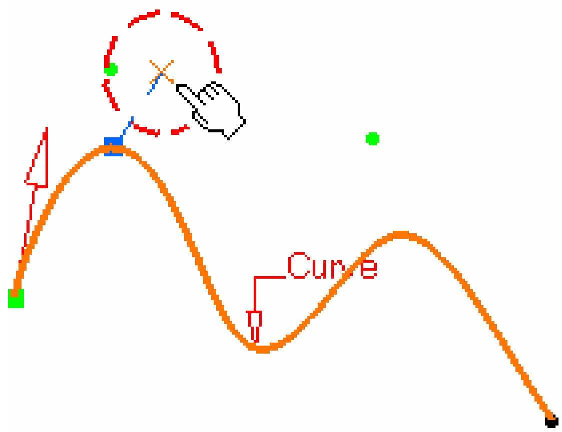


On curve

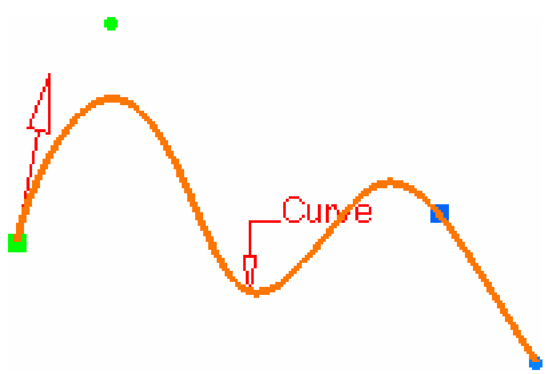
- Select a curve
- Optionally, select a reference point.

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.

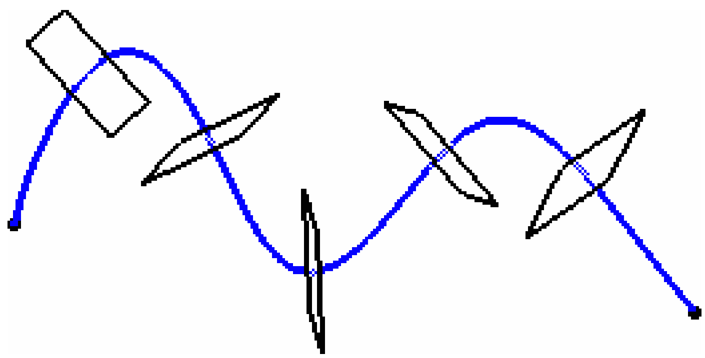
You can also:

- 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.

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

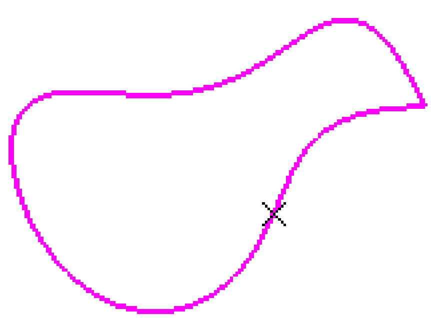
- use the **Reverse Direction** button to display:
  - the point on the other side of the reference point (if a point was selected originally)
  - the point from the other extremity (if no point was selected originally).
- click the **Repeat object after OK** if you wish to create equidistant points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

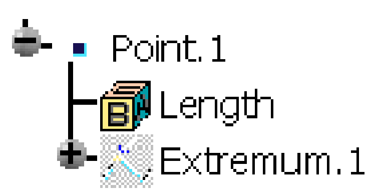
You will also be able to create planes normal to the curve at these points, by checking the **Create normal planes also** button, and to create all instances in a new geometrical set by checking the **Create in a new geometrical set** button.
If the button is not checked the instances are created in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



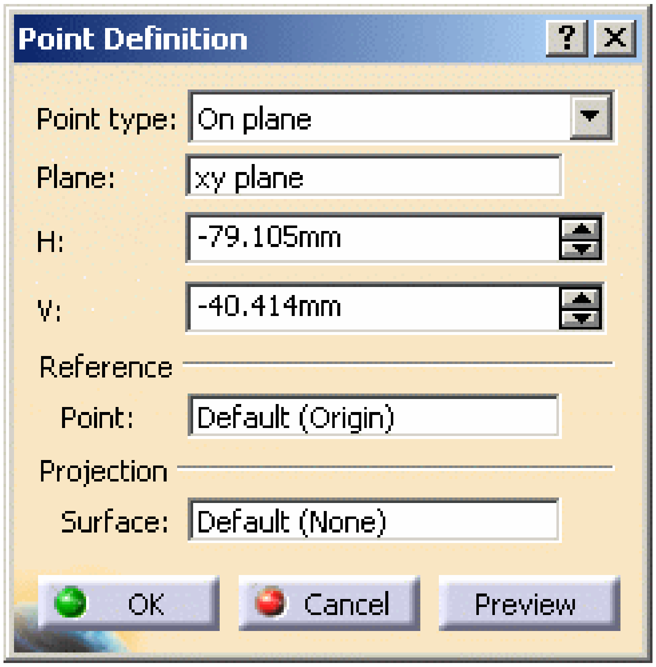
- If the curve is infinite and no reference point is explicitly given, by default, the reference point is the projection of the model's origin
- If the curve is a closed curve, either the system detects a vertex on the curve that can be used as a reference point, or it creates an extremum point, and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

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





On plane



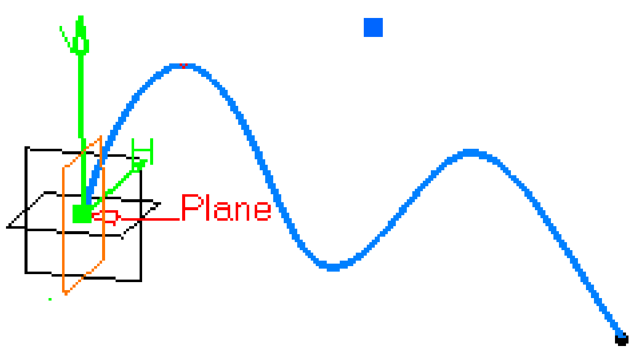
- Select a plane.
- Optionally, select a point to define a reference for computing coordinates in the plane.

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

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

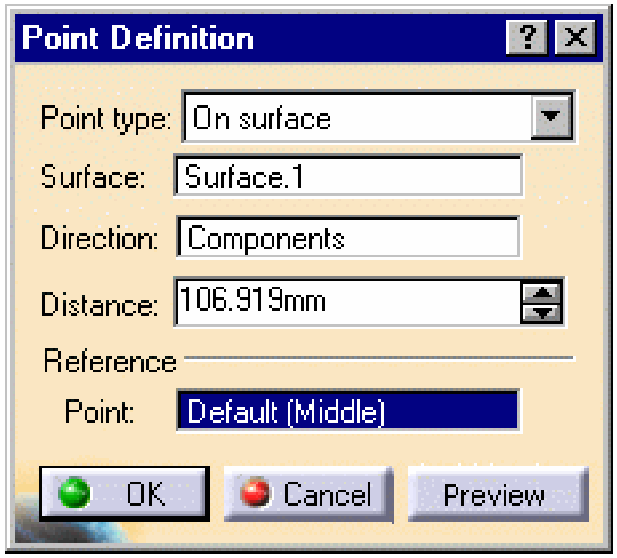
  If no surface is selected, the behavior is the same.

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



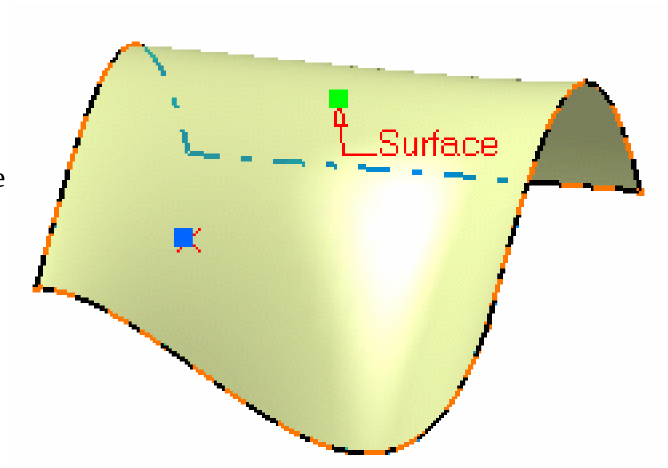
  Would the plane move, during an update for example, the reference direction would then be projected on the plane.

- Click in the plane to display a point.



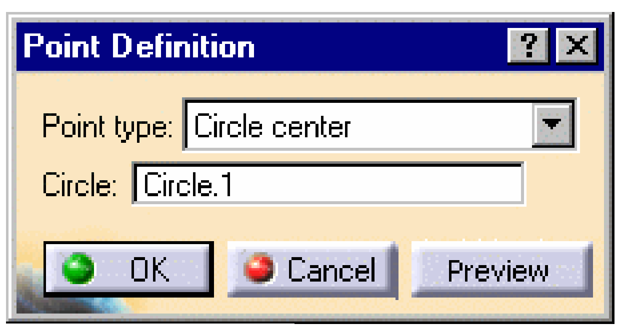
On surface

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

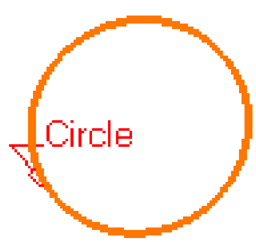


Circle center

- Select a circle, circular arc, or ellipse.



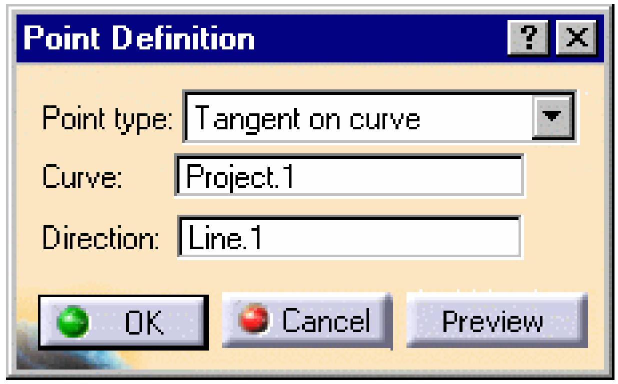
A point is displayed at the center of the selected element.



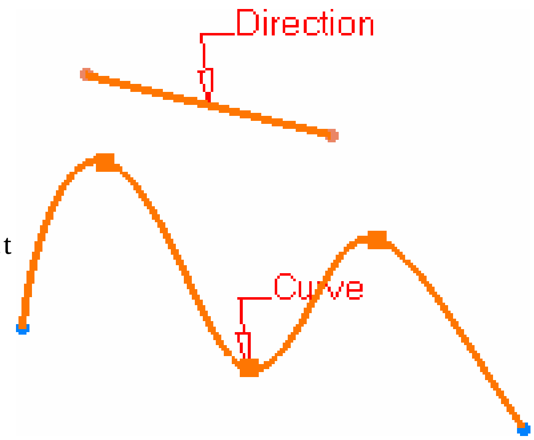
Tangent on curve

- Select a planar curve and a direction line.

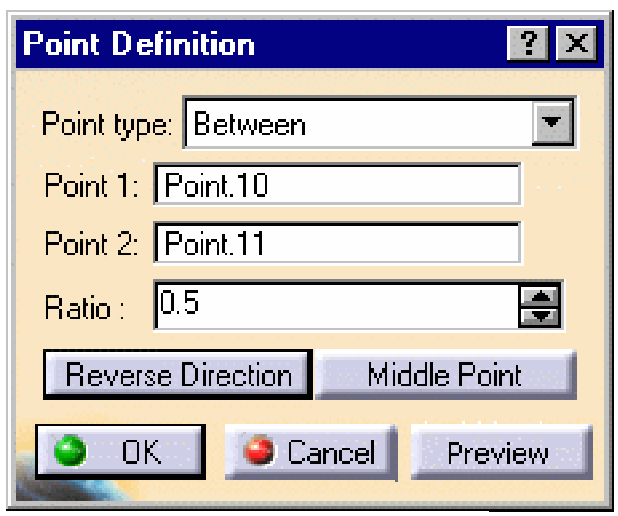
A point is displayed at each tangent.



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



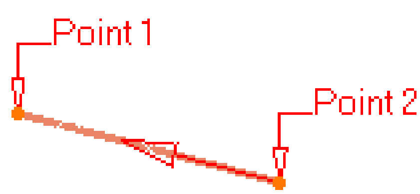
- Click **YES**: you can then select a reference element, to which only the closest point is created.
- Click **NO**: all the points are created.



Between

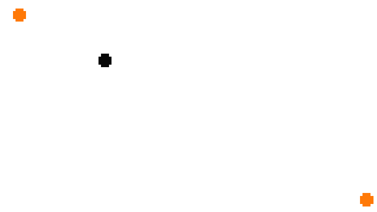
- Select any two points.

- 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).



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

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



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

3. Click OK to create the point.

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

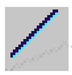
Creating Lines

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.

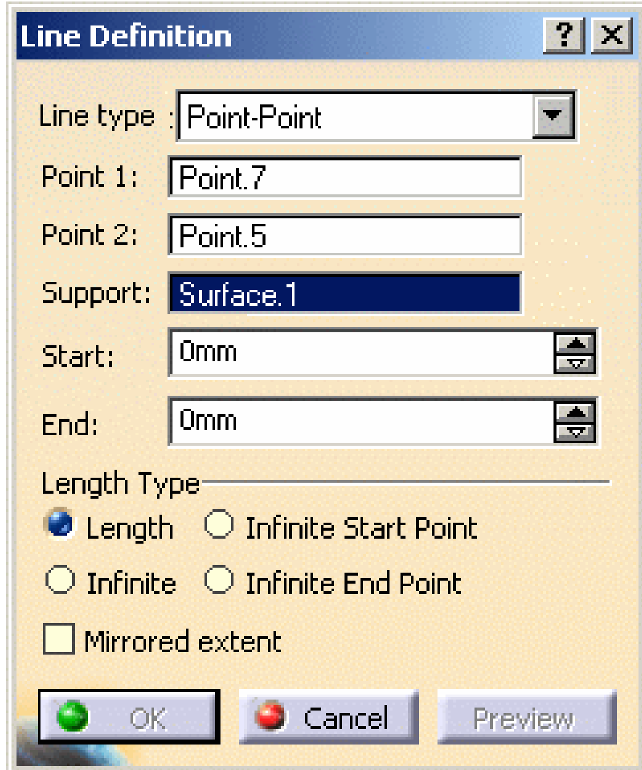
A line type will be proposed automatically in some cases depending on your first element selection.

Point - Point

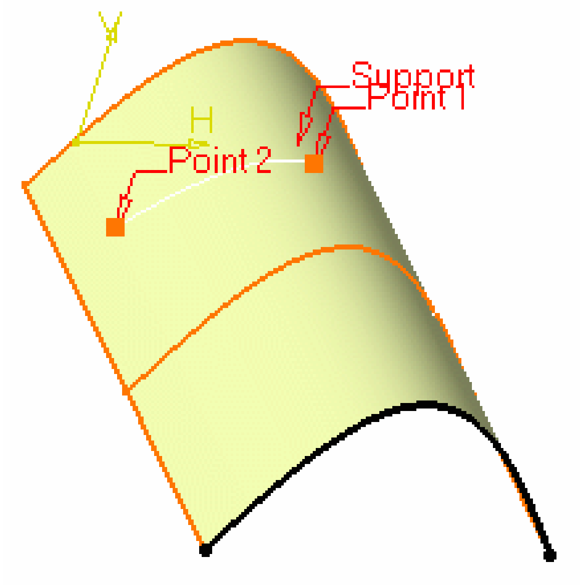
This command is only available with the Generative Shape Design 2 product.

- Select two points.

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

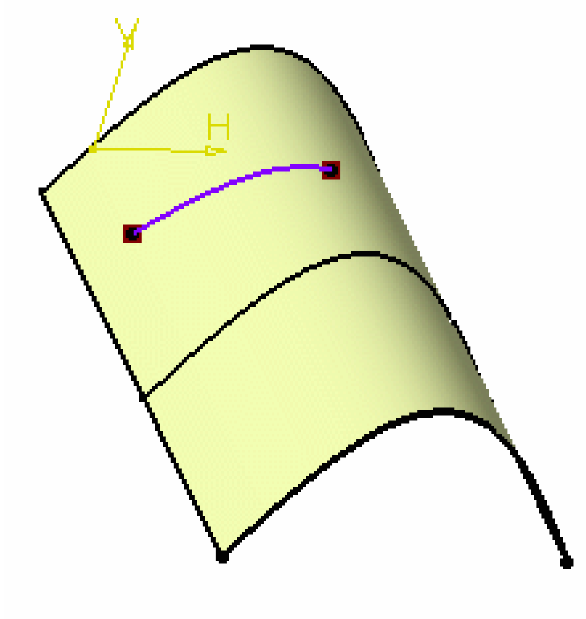


- If needed, select a support surface.
  In this case a geodesic line is created, i.e. going from one point to the other according to the shortest distance along the surface geometry (blue line in the illustration below).
  If no surface is selected, the line is created between the two points based on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

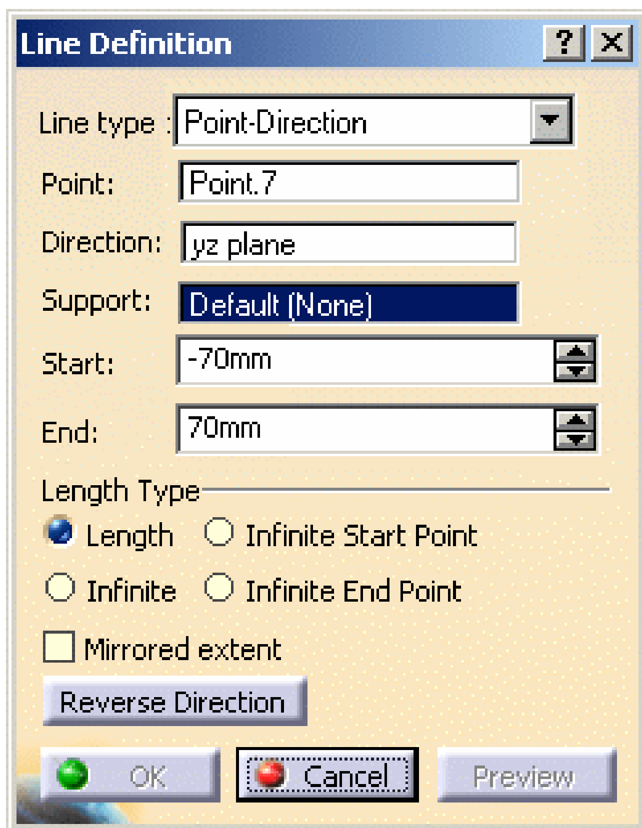
The geodesic line is not available with the Wireframe and Surface workbench.



- 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.

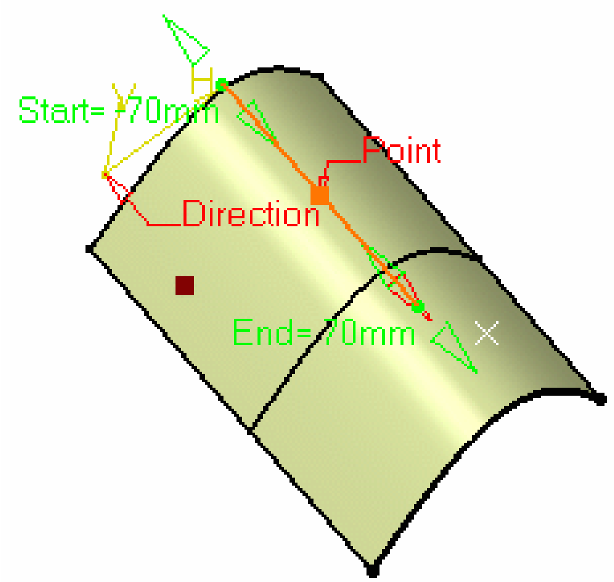
The projections of the 3D point(s) must already exist on the selected support.

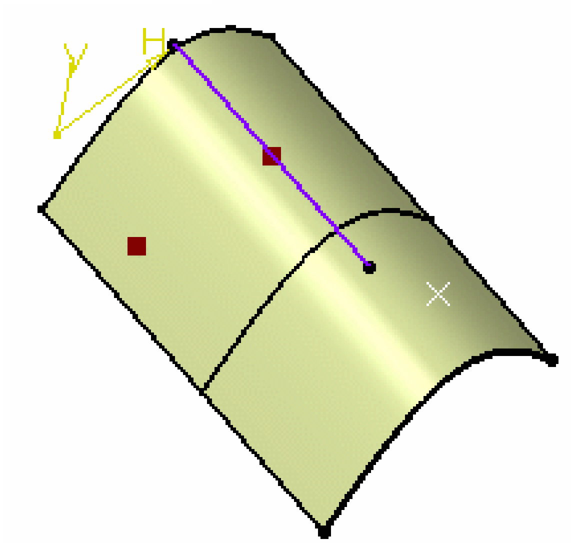
Point - Direction



- 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.





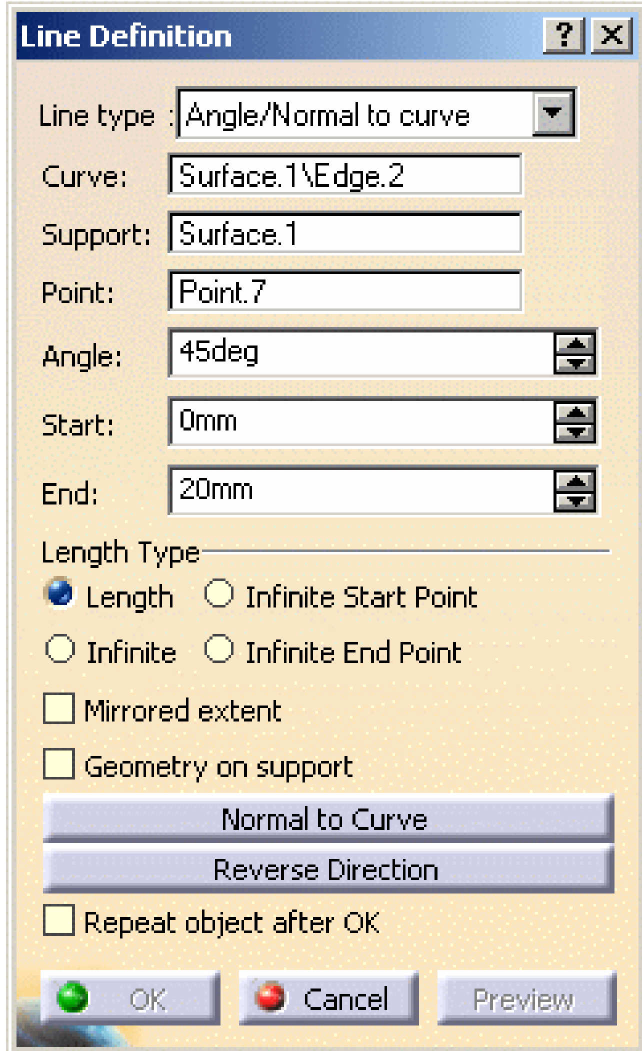
The projections of the 3D point(s) must already exist on the selected support.

Angle or Normal to curve

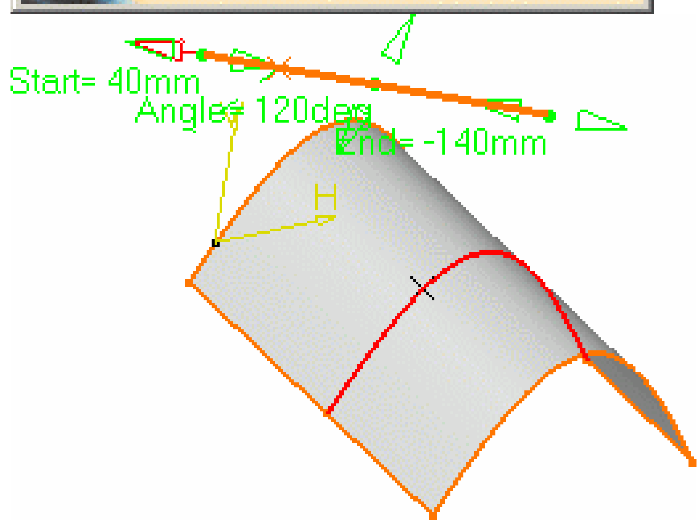
- Select a reference **Curve** and a **Support** surface containing that curve.

- 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.



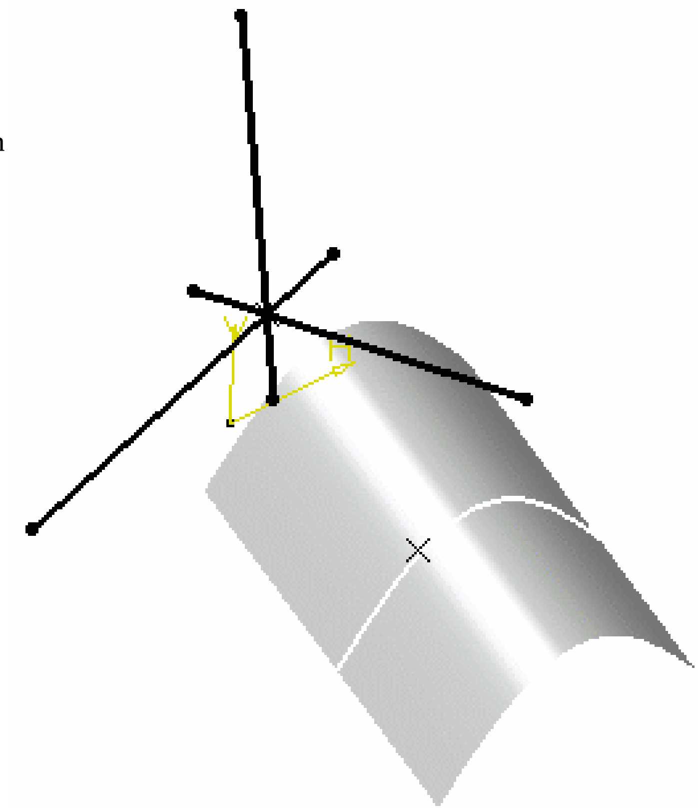
- Select a **Point** on the curve.
- Enter an **Angle** value.



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.

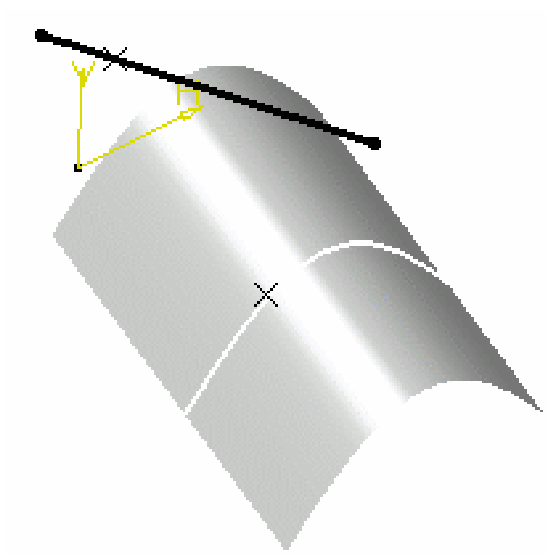
- Click the **Repeat object after OK** if you wish to create more lines with the same definition as the currently created line.
  In this case, the Object Repetition dialog box is displayed, and you key in the number of instances to be created before pressing OK.



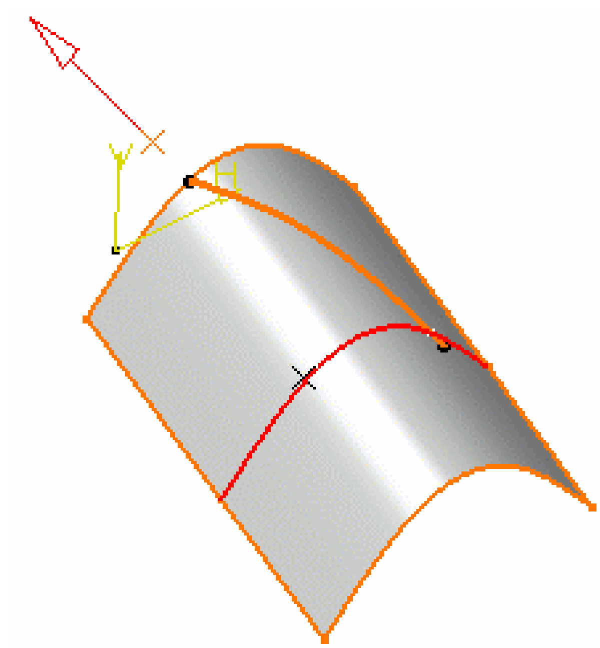
As many lines as indicated in the dialog box are created, each separated from the initial line by a multiple of the **angle** value.

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.



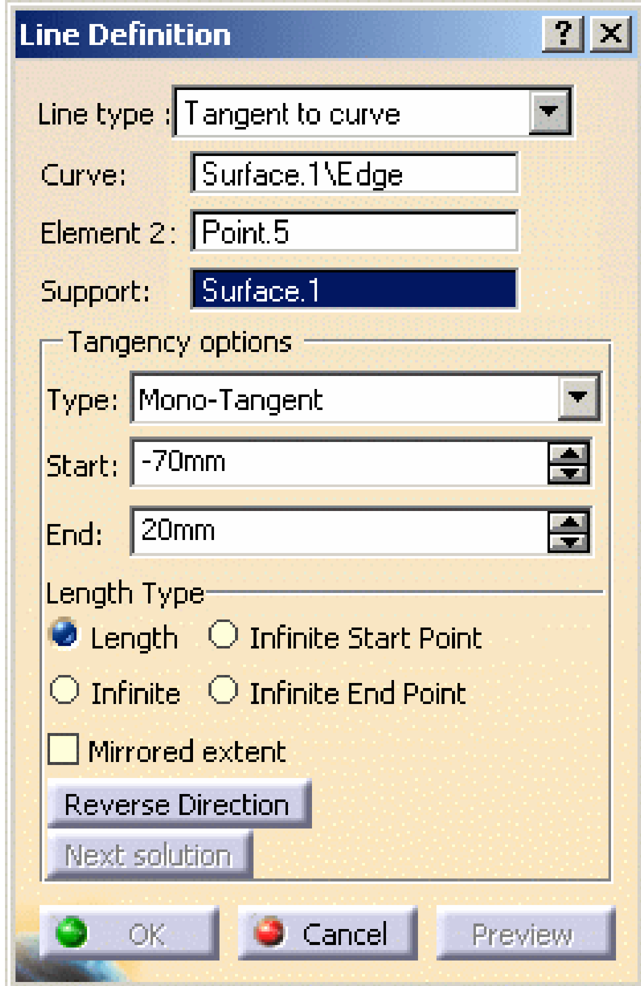
Geometry on support option not checked



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

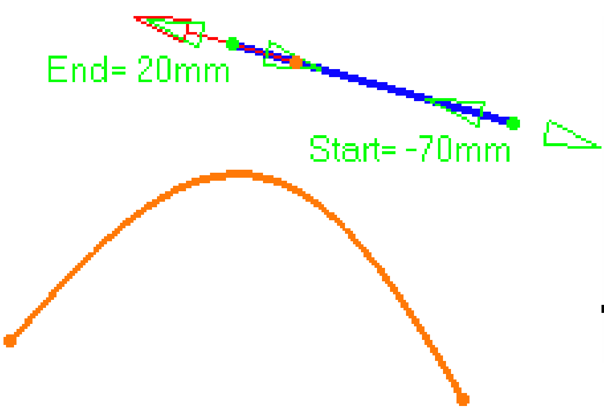


- Select a reference **Curve** and a **point** or another **Curve** to define the tangency.
  - if a point is selected (mono-tangent mode): a vector tangent to the curve is displayed at the selected point.
  - If a second curve is selected (or a point in bi-tangent mode), you need to select a support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

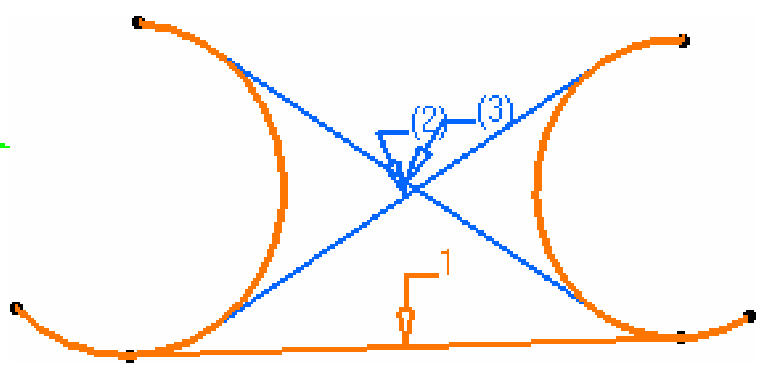
- If the selected curve is a line, then the **Support** is set to Default (Plane).

- 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.

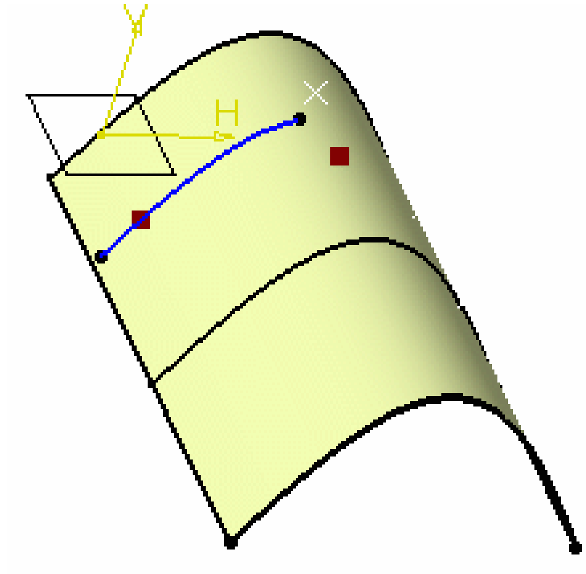


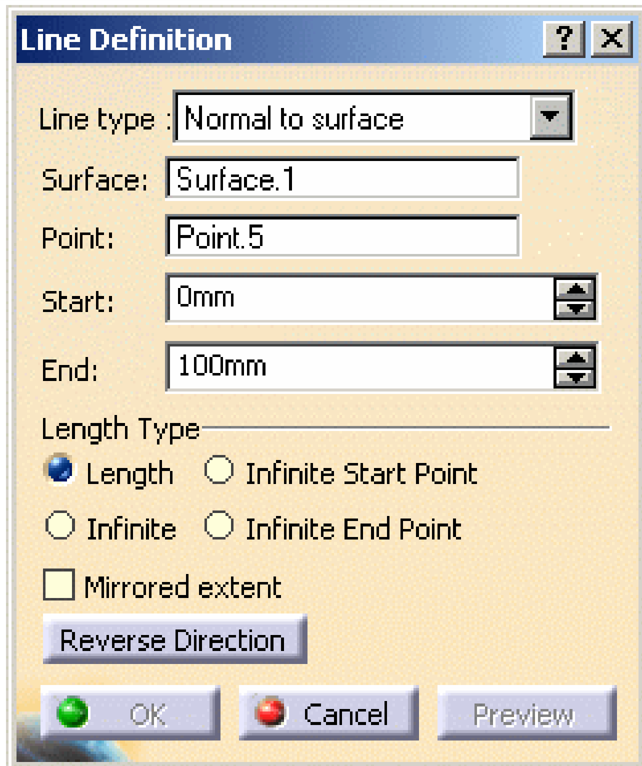
Line tangent to curve at a given point



Line tangent to two curves

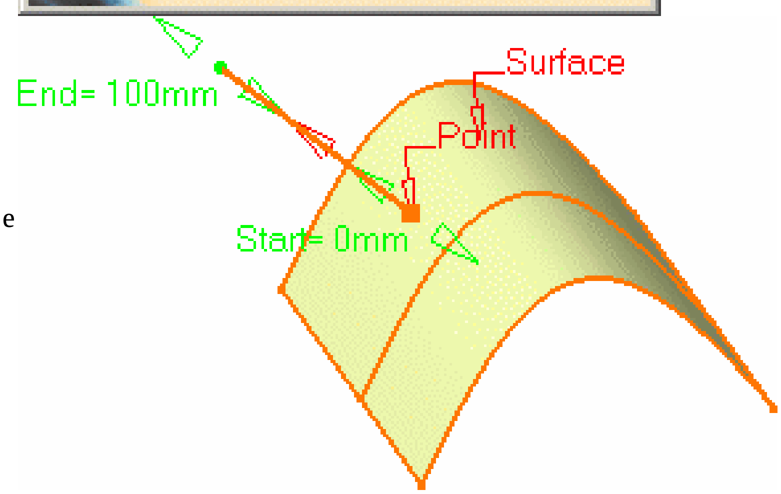
- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.



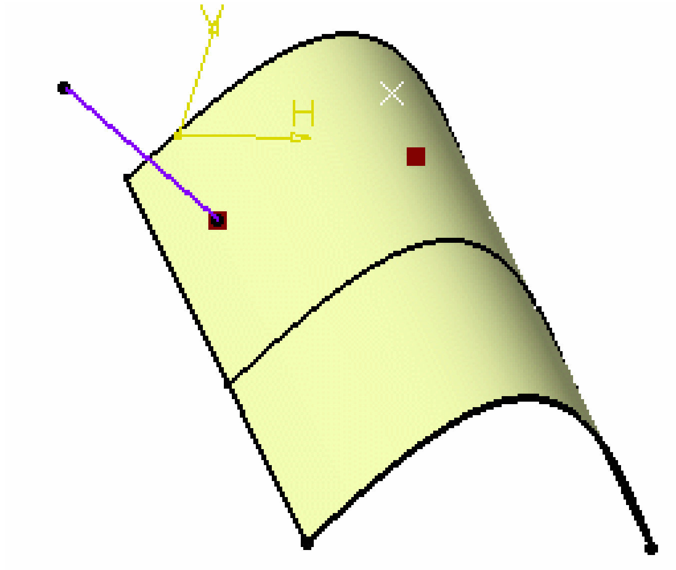


Normal to surface

- 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.

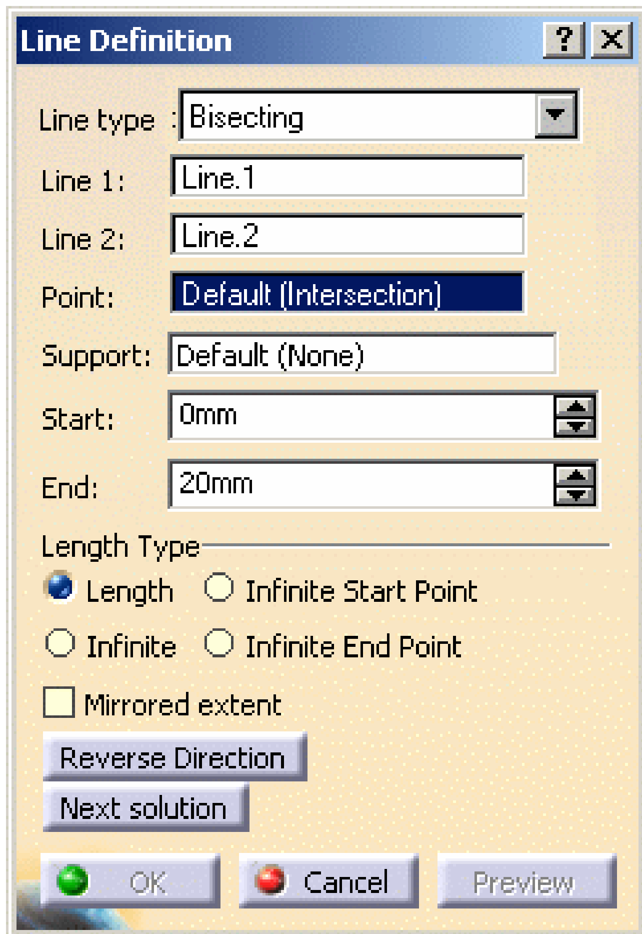


- Specify **Start** and **End** points to define the new line.
  The corresponding line is displayed.

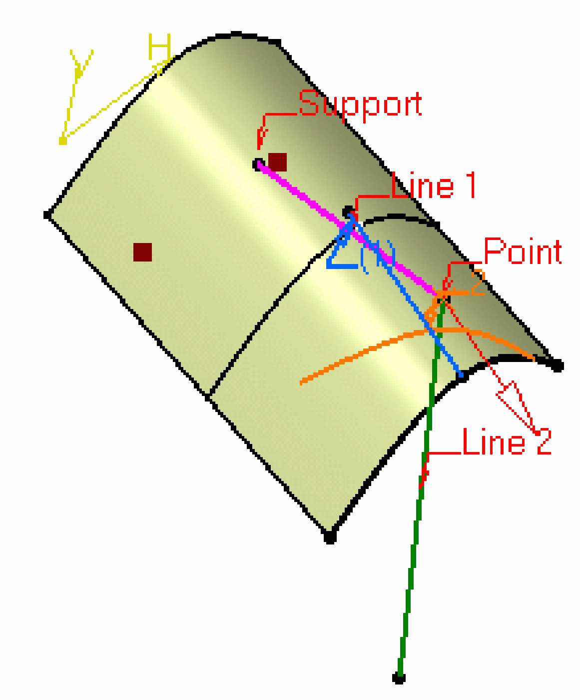


Bisecting

- 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).

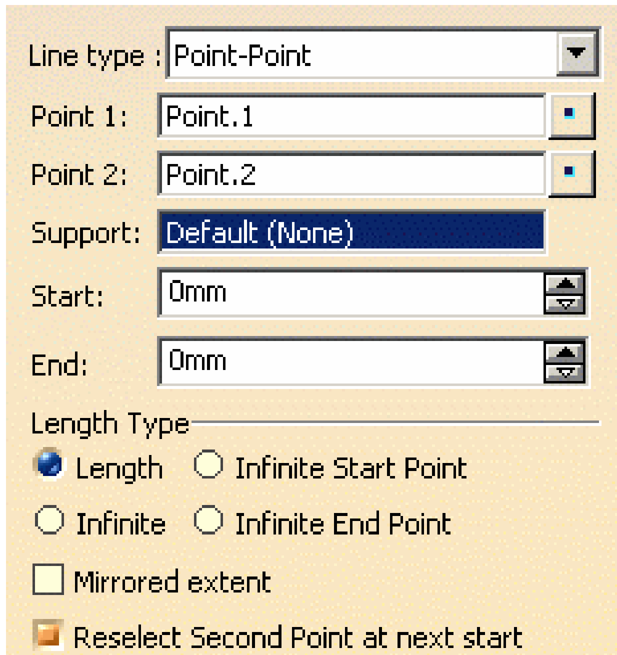
Automatic Reselection

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

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

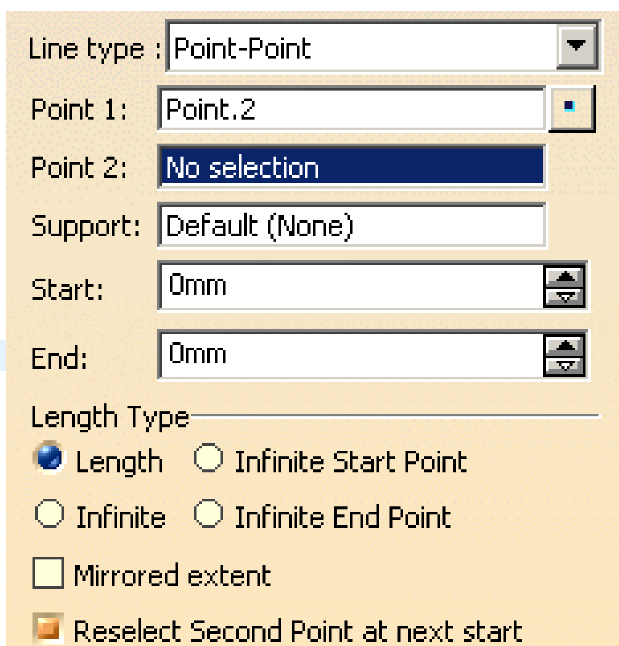
   The Line dialog box is displayed.

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.

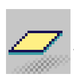
To stop the repeat action, simply uncheck the option or click Cancel in the Line dialog box.

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.

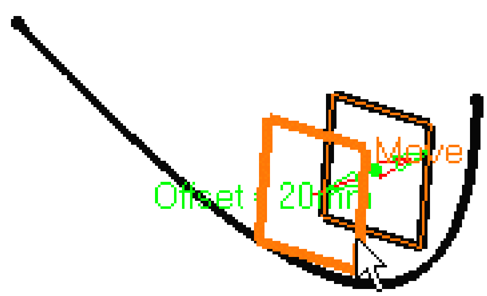
2. Use the combo to choose the desired **Plane type**.

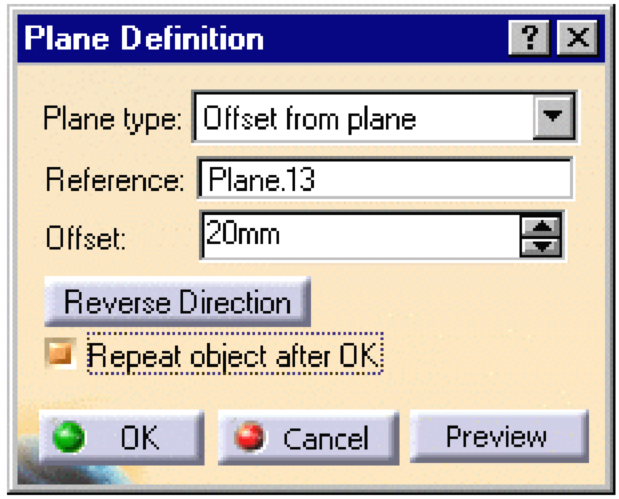
Once you have defined the plane, it is represented by a red square symbol, which you can move using the graphic manipulator.

Offset from plane

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

A plane is displayed offset from the reference plane.

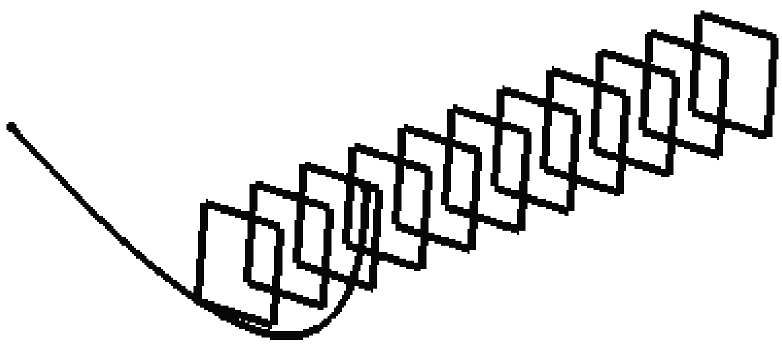




Use the **Reverse Direction** button to reverse the change the offset direction, or simply click on the arrow in the geometry.

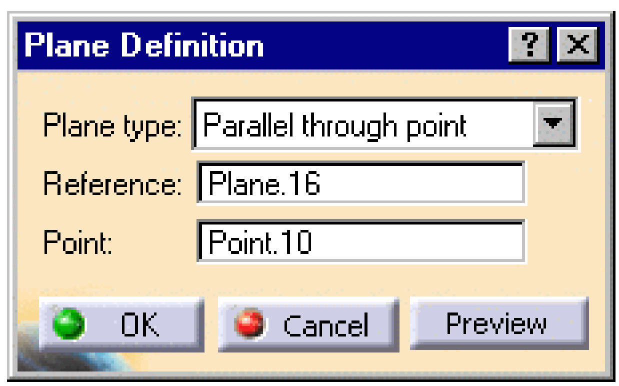
- Click the **Repeat object after OK** if you wish to create more offset planes .
  In this case, the **Object Repetition** dialog box is displayed, and you key in the number of instances to be created before pressing OK.

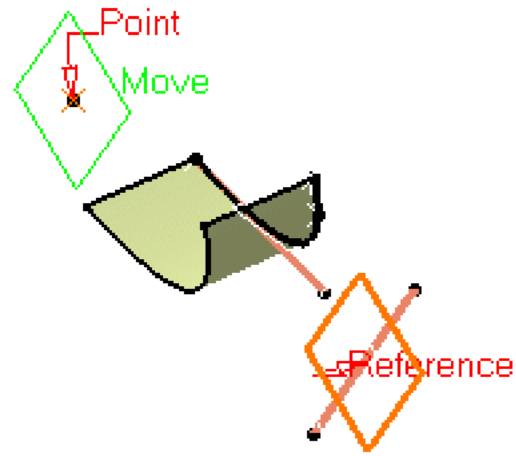
As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Offset** value.



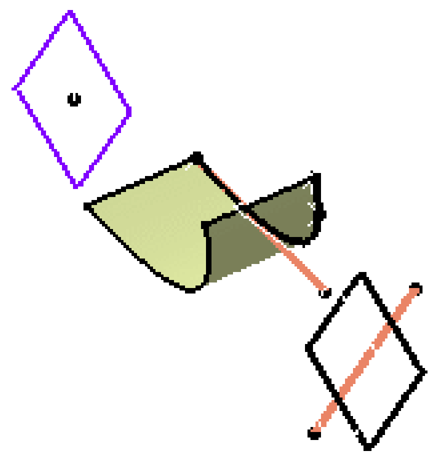
Parallel through point

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

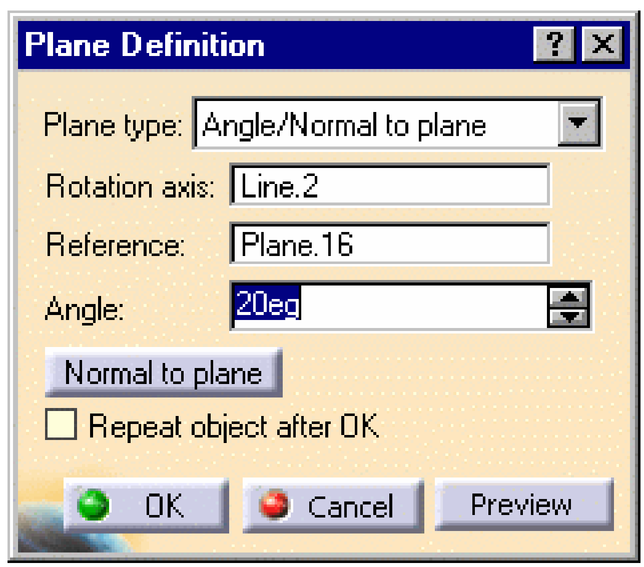




A plane is displayed parallel to the reference plane and passing through the selected point.

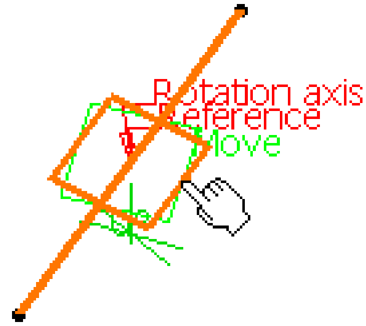


Angle or normal to plane



- 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.

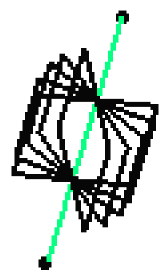
A plane is displayed passing through the rotation axis. It is oriented at the specified angle to the reference plane.



- 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.

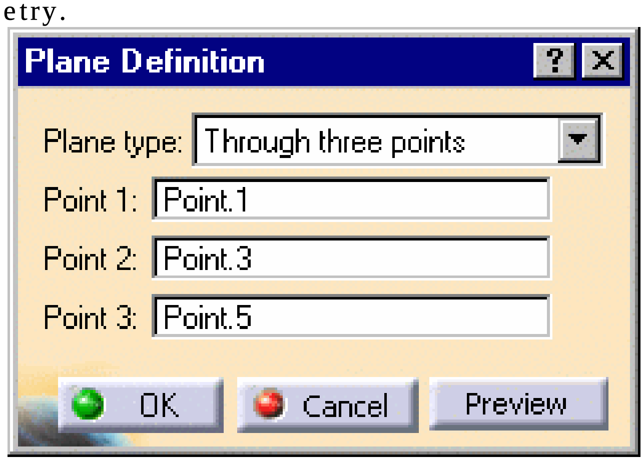
As many planes as indicated in the dialog box are created (including the one you were currently creating), each separated from the initial plane by a multiple of the **Angle** value.

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



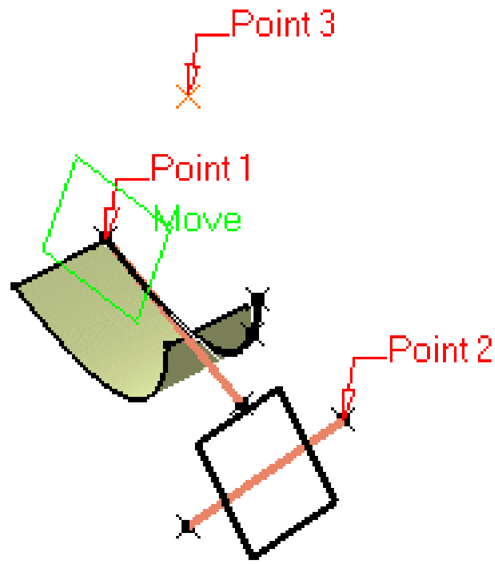
This plane type enables to edit the plane's parameters. Refer to Editing Parameters to find out how to display these parameters in the 3D geometry.

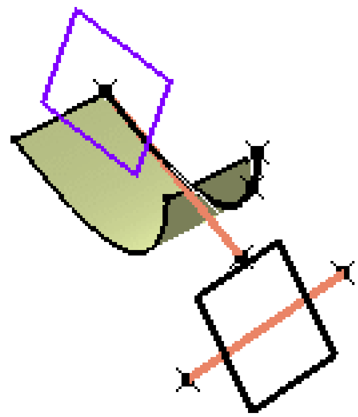
Through three points



- Select three points.

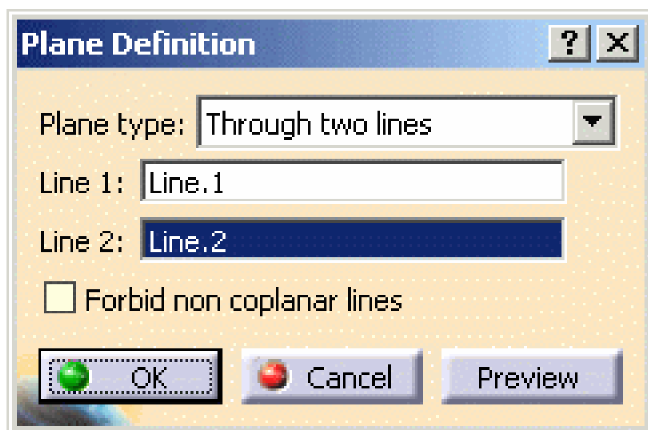
The plane passing through the three points is displayed. You can move it simply by dragging it to the desired location.



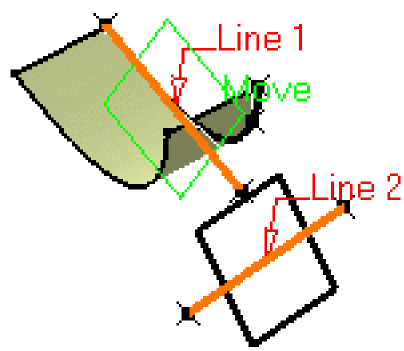


Through two lines

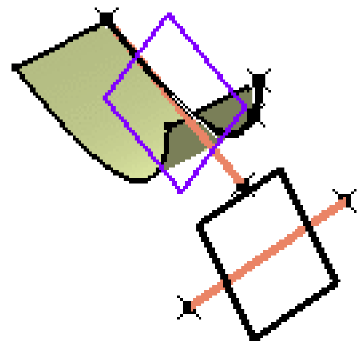
- Select two lines.



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.

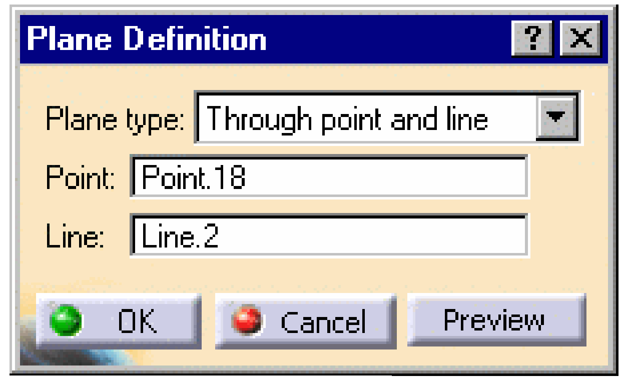


Check the **Forbid non coplanar lines button** to specify that both lines be in the same plane.

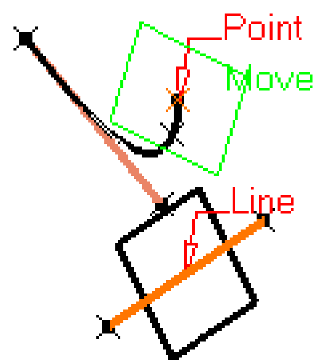


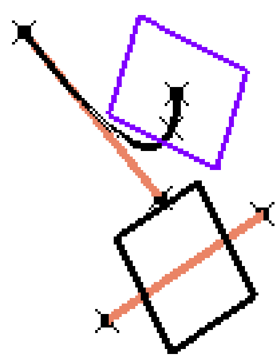
Through point and line

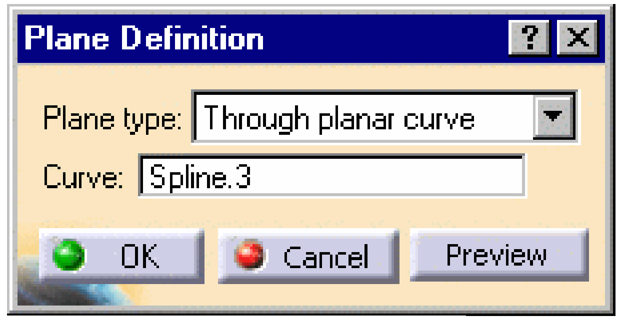
- Select a **Point** and a **Line**.



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

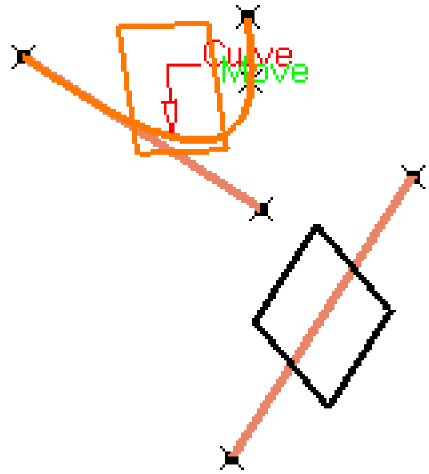




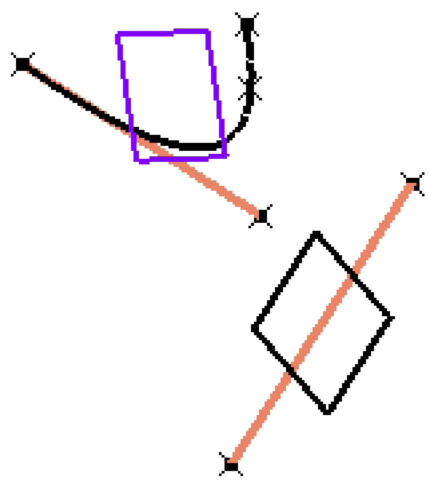


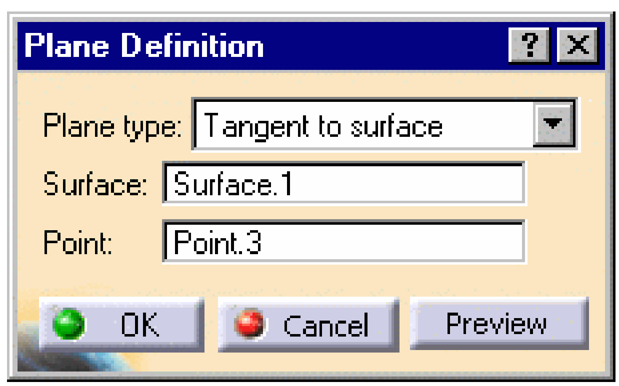
Through planar curve

- Select a planar **Curve**.



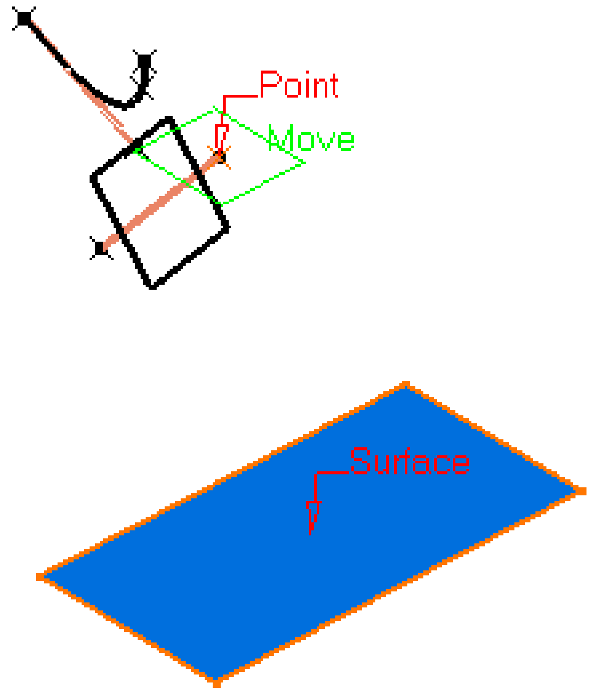
The plane containing the curve is displayed.



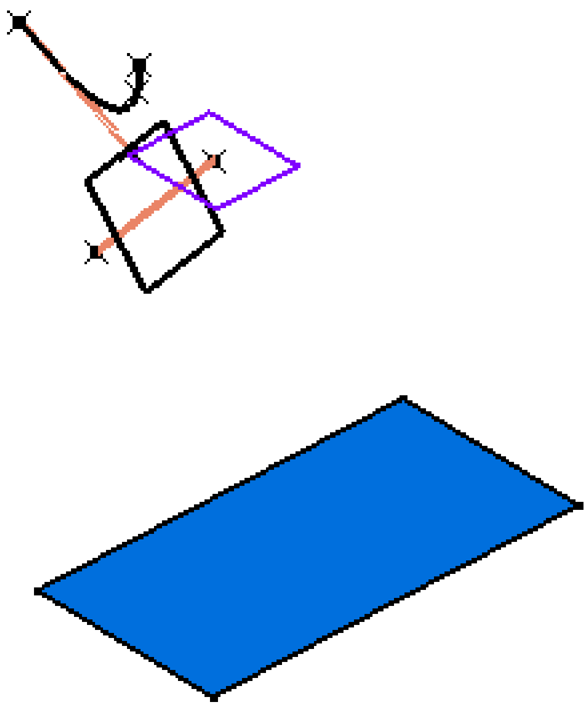


Tangent to surface

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

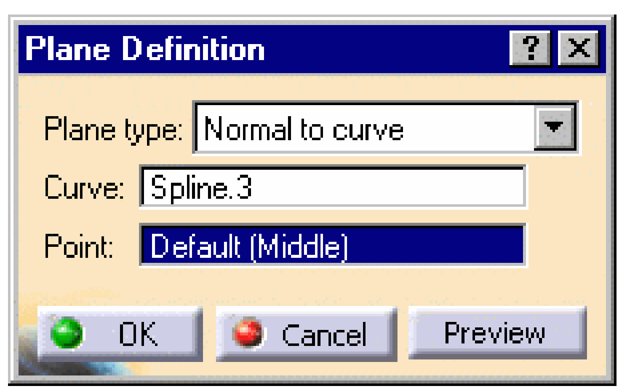


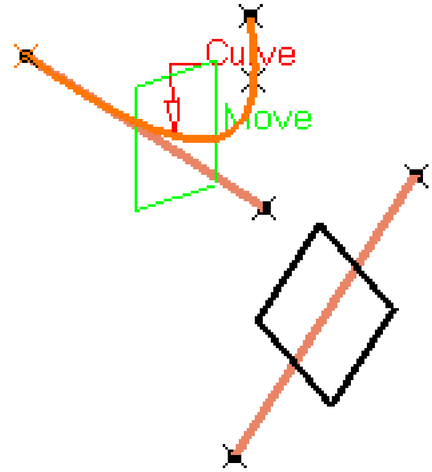
A plane is displayed tangent to the surface at the specified point.



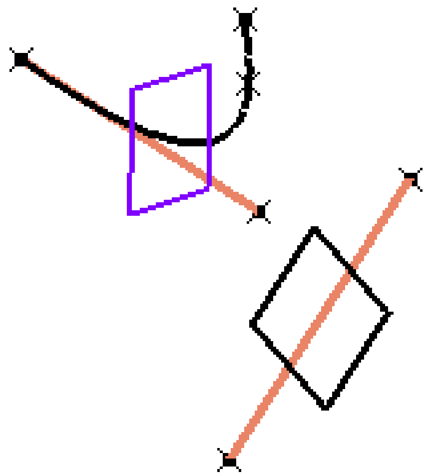
Normal to curve

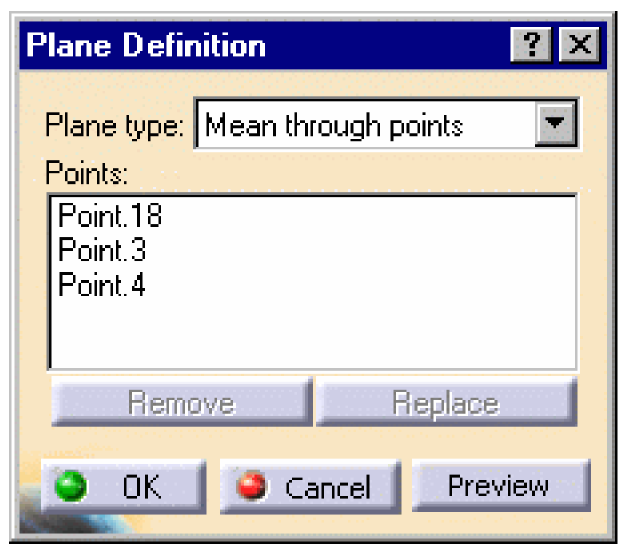
- Select a reference **Curve**.
- You can select a **Point**. By default, the curve's middle point is selecte.





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

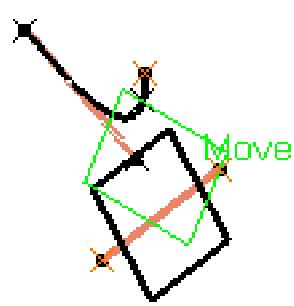




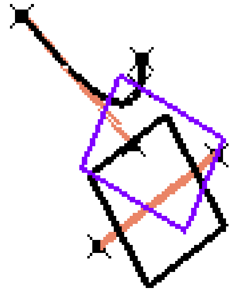
Mean through points

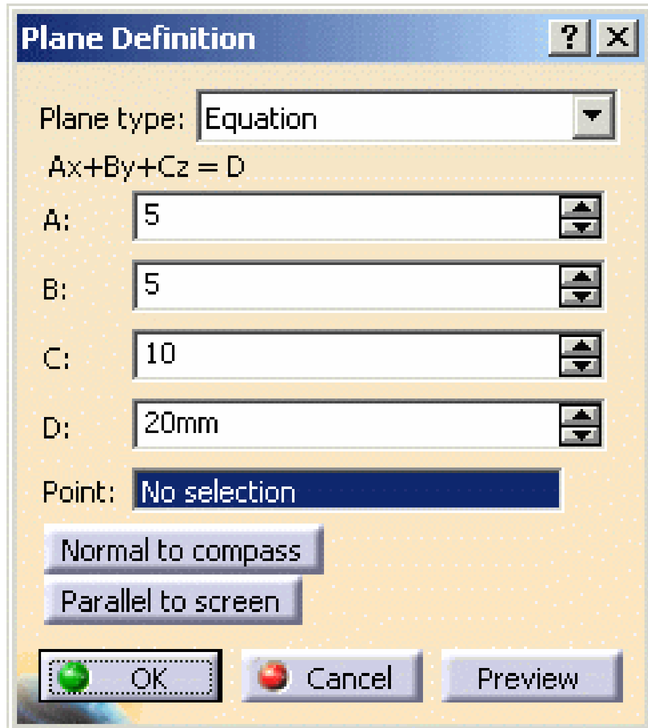
- Select three or more points to display the mean plane through these points.

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.

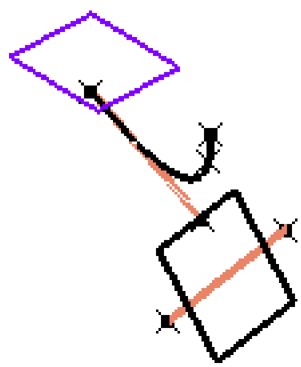




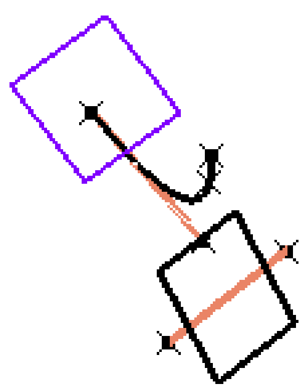
Equation

- 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

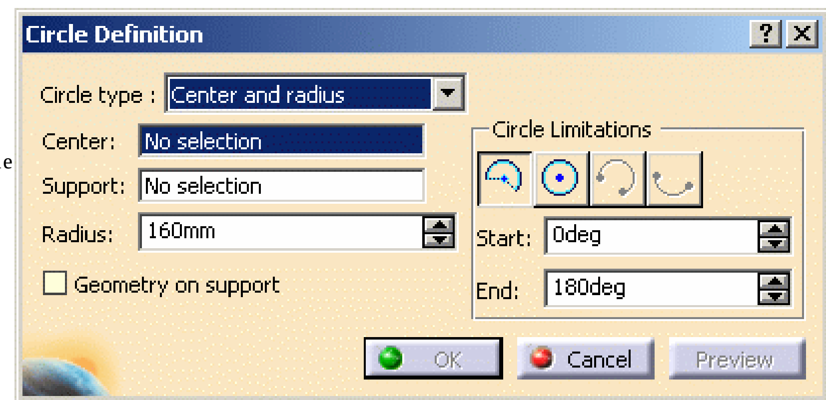
Open the Circles1.CATPart document.
Please note that you need to put the desired geometrical set in show to be able to perform the corresponding scenario.

1. Click the **Circle** icon.

The Circle Definition dialog box appears.

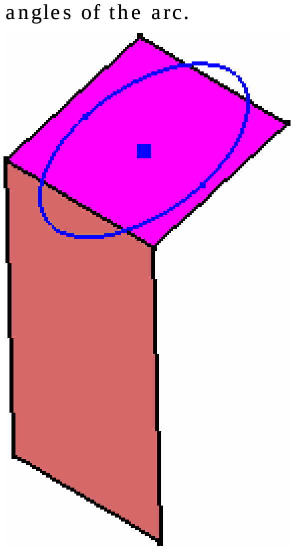
2. Use the combo to choose the desired circle type.

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.

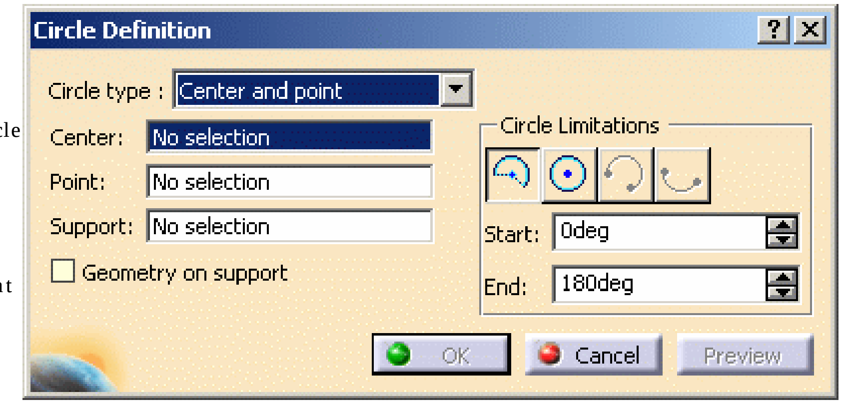
Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the **Start** and **End** angles of the arc.



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.

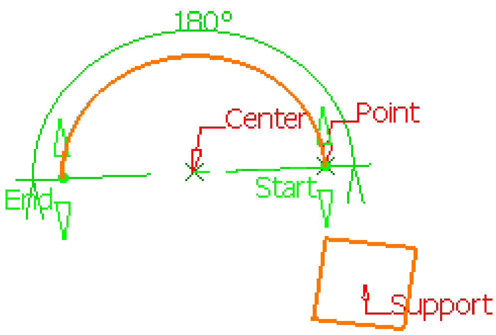
Center and point



- 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.

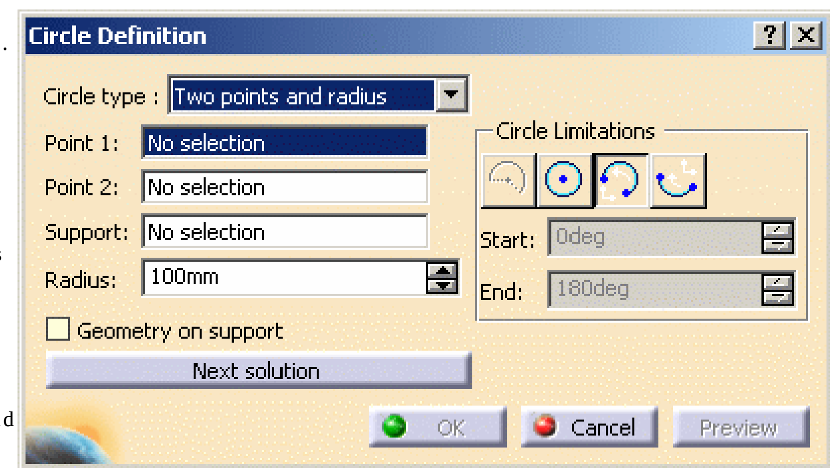
Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the **Start** and **End** angles of the arc.



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

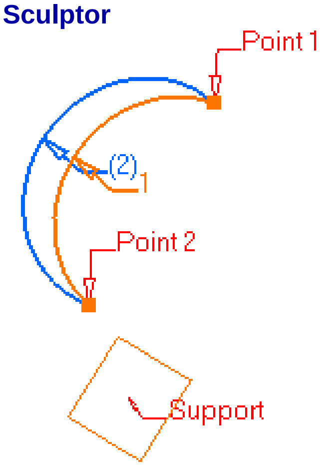


- Select two points on a surface or in the same plane.
- Select the **Support** plane or surface.
- Enter a **Radius** value.

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.

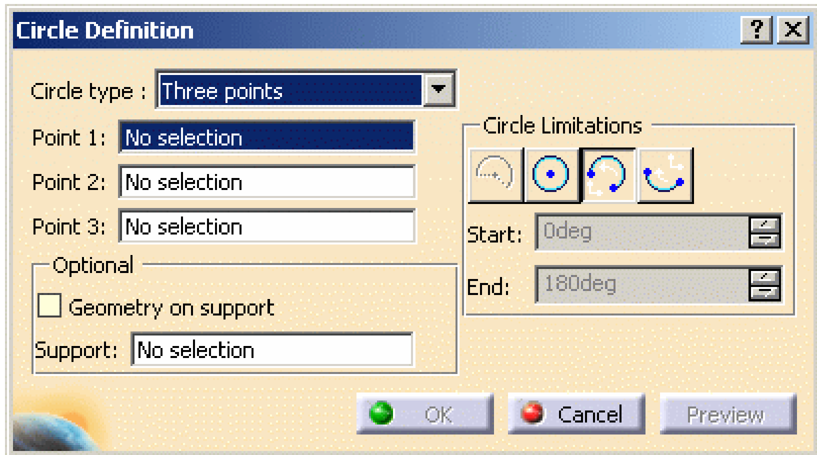


You can select the **Geometry on Support** check box if you want the circle to be projected onto a support surface.

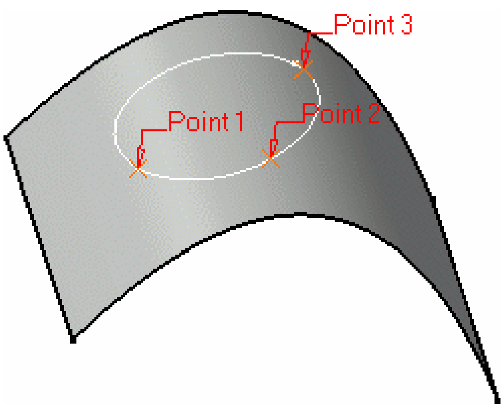
In this case just select a support surface.

Three points

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



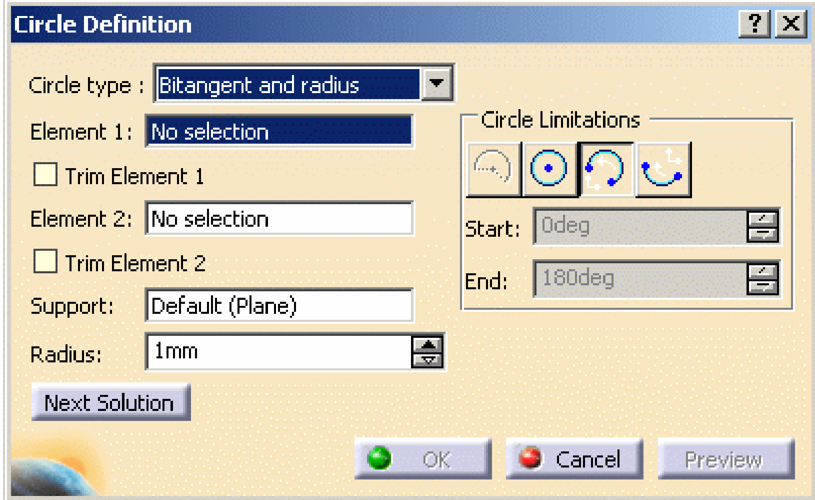
Depending on the active **Circle Limitations** icon, the corresponding circle or circular arc is displayed.
For a circular arc, you can specify the trimmed or complementary arc using the two of the selected points as end points.



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



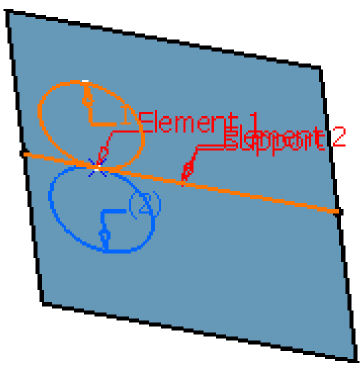
- Select two **Elements** (point or curve) to which the circle is to be tangent.
- Select a **Support** surface.

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.

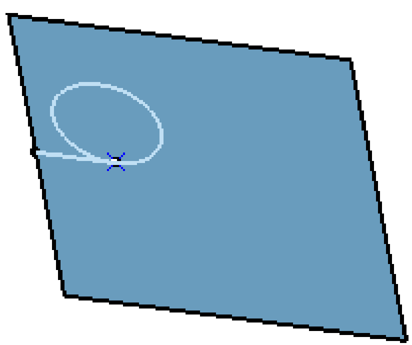
This automatic support definition saves you from performing useless selections.

- 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.

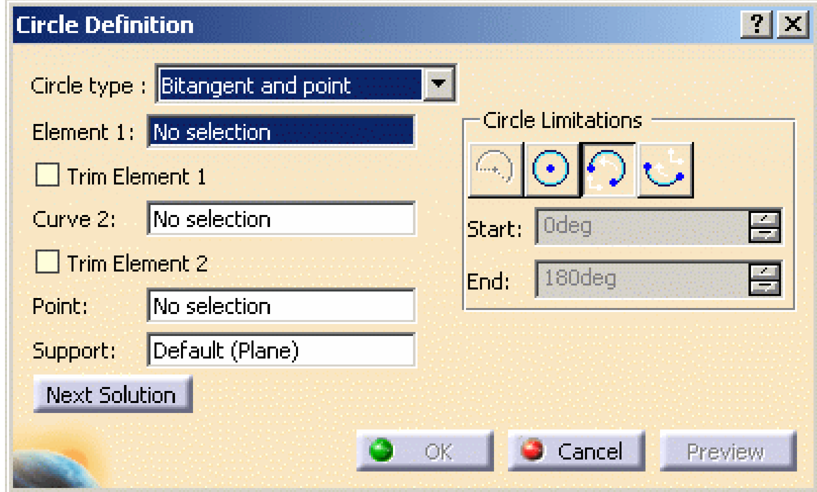


You can select the **Trim Element 1** and **Trim Element 2** check boxes to trim the first element or the second element, or both elements.
Here is an example with Element 1 trimmed.



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.

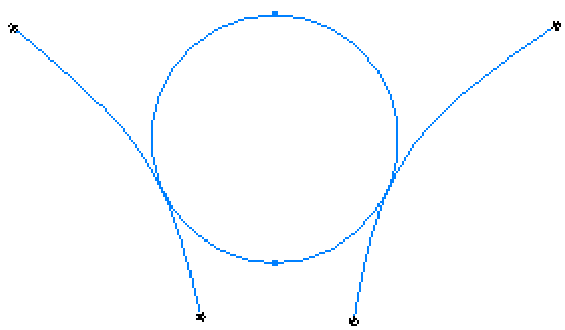
The point will be projected onto the curve.

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.

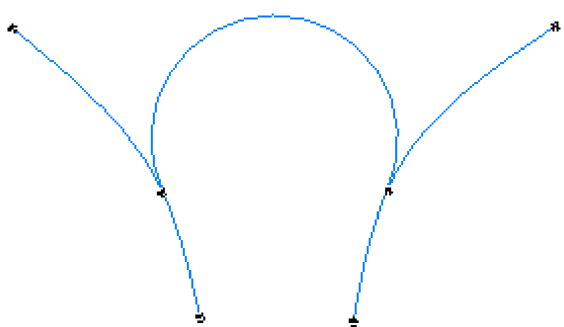
- Several solutions may be possible, so click in the region where you want the circle to be.

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

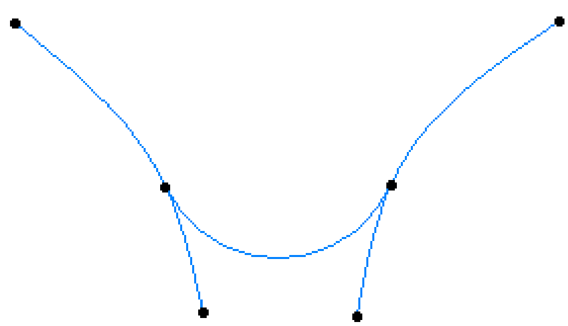


Complete circle

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

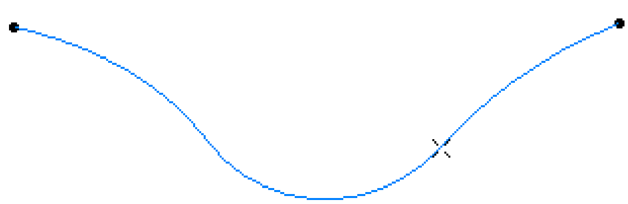


Trimmed circle



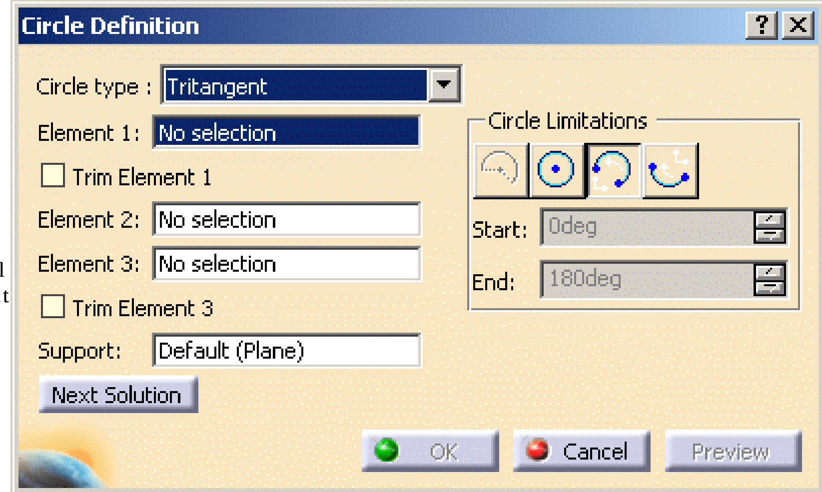
Complementary trimmed circle

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



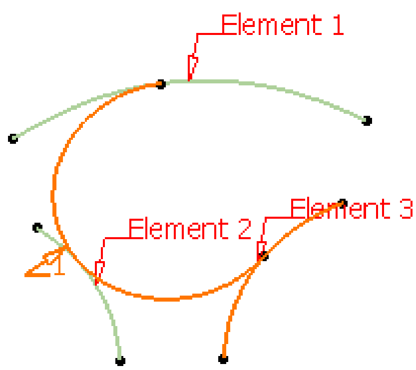
- Select three **Elements** to which the circle is to be tangent.
- Select a **Support** planar surface.

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.

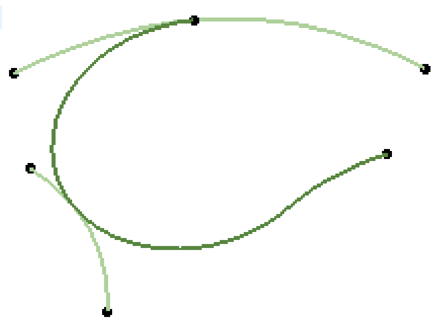
This automatic support definition saves you from performing useless selections.

- 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.



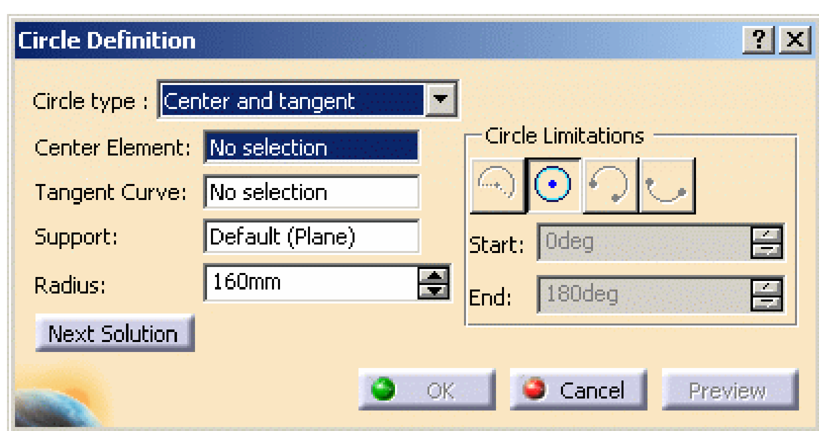
You can select the **Trim Element 1** and **Trim Element 3** check boxes to trim the first element or the third element, or both elements.
Here is an example with Element 3 trimmed.



These options are only available with the Trimmed Circle limitation.

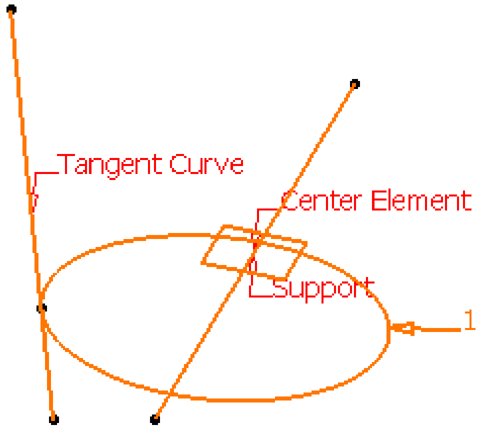
Center and tangent

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



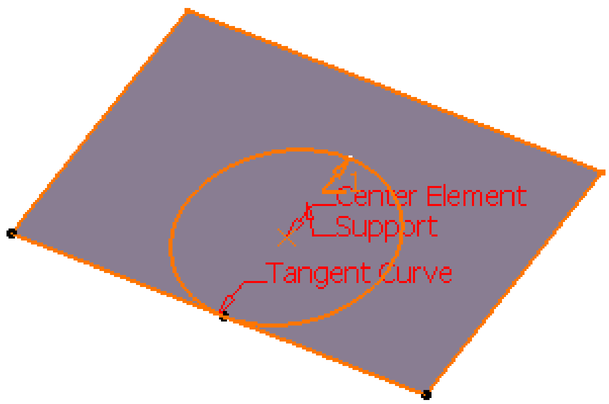
1. Center curve and radius

- 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.

Generic Tools

Managing the Compass

Managing the Compass

This task shows you how to quickly manage the compass orientation.

Create a planar surface.

Press F5 or click the **Quick Compass Orientation** icon .

The Quick compass orientation toolbar is displayed.

Here are summarized the main features of its eight icons:

- Click the **Flip to UV or XY** , **Flip to VW or YZ** or **Flip to WU or XZ** icon to switch the compass base to the three planes of its trihedron.

If the compass is "in" the main axes of the model, the icons indicate X, Y and Z otherwise U, V and W.

- Click the **Most Seen Plane** icon to activate and deactivate the **Most Seen Plane** mode.

- Click the **Set Compass Orientation** icon or press F6 to orientate the compass by selecting either an existing plane or three points (via the Autodetection command).
The point selection is based on Autodetection parameters.

Please refer to the chapter called Editing Curves Using Control Points documentation to orientate the compass using control points.

- Click the **Reset Compass to XYZ** icon or press F7 to reset the compass parallel to the main axes (X, Y and Z) of the model.
This option is not active when the compass is already set according to the axes.

- Click the **In Model or on Perch** icon to switch the compass from the perch to the model or vice versa.
The origin is kept in the model until the toolbar remains open.

- Click the **Create Compass Plane** icon or press F8 to drop the compass plane, that is to create a plane corresponding to the compass basis.
  This icon is activated only when the compass is in the model.

These four shortcuts (F5, F6, F7 and F8) are effective only when the Quick compass orientation toolbar is displayed. Therefore when you first hit one of the keys, the toolbar is displayed and the shortcuts are effective from then on.

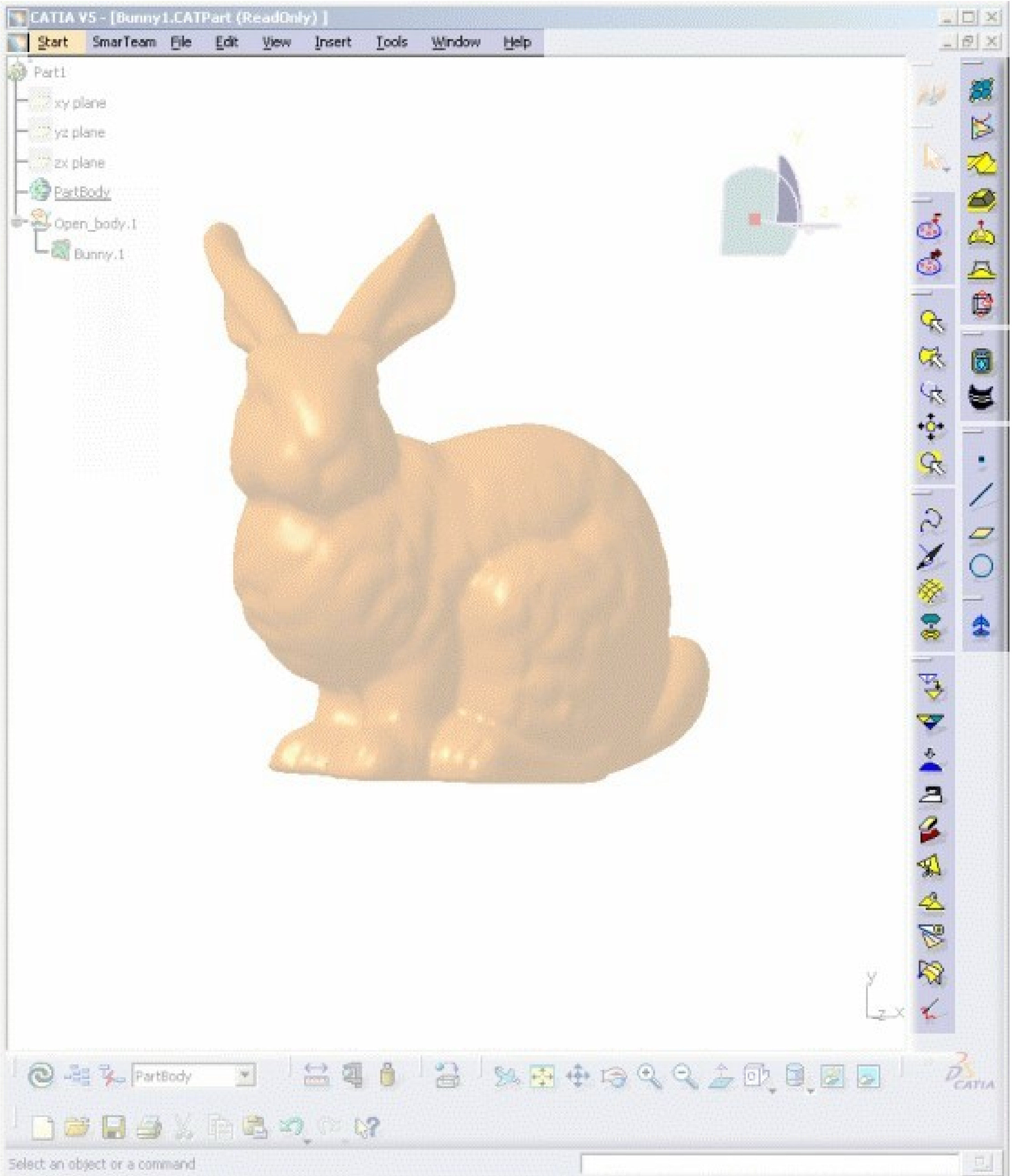
The Quick compass orientation toolbar remains active until you close it by clicking the cross in the upper-right corner or, if you have activated it with the icon, by clicking the icon again.

By default, elements are created in the current active plane as defined using the Current plane orientation toolbar containing the **Flip to UV or XY**, **Flip to VW or YZ** or **Flip to WU or XZ** icons.

Workbench Description

This section contains the description of the icons, menus and Historical Graph that are specific to the CATIA - Shape Sculptor workbench, which is shown below.

You can click the hotspots on this image to see the related documentation.



Menu Bar

This section presents the tools and commands which are available in the Shape Sculptor workbench. Many other operations are documented in the *Infrastructure User's Guide*.

Start	File	Edit	View	Insert	Tools	Windows	Help

File

The File menu lets you perform file creation, opening saving, printing operations

Edit

The Edit menu lets you manipulate selected objects. Refer to the *Infrastructure User's Guide* and *Part Design User's Guide*.

View

The View menu lets you view document contents Refer to the *Infrastructure User's Guide*.

Insert

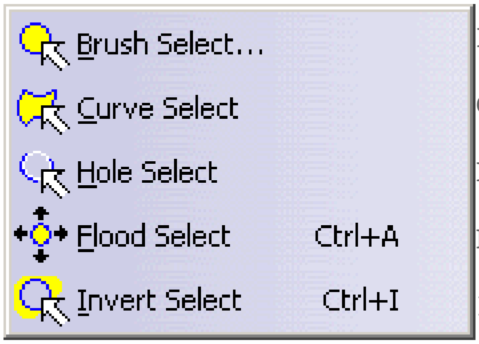
The Insert menu lets you insert Shape Sculptor's elements.

For...	See...
Open body	Managing Geometrical Sets
Input/Output	Insert -> Input/Output
Component Selection	Insert -> Component Selection
Creation	Insert -> Creation
Editing	Insert -> Editing
Modeling	Insert -> Modeling
Analysis	Insert -> Analysis
Wireframe	Insert -> Wireframe

Insert -> Input/Output

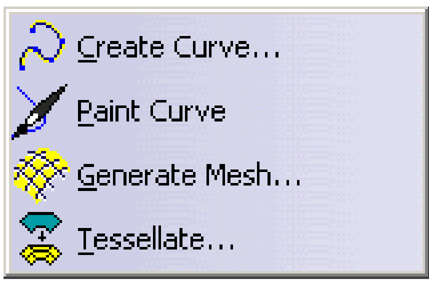
For...	See...
Import...	Importing Files
Export...	Exporting to STL

Insert -> Component Selection



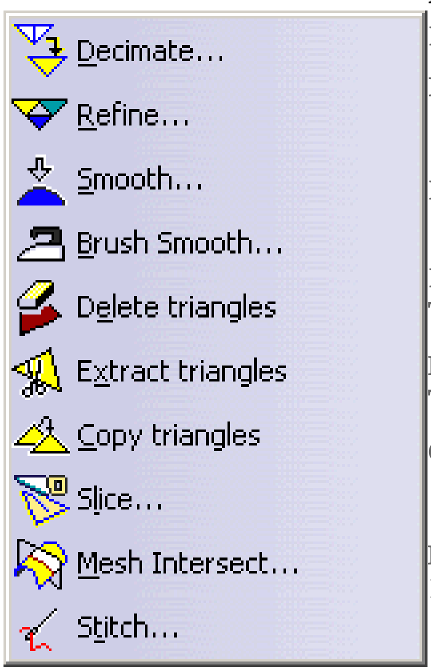
For...	See...
Brush Select...	Selecting With a Brush
Curve Select	Selecting With Curves
Hole Select	Selecting Holes
Flood Select	Selecting With Flood
Invert Select	Inverting the Selection

Insert -> Creation



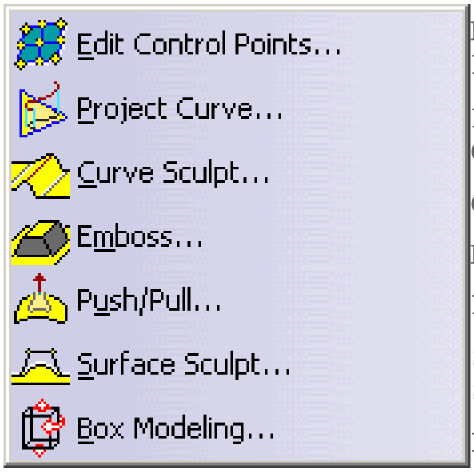
For...	See...
Create Curve...	Creating 3D Curves
Paint Curve	Creating Paint Curves
Generate Mesh...	Generating Meshes
Tessellate...	Tessellating

Insert -> Editing



For...	See...
Decimate...	Decimating
Refine...	Refining
Smooth...	Smoothing
Brush Smooth...	Smoothing With a Brush
Delete Triangles	Deleting Triangles
Extract Triangles	Extracting Triangles
Copy Triangles	Copying Triangles
Slice...	Slicing
Mesh Intersect...	Intersecting Meshes
Stitch...	Stitching

Insert -> Modeling

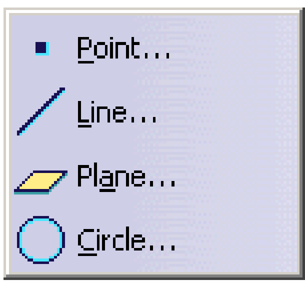


For...	See...
Edit Control Points...	Editing Curves Using Control Points
Project Curve...	Projecting Curves
Curve Sculpt...	Sculpting Curves
Emboss...	Embossing
Push/Pull...	Pushing / Pulling a Mesh
Surface Sculpt...	Sculpting With Surfaces
Box Modeling...	Modeling Using a Box

Insert -> Analysis

For...	See...
Display Options...	Display Options and Graphic Properties
Highlight	Analyzing Using Highlights

Insert -> Wireframe



For...	See...
Point...	Creating Points
Line...	Creating Lines
Plane...	Creating Planes
Circle...	Creating Circles

Window

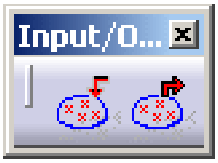
The Window menu lets you arrange document windows in relation one to the other. Refer to the *Infrastructure User's Guide*.

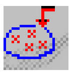
Help

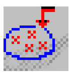
The Help menu lets you get help on the currently active command, and the product in general. Refer to the *Infrastructure User's Guide*.

Input / Output Toolbar

This toolbar contains the following tools to manage the import and export of polygonal meshes.

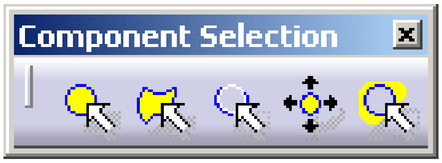


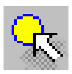
 See Importing Files

 See Exporting to STL

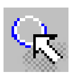
Component Selection Toolbar

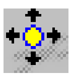
This toolbar contains the following tools to manage the selection of triangles on the polygonal mesh.



 See Selecting Using a Brush

 See Selecting Using Curves

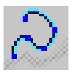
 See Selecting Using a Hole

 See Selecting Using Flood

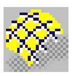
 See Inverting the Selection

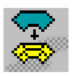
Creation Toolbar

This toolbar contains the following tools to manage the creation of curves and polygonal meshes.

 See Creating 3D Curves

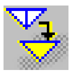
 See Creating Paint Curves

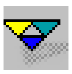
 See Generating Meshes

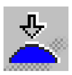
 See Tessellating

Editing Toolbar

This toolbar contains the following tools to manage the operations that can be performed on a polygonal mesh.

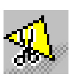
 See Decimating

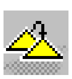
 See Refining

 See Smoothing

 See Smoothing Using a Brush

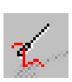
 See Deleting Triangles

 See Extracting Triangles

 See Copying Triangles

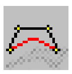
 See Slicing

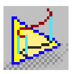
 See Intersecting Meshes

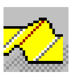
 See Stitching

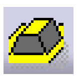
Modeling Toolbar

This toolbar contains the following tools to model the polygonal mesh.

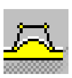
 See Editing Curves Using Control Points

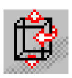
 See Projecting Curves

 See Sculpting Curves

 See Embossing

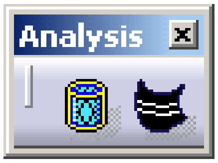
 See Pushing / Pulling a Mesh

 See Sculpting Surfaces

 See Modeling Using a Box

Analysis Toolbar

This toolbar contains the following tools to manage the options and graphic properties.

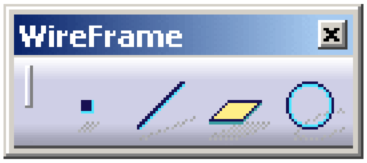


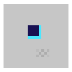
 See Display Options and Graphic Properties

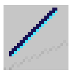
 See Analyzing Using Highlights

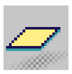
Wireframe Toolbar

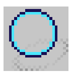
The Wireframe Toolbar contains the following tools:



 See Creating Points

 See Creating Lines

 See Creating Planes

 See Creating Circles

Generic Tools Toolbar

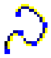
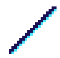
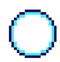
This toolbar contains the following tool to manage the compass.

 See Managing the Compass

Specification Tree

Within the Shape Sculptor workbench, you can generate a number of elements that are identified in the specification tree by the following icons.

Further information on general symbols in the specification tree are available in Symbols Used in the Specification Tree.

	3D Curve		Point
	Mesh Curve		Line
	Project Curve		Plane
	Mesh Generation		Circle
	Tessellation		
	Stitching		
	Polymesh		

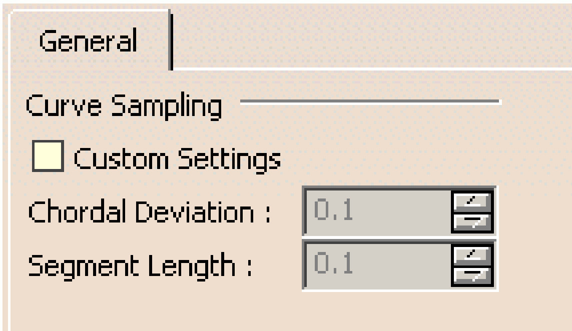
Customizing For Shape Sculptor

This task explains how to customize general settings for the Shape Sculptor workbench.

1. Select the **Tools -> Options** menu item.

   The Options dialog box is displayed.

2. Select the **Shape** category then the **Shape Sculptor** subcategory to display the General tab:



3. Check **Custom Settings** to enable the modification of parameters that control the precision of the curves.
4. Define the **Chordal Deviation** by entering a value or using the spinners.
5. Define the **Segment Length** by entering a value or using the spinners.
6. Click **OK** to confirm setting these options.

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Numerics

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