STL Rapid Prototyping

Overview

Welcome to the *STL Rapid Prototyping User's Guide!*
This guide is intended for users who need to become quickly familiar with the product.

This overview provides the following information:

- STL Rapid Prototyping in a Nutshell
- Before Reading this Guide
- Getting the Most Out of this Guide
- Accessing Sample Documents
- Conventions Used in this Guide

STL Rapid Prototyping in a Nutshell

STL Rapid Prototyping  helps the stereolithography specialist to build and manage STL files.

The *STL Rapid Prototyping User's Guide* has been designed to show you how to do that. Although stereolithography knowledge is not mandatory, it is recommended to have some technical background in the area.

Before Reading this Guide

Prior to reading the *STL Rapid Prototyping User's Guide*, you are recommended to have a look at the *Infrastructure User's Guide* for information on the generic capabilities common to all products.

Getting the Most Out of this Guide

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

Accessing Sample Documents

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

What's New?

New Functionalities

Interactive Triangle Creation

Creates mesh triangles from points or edges.

Optimize

Improves the homogeneity of meshes.

Enhanced Functionalities

Activation, Remove Point, Split

A brush has been added for a quick and rough area selection.

Mesh Smoothing

Deviation statistics are now available.

Decimation

An analysis option and a progress bar have been added.

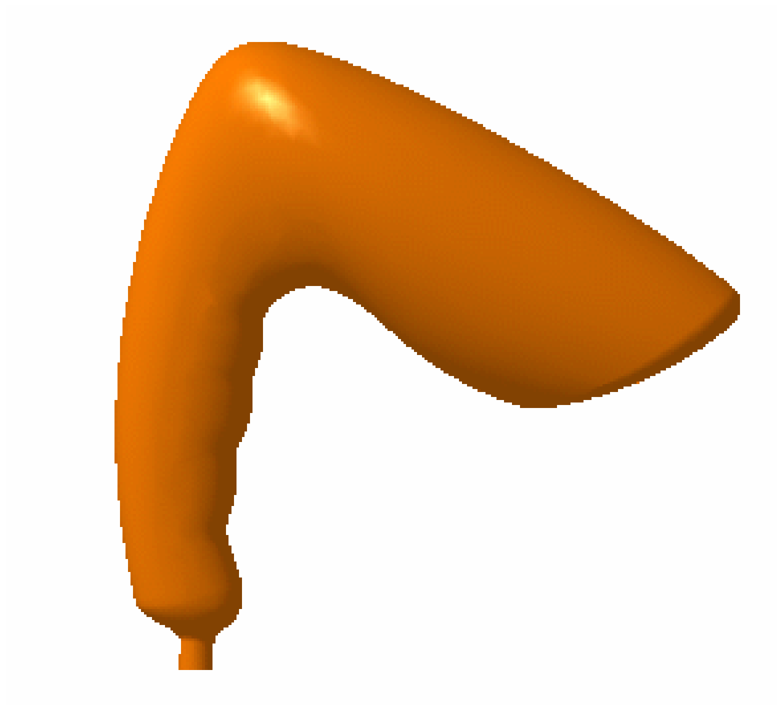
Getting Started

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

The main tasks proposed in the chapter are:

All together this scenario should take 15 minutes to complete.

The final cloud element will look like this:



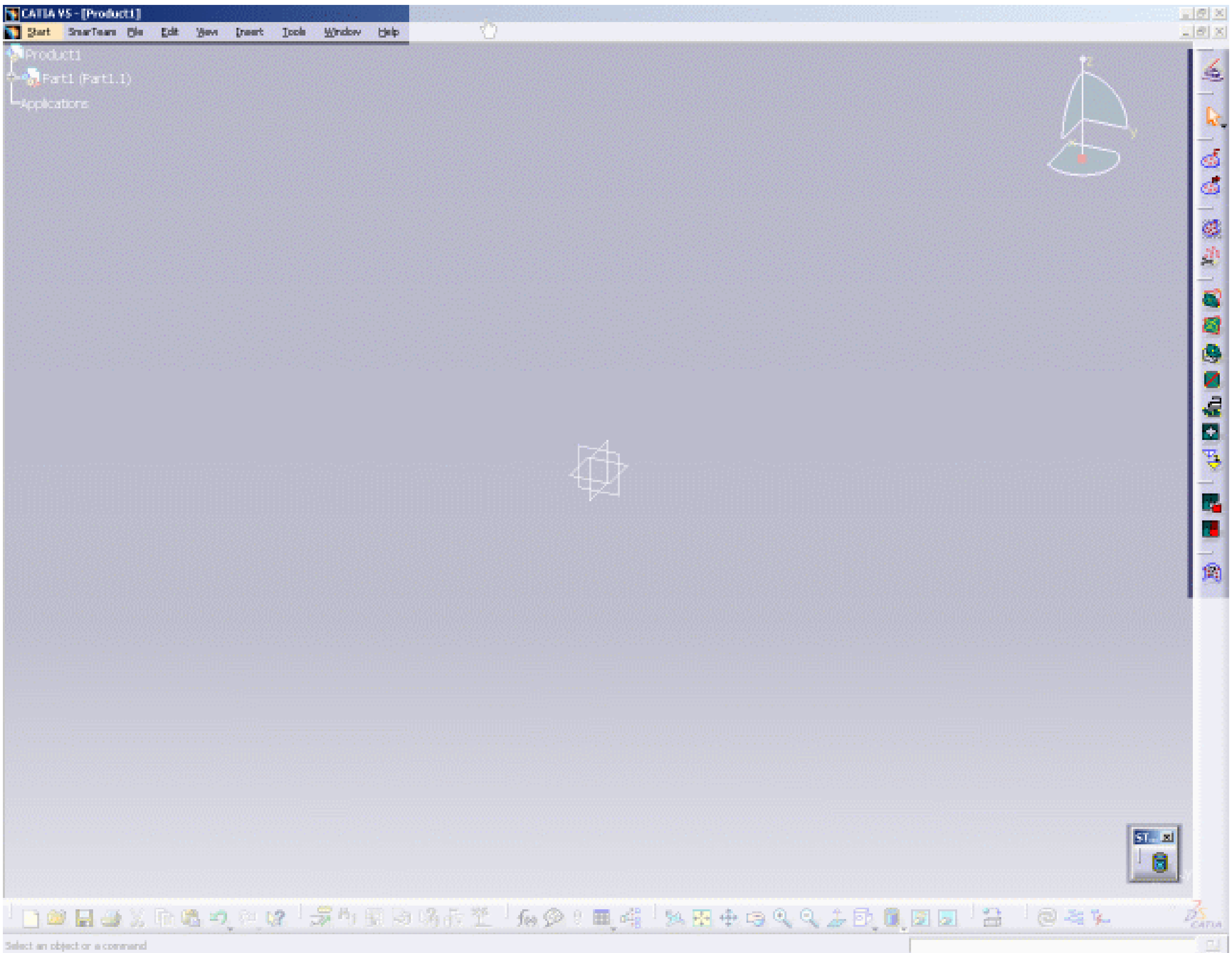
Creating a Basic Surface Tessellation

The first task will show you how to enter the STL Rapid Prototyping workbench and create a basic surface tessellation.

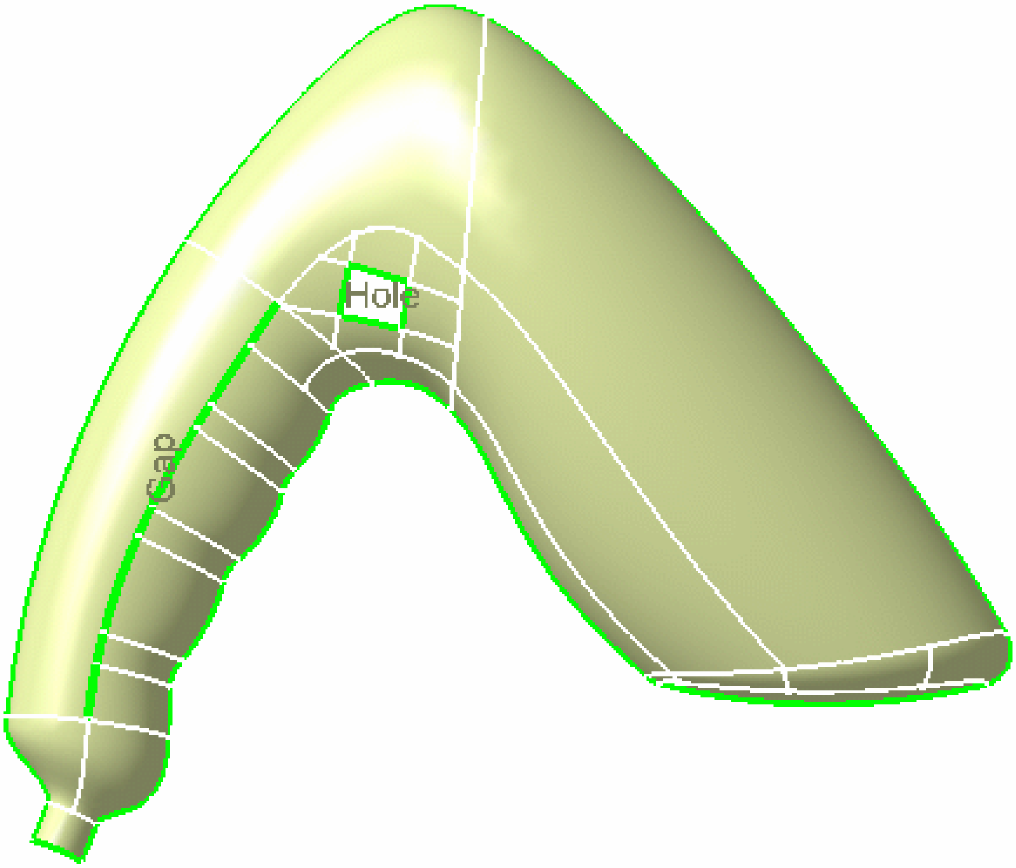
The only pre-requisites for this task is to have a current session running.

1. Choose **STL Rapid Prototyping** from the **Start / NC Manufacturing** menu.

The STL Rapid Prototyping workbench is displayed and ready to use.



2. Open the GettingStartedSTL.CATPart from the samples directory. It is a hairdryer, made of surfaces, with a gap and a hole in it.



3. Click the **Surface Tessellate** icon and select the hairdryer. Click **Apply** and **OK**.

You can see clearly the hole and the gap in the tessellation.

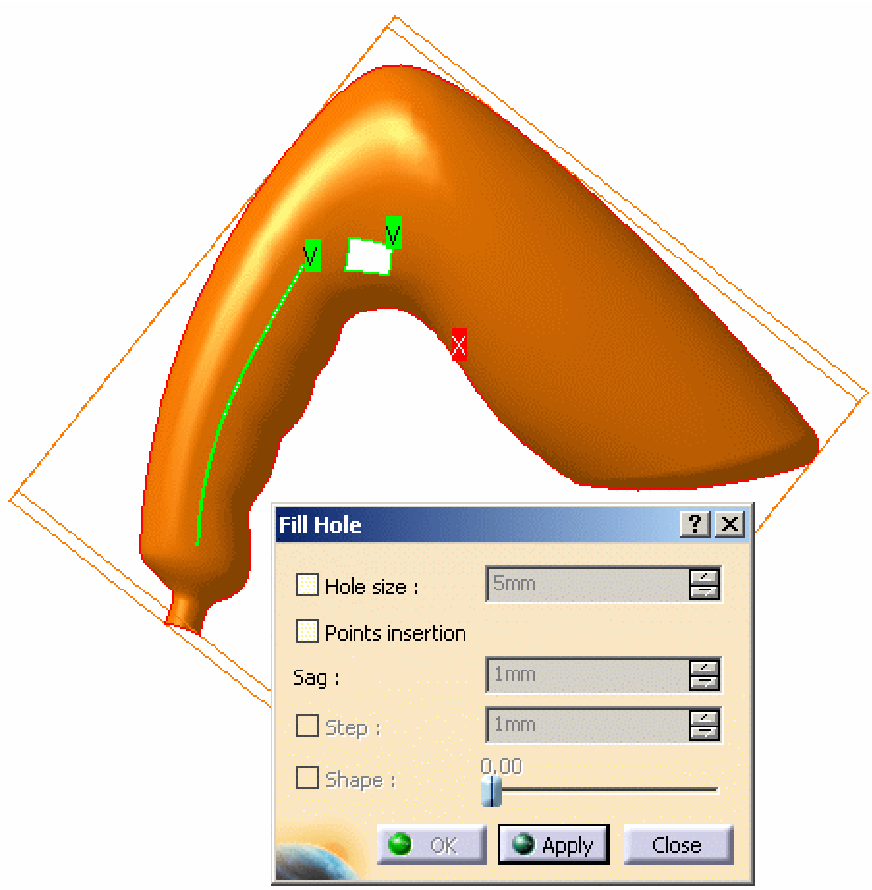
Repairing the Mesh

This task will show you how to repair a mesh.

Use the mesh you have created in the previous task.

Repairing the hole:

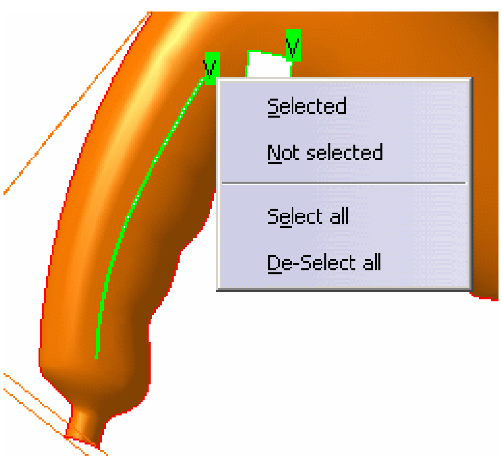
1. Click the **Fill Holes** icon and select the mesh.

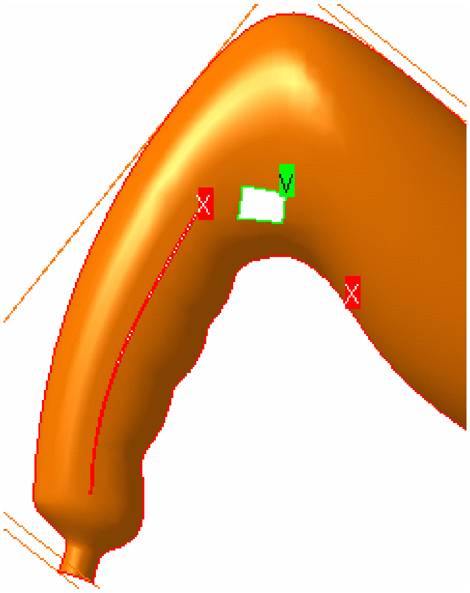


The exterior edge of the model is displayed in red, with an "x" meaning this edge is not candidate for a filling.

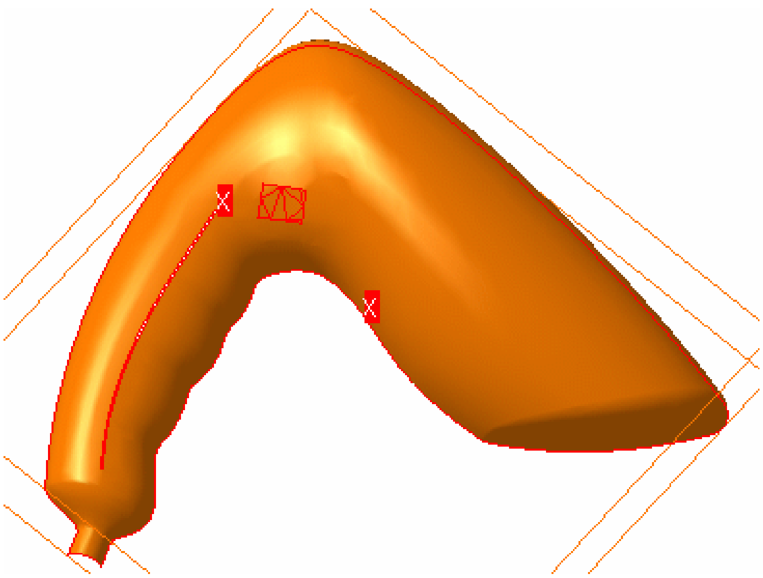
The two other edges in green with a "v" surround holes candidates for the filling.

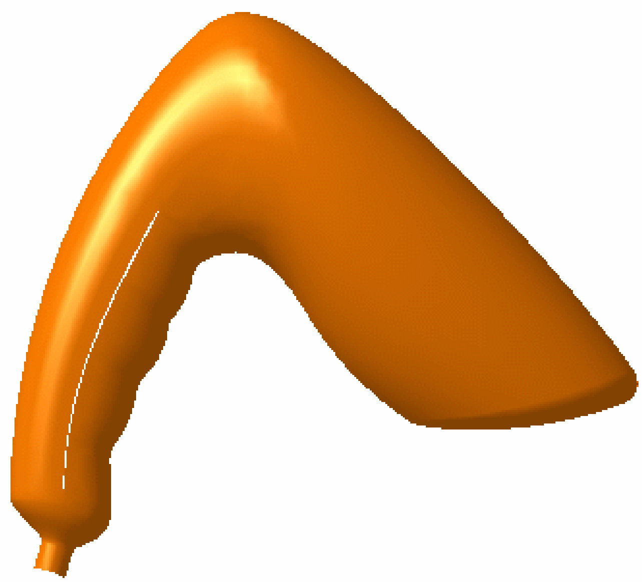
2. Place the cursor over the label of the long hole and call the contextual menu. Choose Not selected.



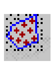


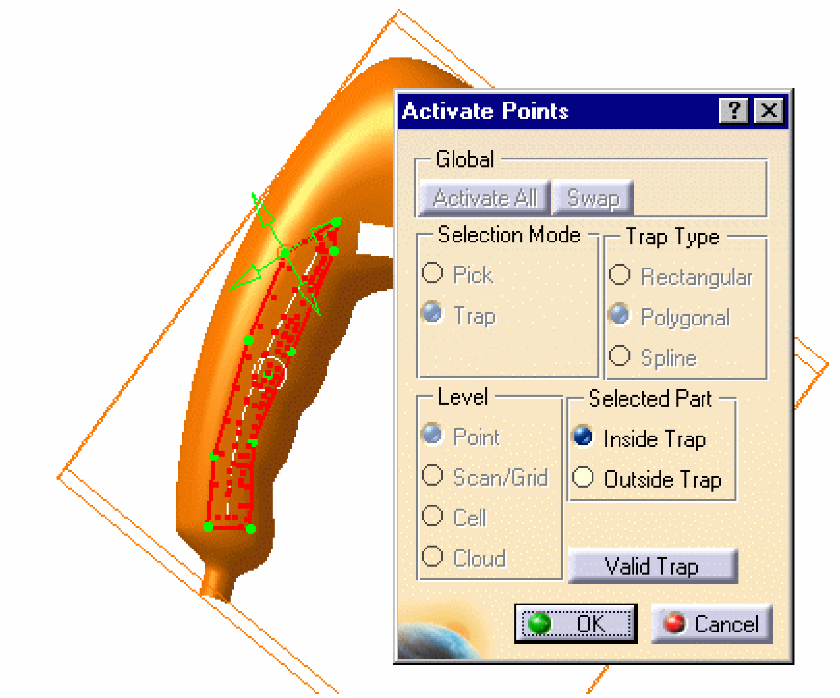
3. Click **Apply** and **OK**. The hole is filled:



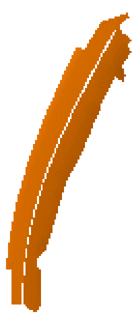


Repairing the gap:

1. Click the Define a Mesh Area icon  and select the mesh (Tessellation.1).
2. Draw a trap around the gap and check the **Inside Trap** option.

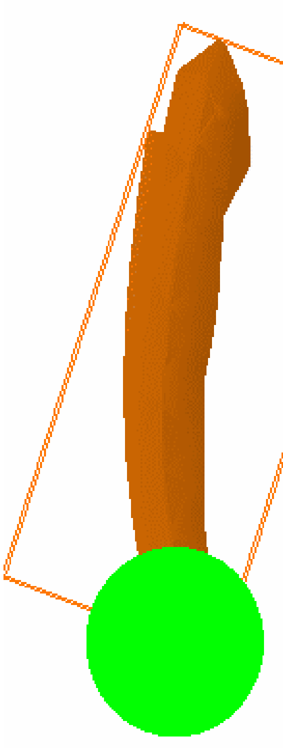


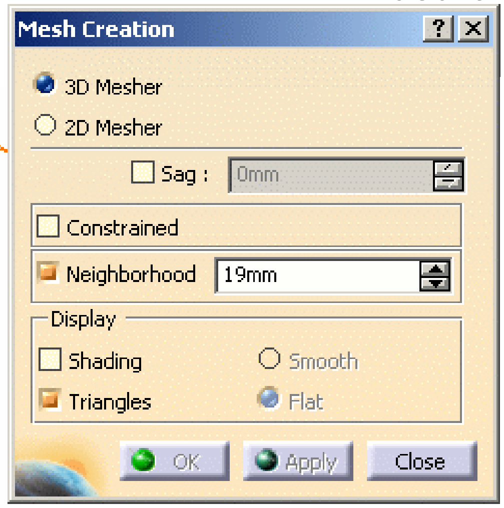
Click Apply and OK. A portion of the mesh is activated:

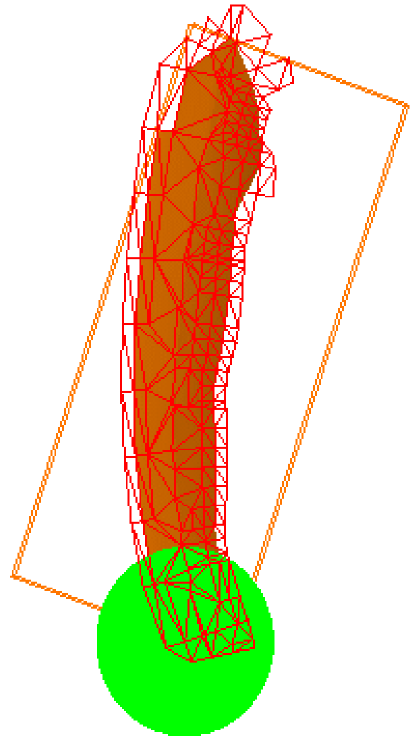


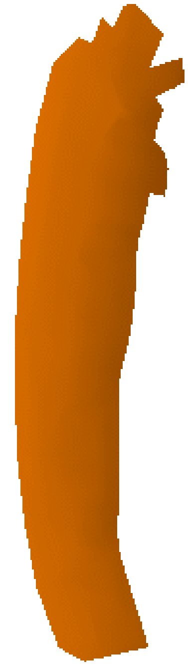
3. Click the **Mesh Regeneration** icon  and select the mesh (**Tessellation.1**). Check the **Constrained** option.

Click **Apply** and **OK**. A **MeshCreation.1** element is created in the specification tree: the mesh on this portion has been regenerated.

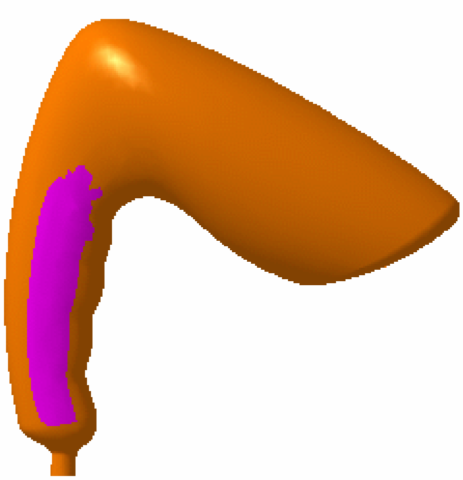


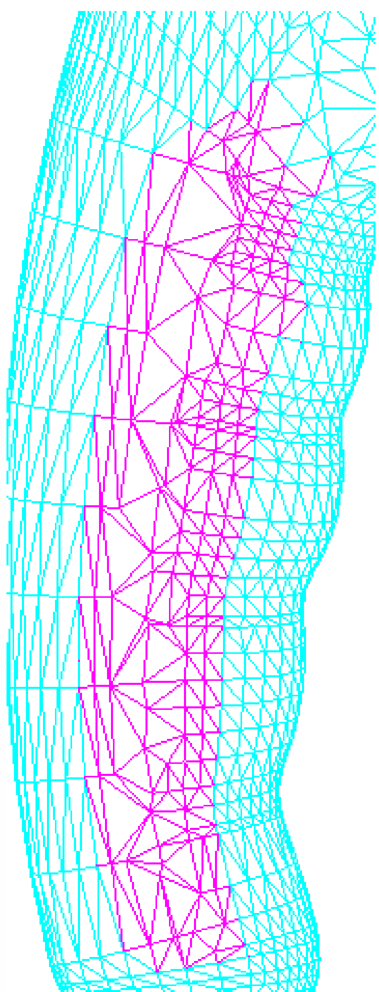




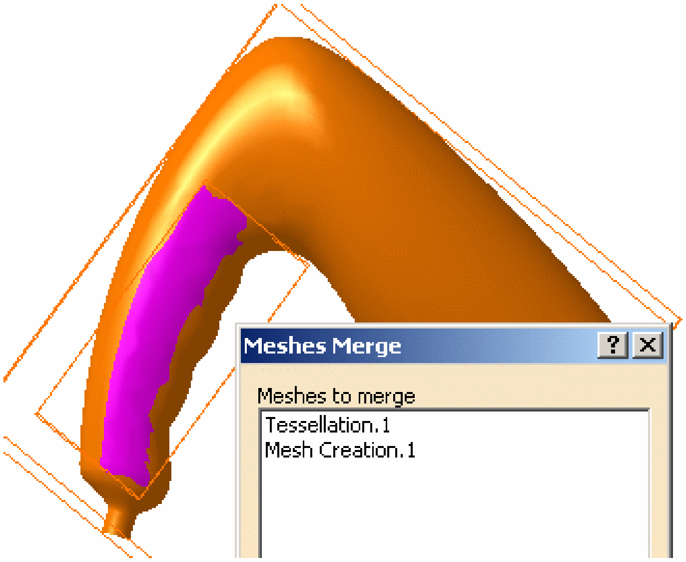


4. Select the **Tessellation.1** element in the specification tree and click the **Activation** icon. Push the **Activate All** button. Both meshes are now active (we have changed **MeshCreation.1** to pink). From the picture below you can see that they are complementary:

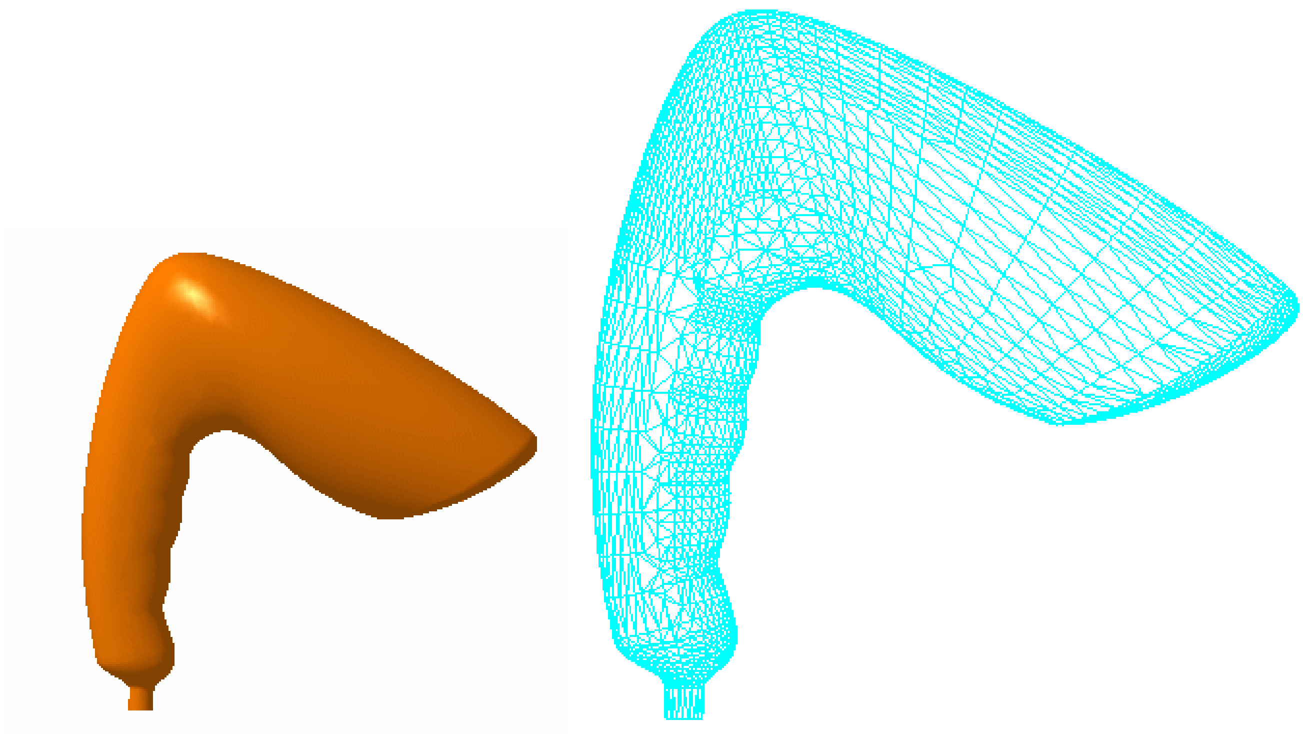




5. Click the **Merge** icon  and select the two meshes:



6. Click OK. A **Meshes Merge.1** element is created in the specification tree. You have now a flawless mesh.

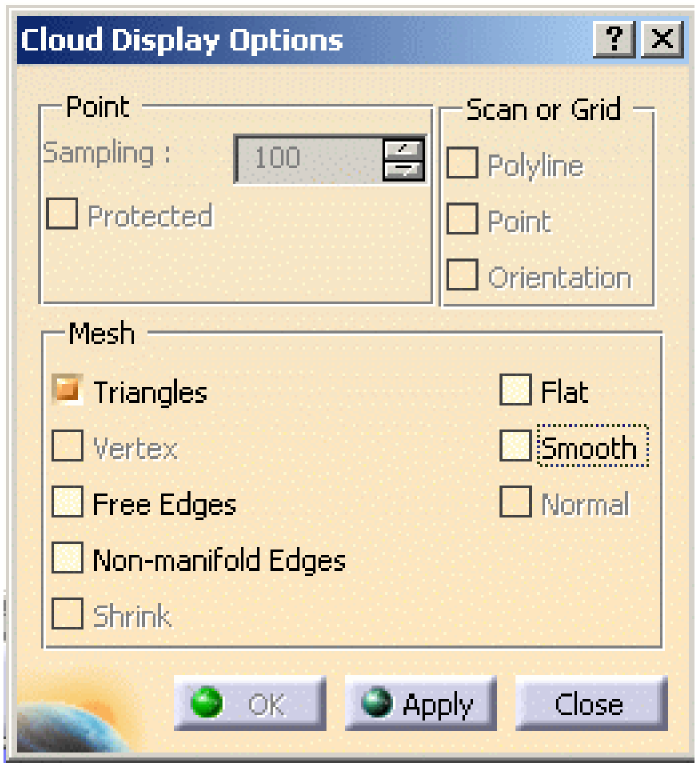


Checking the Mesh Quality

This task will show you how to quickly check the quality of the mesh.

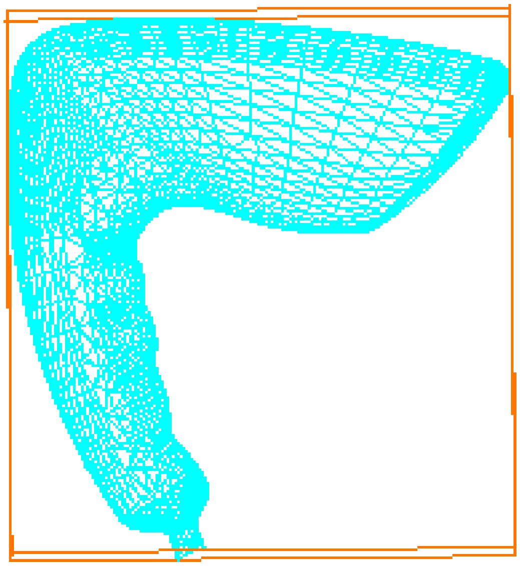
Use the mesh you have created in the previous task.

1. Click the **Display** icon and select the **Meshes Merge.1** mesh. The **Cloud Display Options** dialog box is displayed.

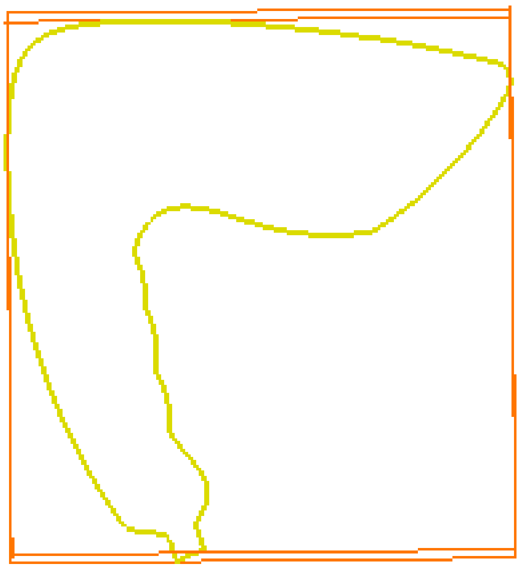


The default option is **Smooth**. You can check :

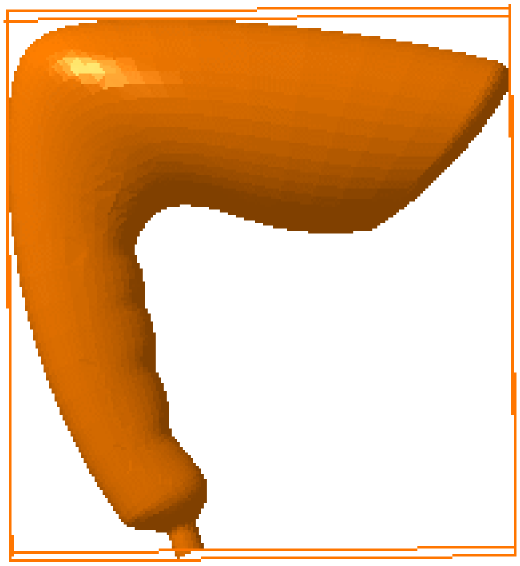
- the **Triangles** option:



- the **Free Edges** option:



- the **Non-manifold Edges** option: (none in our example)
- the **Flat** option:



You can combine all of them too.

User Tasks

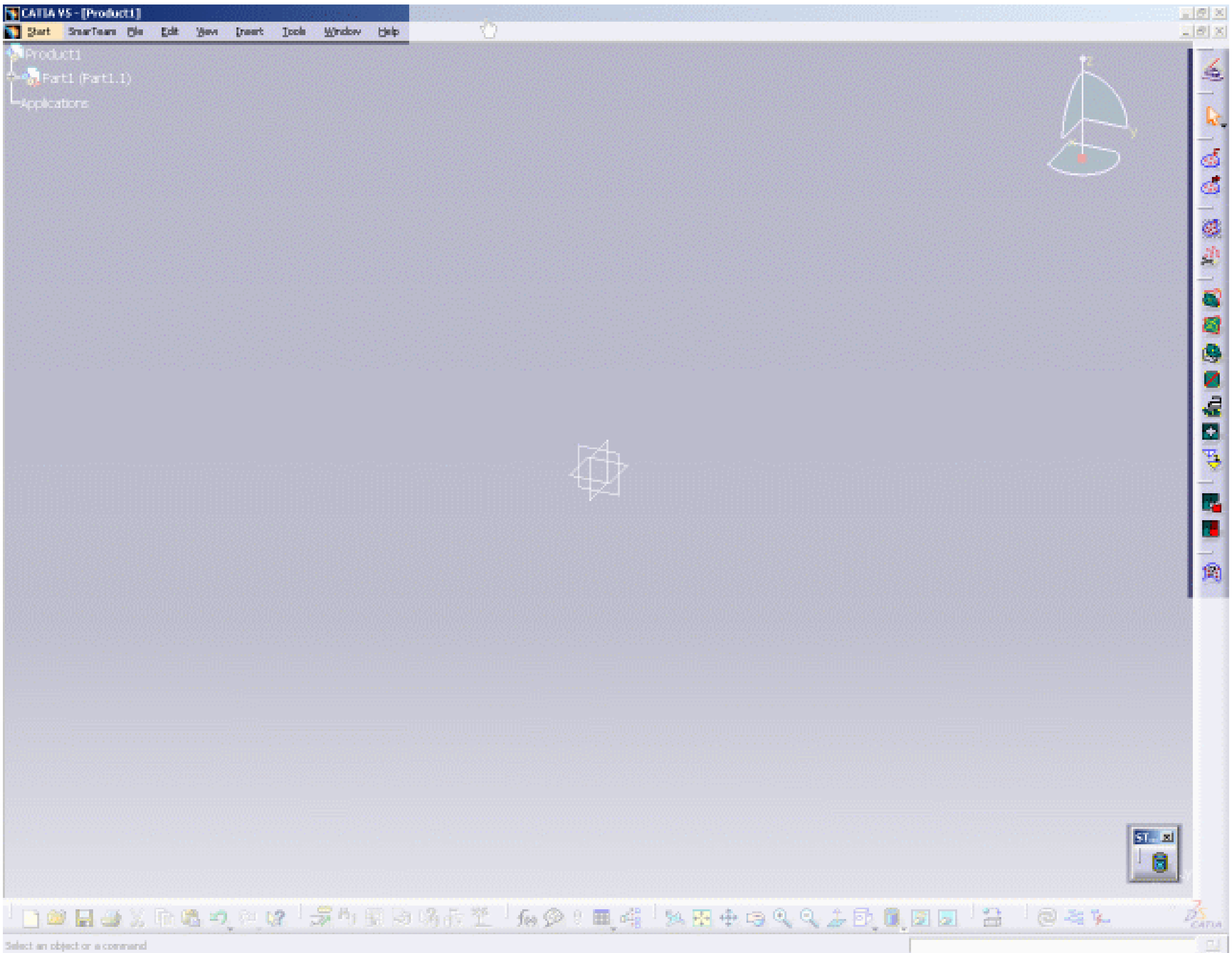
Opening the STL Rapid Prototyping Workbench

The first task will show you how to enter the STL Rapid Prototyping workbench and create a basic surface tessellation.

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

1. Choose **STL Rapid Prototyping** from the **Start / NC Manufacturing** menu.

The STL Rapid Prototyping workbench is displayed and ready to use.



2. You can then either import an STL file, or open a CAD model, or standard IGES or STEP data.

Using the Keyboard

Key	Command	Action
Shift	Activation Remove (Pick or Brush), Split	Deselects selected elements

STL Files

These actions deal with the import and the export of files

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.

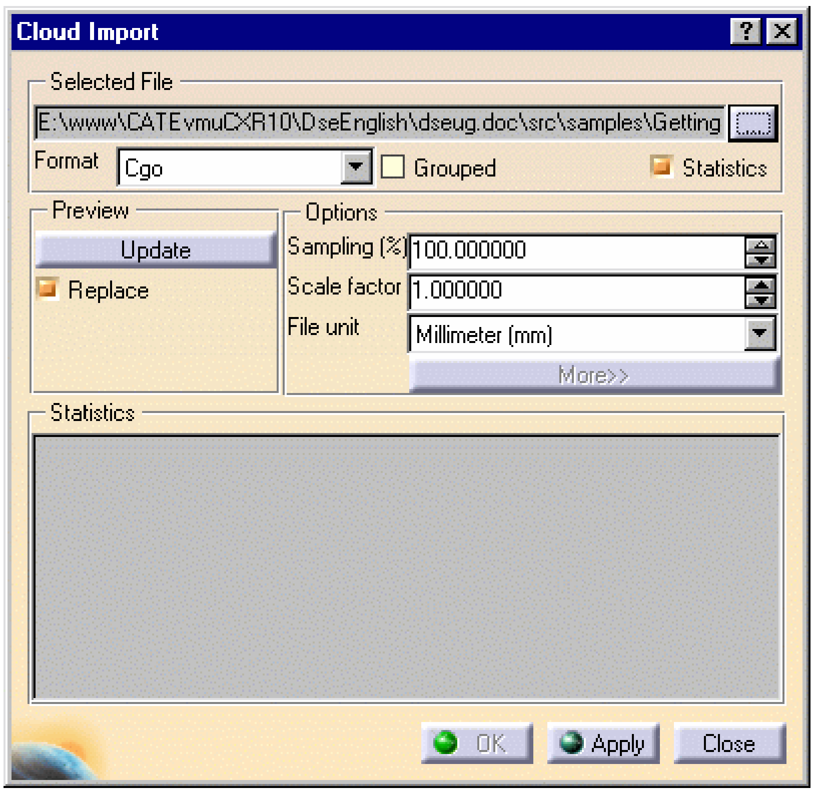
4. Check the box **Statistics** to display information about the model you are importing. If you want to import several files in one shot, please refer to the Grouped explanations.

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.

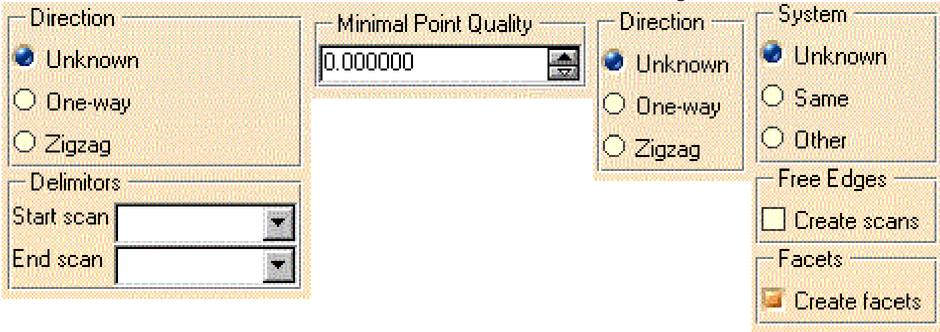
- Enter the **Scale factor** to apply to the model, as well as the **Unit** used in the file.



- 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.

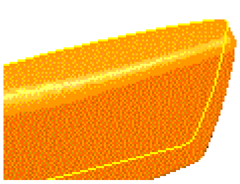
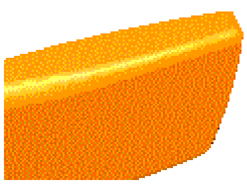
Minimal Point Quality is used to clean Atos file from invalid points. The quality value of a point lies between 0 and 255 (low to high). Choose a value to ignore points below that value:

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

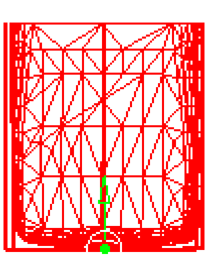
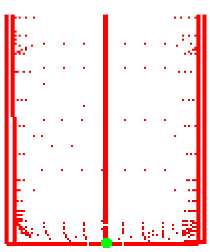
- **Minimal Point Quality** value is 75:

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.

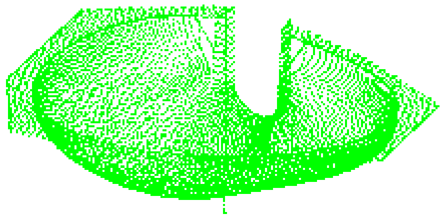
Free Edges is used to create or not the scans representing the free edges of a mesh:

 or 

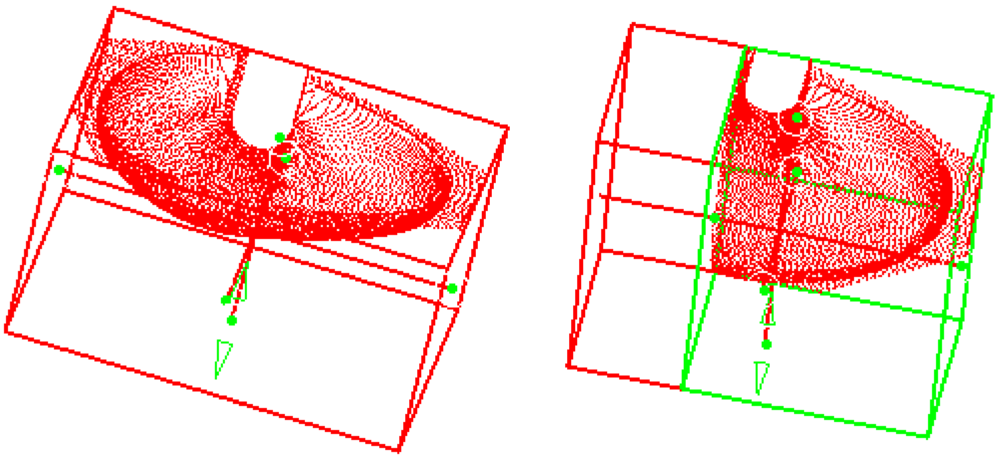
Facets is used to create or not the facets of the imported mesh:

 or 

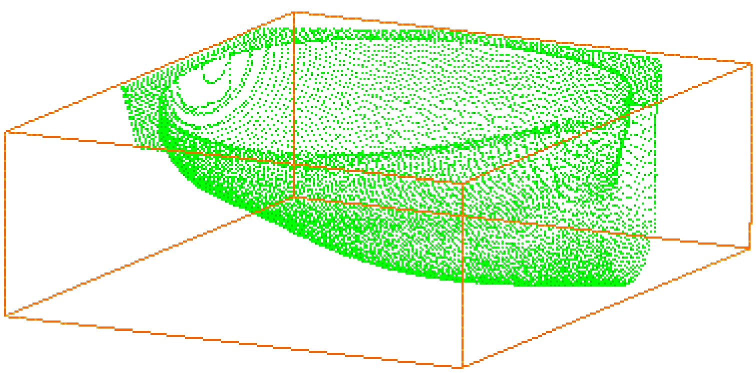
7. Click **Apply** to display the cloud of points or mesh:

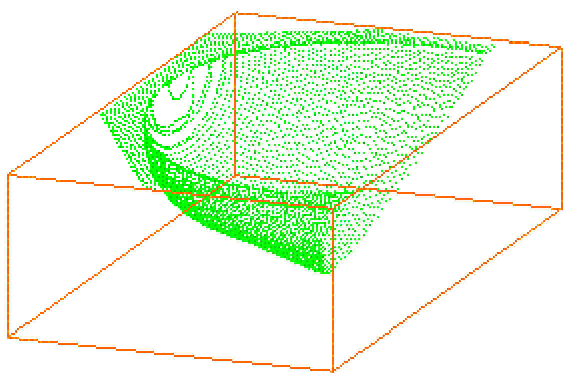


8. Push the button **Update** to display the bounding box of the cloud of points or mesh. Use the green arrow to resize it in order to import only a part of the cloud of points or mesh.



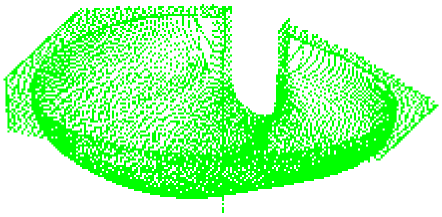
- The bounding box appears every time the cursor passes over a cloud of points or a mesh. Its size corresponds to that of the visible points.
- If a local axis system is set as current, the file will imported in this axis systems and not in the absolute axis system as previously. If no local axis system 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.



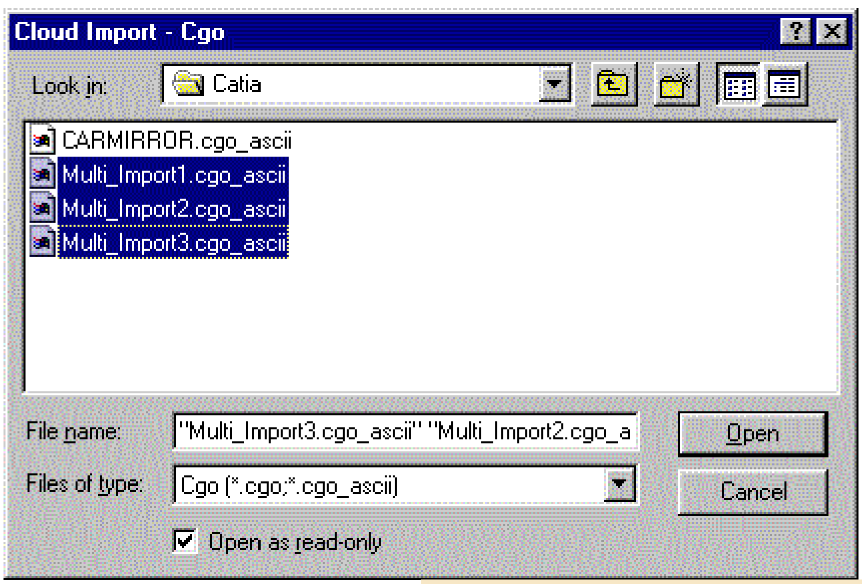
- The name of the element created in the specification tree is the name of the original file, without its extension.
- Undo and Redo are available.
- V5 memorizes the data of the last imported file and proposes them at the next import action.

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.



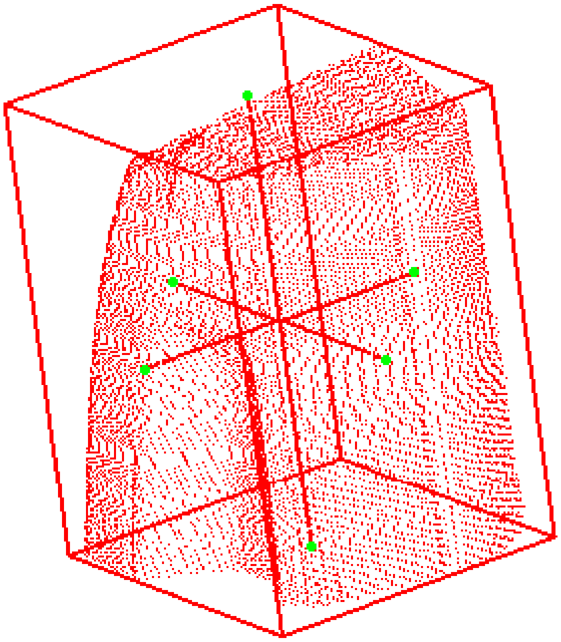
The **Selected File** field looks like this:

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

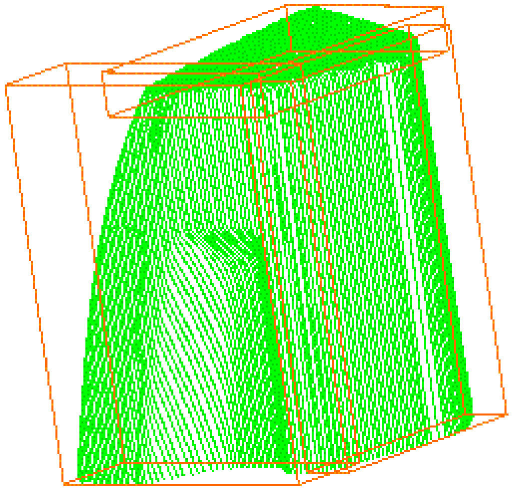
All the files are imported as one single cloud of points instead of several:

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

The files are imported separately.



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.



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.

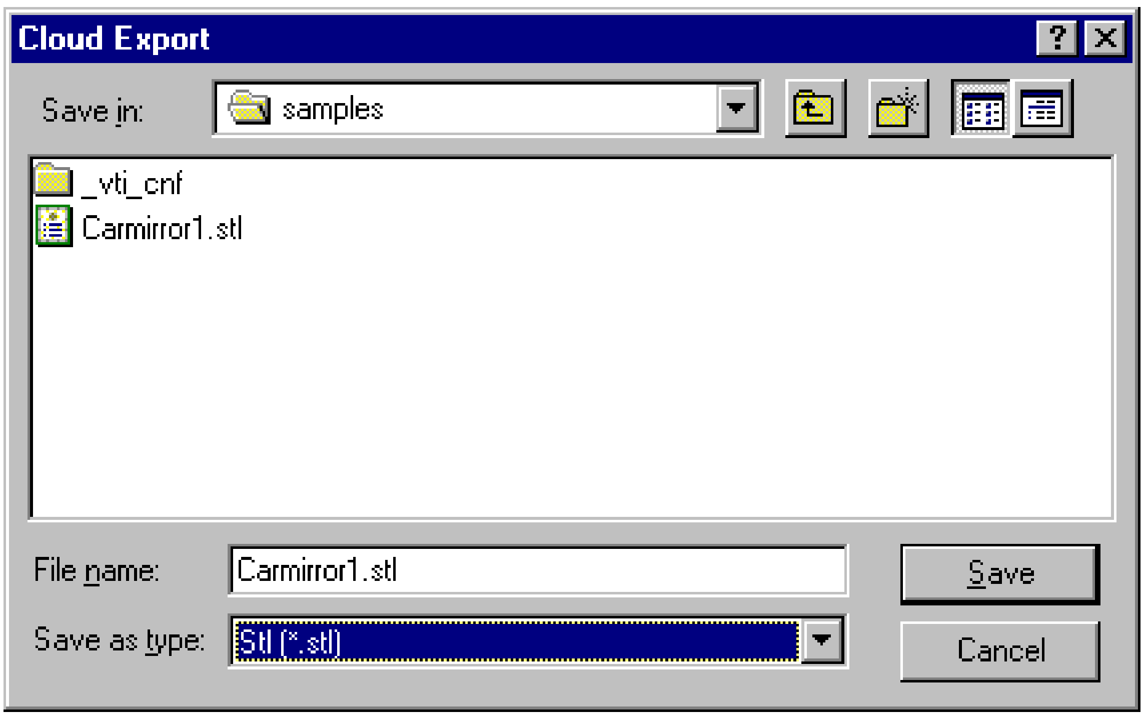
Other formats available are:

- ASCII,
- cgo.

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.

Editing Meshes

This chapter deals with the edition of meshes, i.e. **Selection**, **Remove** actions.

Although the dialog boxes look similar, the operating mode of the **Select** and **Remove** actions are slightly different:

- De-activated triangles can be recalled using **Activate all** and **Invert** in a new activation action.
- Removed triangles can not be recalled! **Activate all** and **Invert** apply only to the current removal action. They can not be used to recall removed triangles, once you have clicked **OK**.
- The **Activation** action displays only triangles that are fully selected (i.e. the whole triangle is inside the selection trap, or all its vertices have been picked). If you select only one or two vertices of a triangle, or if the selection trap intersects the triangle, it is not displayed.
- As a consequence, when you push the button **Swap**, the triangles displayed are not the exact complement of the previous selection.
- The **Remove** action takes into account triangles that are at least partially selected (at least one vertex has been picked, or the selection trap intersects the triangle).

Activating a Portion of a Cloud
Remove

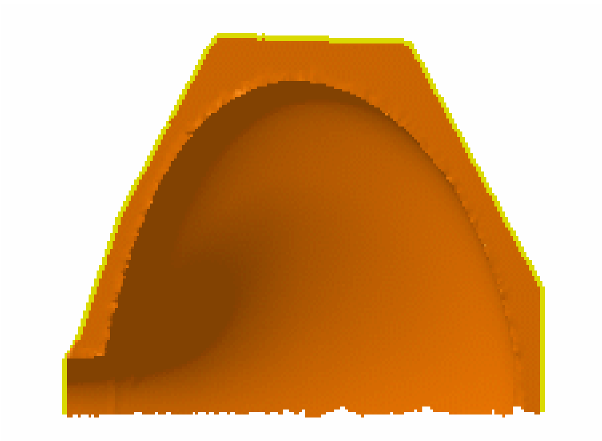
Activating a Portion of a Cloud of Points

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

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

The free edges displayed are those of the complete mesh:

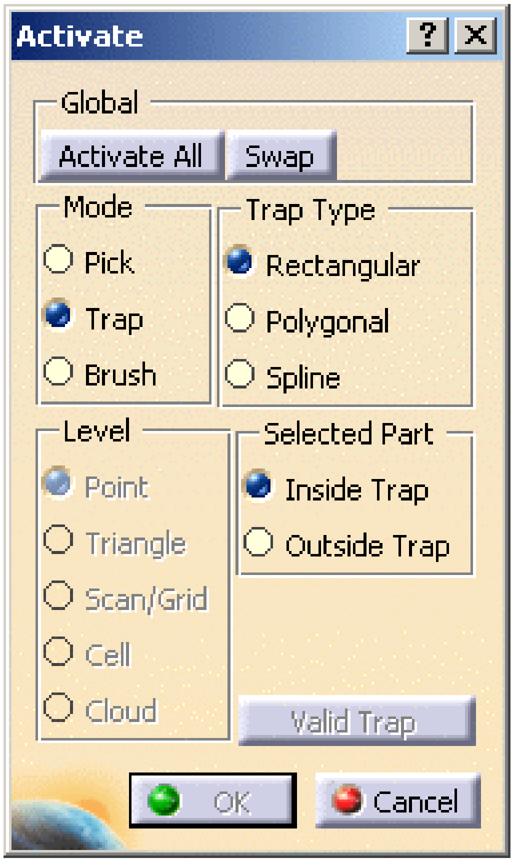
- if you activate only a portion of a mesh, the free edges of that portion are not displayed.



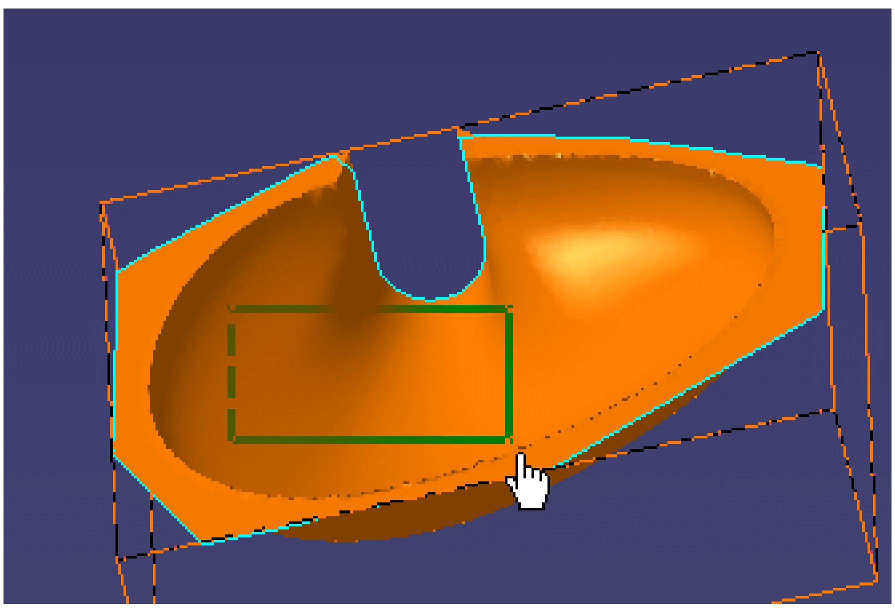
Open the Cloud.CATPart model the from the samples directory.

Click here for more information on the dialog box.

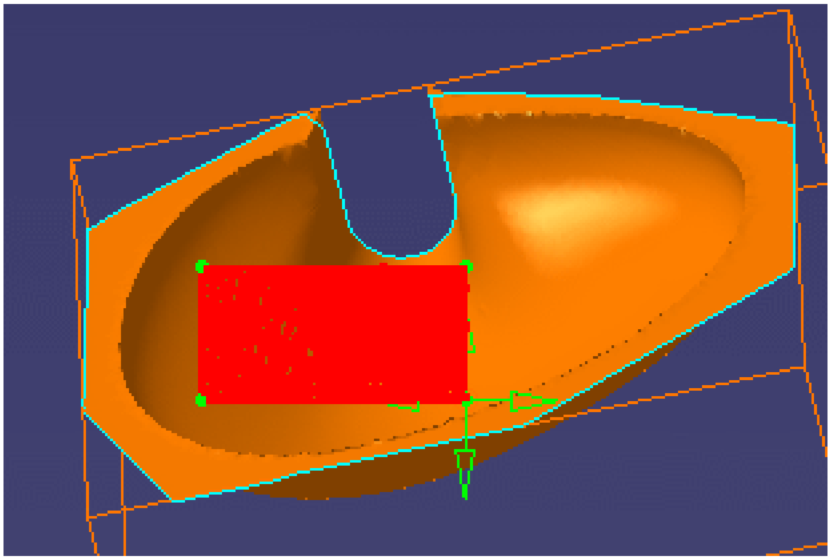
1. Click the **Activate Areas** icon and the mesh. The **Activate** dialog box is displayed.



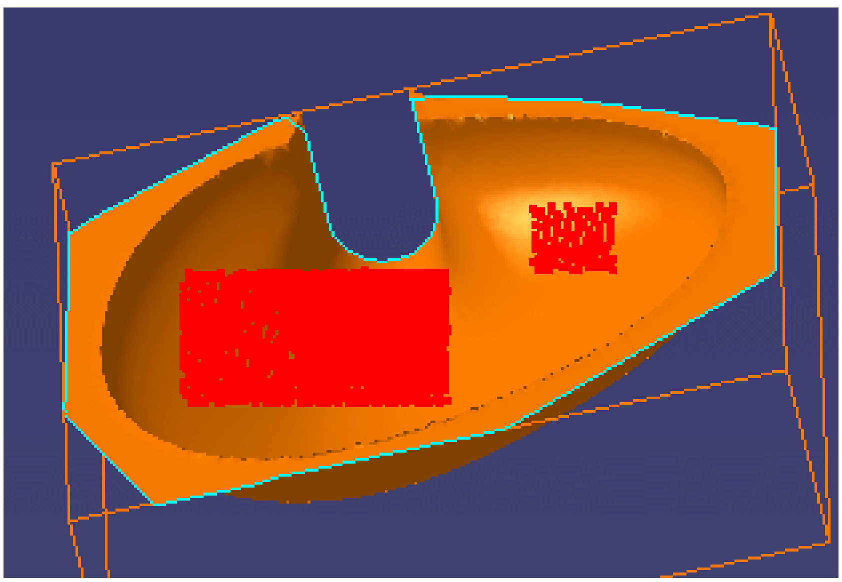
2. Draw a rectangle by dragging the mouse over the portion you want to select.



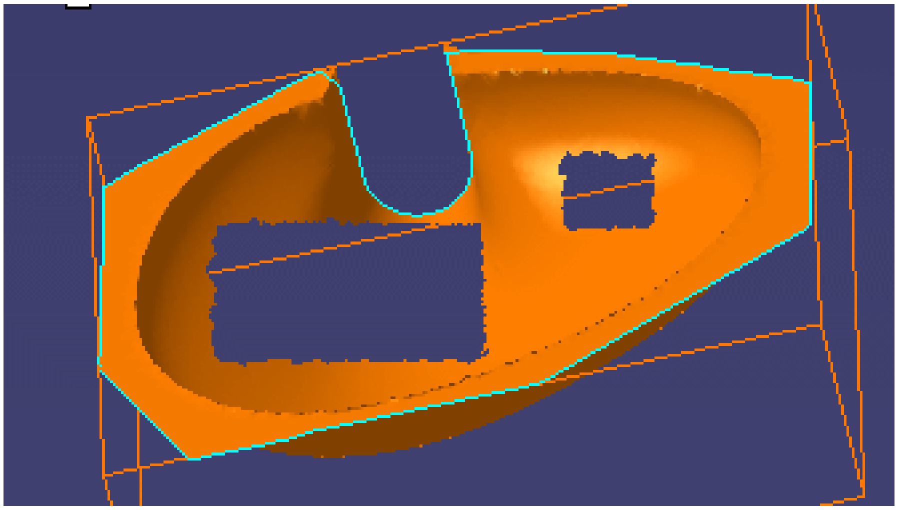
As you release the mouse, the triangles selected are highlighted in red. When you move the mouse over one corner of the rectangle, dimensioning arrows are displayed, enabling you to resize the rectangle.



3. Push the **Valid Trap** button that is now available and draw a second rectangle. Push **Valid Trap** again.



4. Push the Swap button. The selection is inverted.



5. Click OK to validate and exit the action.

Removing Elements

This task shows you how to remove a elements from a cloud or a mesh.

- The deleted elements are those that appear in red during selection.
- By default, the trap is displayed in the view plane (2D trap). You can rotate the model to display the trap as a 3D trap.
- Within one removal action, use **Activate all** to recall all the points of the original cloud of points, or **Swap** to invert the selection (the complement of the current selection becomes active whereas the current selection is hidden).
- The **Remove** action takes into account triangles that are at least partially selected (at least one vertex has been picked, or the selection trap intersects the triangle).

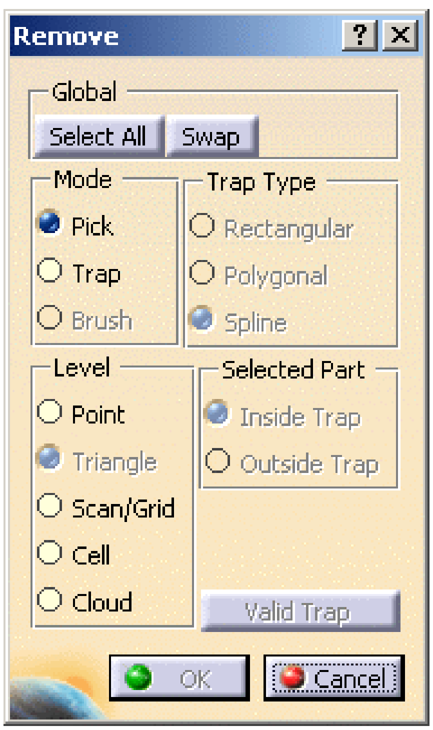
- Undo/Redo are not available on the global action.
- Although the dialog boxes look similar, the operation mode of the Select and the Remove actions are slightly different:

  - Removed elements can not be recalled !
  - **Activate all** and **Swap** apply only to the current removal action. They can not be used to recall removed elements once you have validated the action.
  - All free edges may be displayed.

Open the Cloud.CATPart model from the samples directory.

Click here for more information on the dialog box.

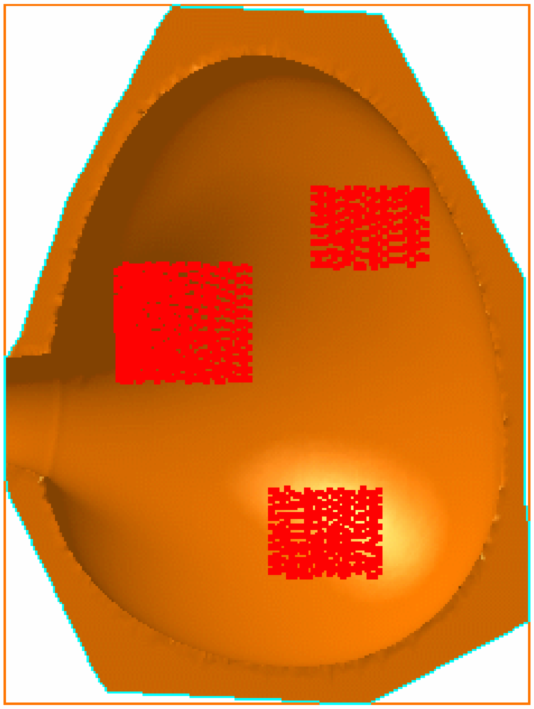
1. Click the **Remove** icon and the mesh. The **Remove** dialog box is displayed.



2. Check the **Pick** option, then the required element type (**Level**) to remove elements using the hierarchical selection.

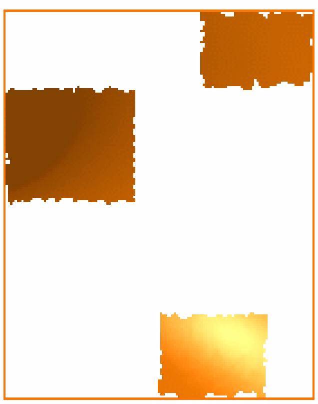
   According to your choice and the application you are working in, only points, or triangles, or scan/grid, or cells (subcloud) or clouds (global cloud) will be removed. Select the unwanted elements on the cloud, then Click OK to confirm their removal and close the dialog box.

Or check the **Trap** option then the required **Trap Type** and the portion of cloud to remove (**Inside** or **Outside Trap**) to remove elements using a graphical trap. You can draw either one single trap, or several traps.



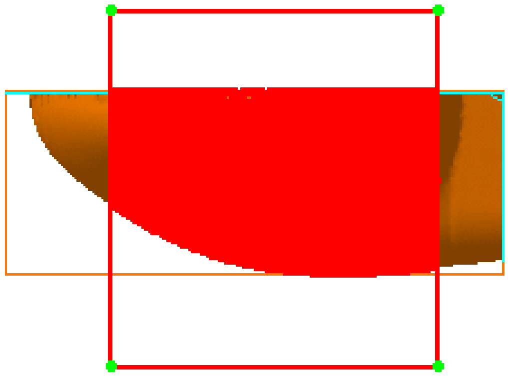
In that case, valid each trap with the **Valid Trap** push button before drawing the next one.

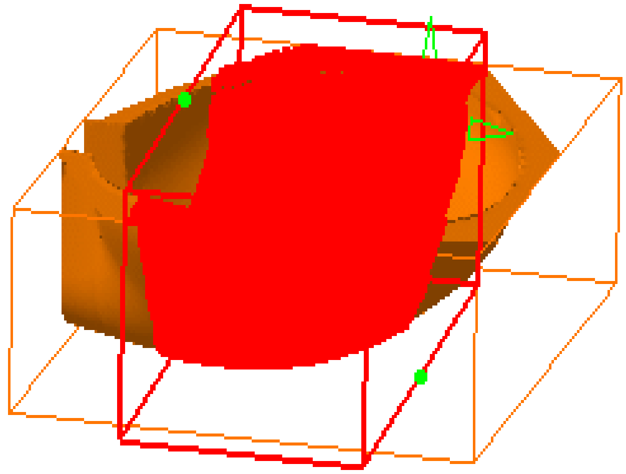
If you draw a trap, push **Valid Trap**, then **Swap**, you will remove the complement of the original selection.



Click OK when all traps have been defined to remove the unwanted elements and close the dialog box.

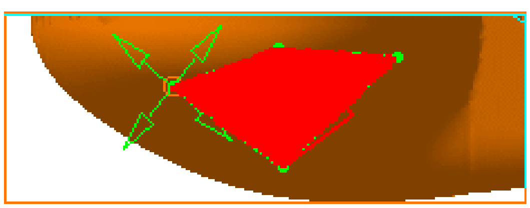
The traps may be either rectangular :

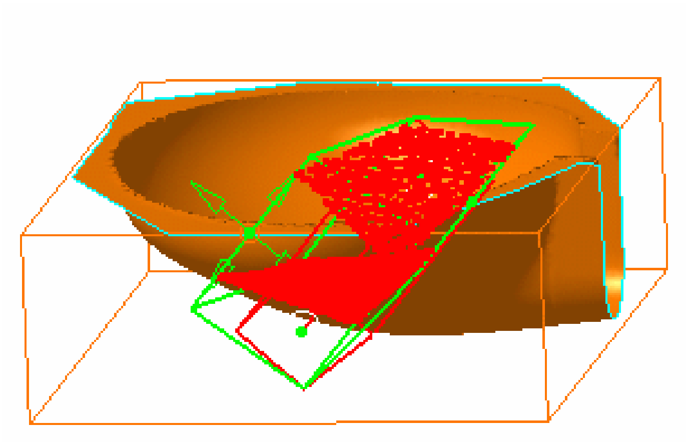




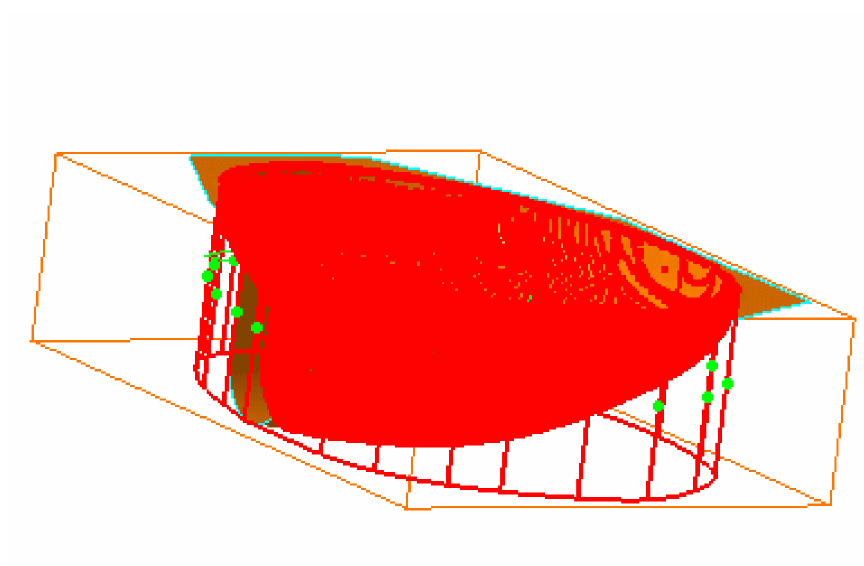
In that case, you can modify the trap using the green manipulators.

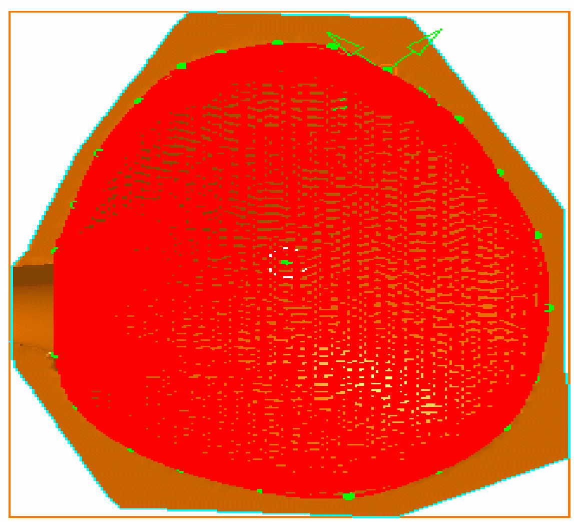
or polygonal :





or spline:





In those cases, you can modify the trap using the green manipulators or use the Undo action on each segment of the trap as long as you have not double-clicked to end the polygonal trap.

Mesh

This chapter deals with the tessellation of clouds of points.

Mesh Creation

P2

This task shows how to mesh a cloud of points or regenerate an existing mesh.

The **Mesh Creation** and the **Mesh Regeneration** actions offer:

- a neighborhood parameter that makes it possible to fill holes or to let some areas unmeshed,
- an automatic detection or a manual definition of the meshing plane for the 2D mesher,
- boundary continuity with contiguous meshes through the **Constrained** option,
- a fully automatic 3D meshing **(c)INRIA**, suitable for mechanical parts with blind or through holes, that respects details, especially sharp edges,
- a sag value to mesh dense clouds with a reduced number of triangles, but still respecting the 3D Shape within a given tolerance.

In STL Rapid Prototyping, **Mesh Regeneration** is allowed on meshes only, i.e. objects whose partially or totally active cells are meshes.

If this is not the case, a fatal error panel is displayed: **Mesh Regeneration is allowed on meshes only.**

Open the Cloud.CATPart model from the samples directory.

1. Click the **Mesh Creation** or **Mesh Regeneration** icon. The **Mesh Creation** dialog box is displayed.

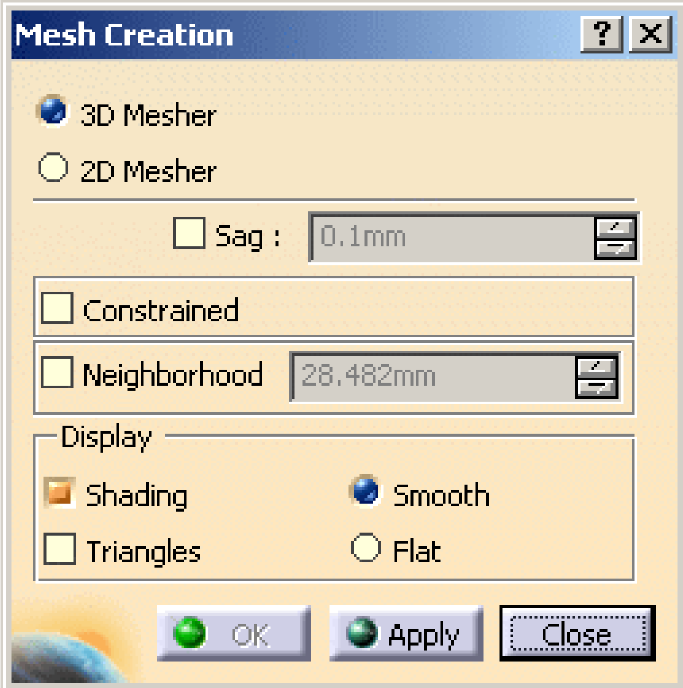
2. Check the **Execution Mode** option you need:

- **3D (c)INRIA:** this is a meshing method for complicated shapes (e.g. mechanical objects, clouds that can not be projected onto a single plane, or without draft characteristics).

This is the default option.

- **2D:** this is a less complex meshing method, to apply to simple objects, i.e. that can be projected onto a single plane (smooth shapes).

The entry dialog box is replaced with that of the 2D Mesher.

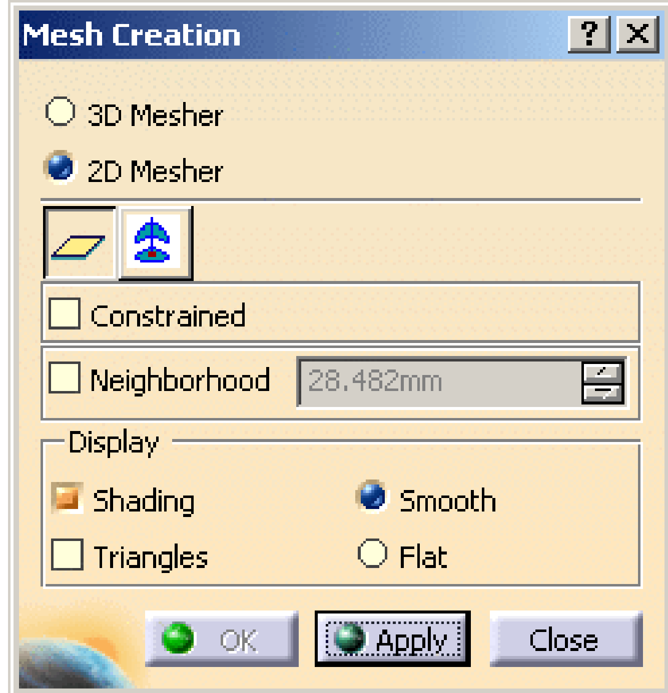


This mesher provides a **Sag** option to reduce the number of triangles computed on dense clouds. However, this option respects the shape of the object.

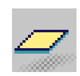
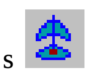
You would achieve the same result by filtering the cloud with the adaptative option set to the sag value and meshing the output.

You can also mesh a cloud with a sag equal to 0. This means that all the points are meshed. This algorithm is more suitable to mesh large clouds quickly.

3. If necessary check the sag option and enter its value, or keep the default 3D Mesher.

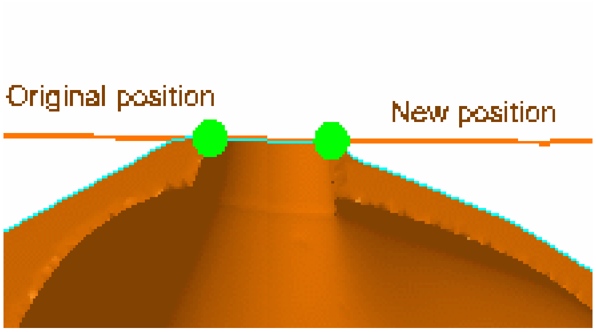


3. Select the plane that is the computation reference for the meshing:

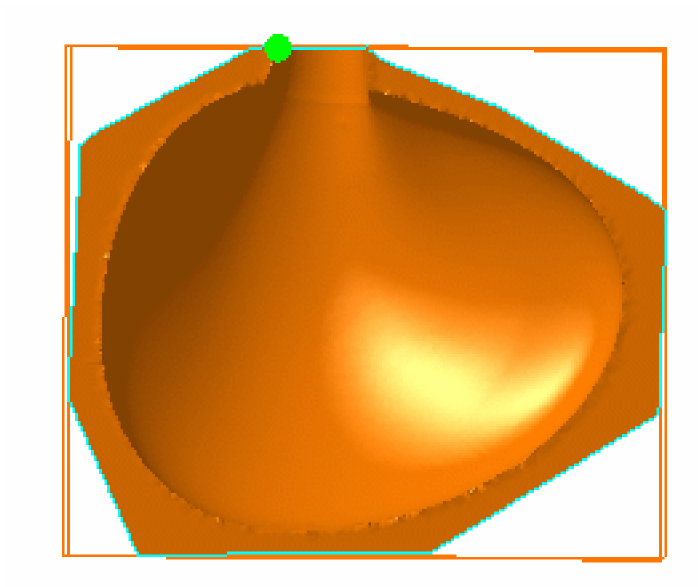
- either one main plane 
- or one defined with the compass 

The quality of the mesh depends on the computation direction.

4. A **Neighborhood** value is proposed in accordance with the model. This value represents the maximal edge length of the triangles displayed. The value proposed is just an approximation of this value. Its relevance depends on the distribution of the points in the cloud. It is visualized by a sphere. You can change its position by a simple mouse click.



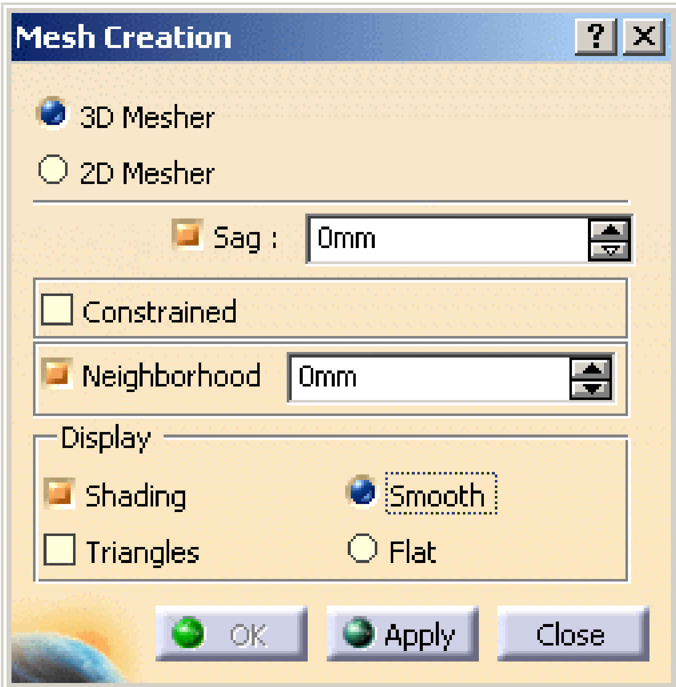
The sphere is updated when you change the **Neighborhood** value.



5. Check the **Display** option you need:

- **Triangles** to display only the mesh,
- **Shading** to simulate the surface of the object:
  - with the **Flat** option, the light is sent on the triangles along their normal,
  - with the **Smooth** option, the light is smoothed over the triangles, giving a better image of the quality of the surface.

- Meshing requires a complex computation. The computation time will increase according to the size and complexity of your model. You may want to filter the cloud before starting the meshing.
- To mesh large quantities of points, we recommend the following settings:
  - **Sag**=0mm
  - **Triangles** not activated
  - **Shading** activated, with the **Smooth** option.



These display options are applied within this action only. Once you have validated the result with OK, the result is displayed in the Smooth mode, even if the input element or the computed mesh were displayed in another mode.

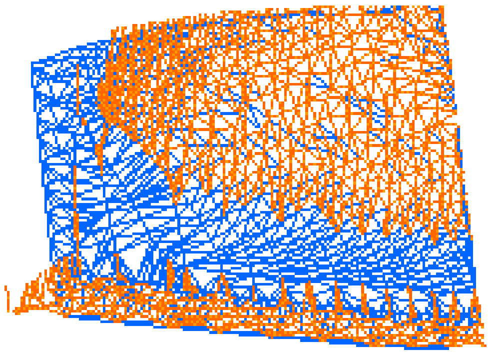
6. **Constrained** is used to:

- re-process a portion of a mesh by adding points to an existing mesh or reprocessing a meshed cloud that has been unfiltered::

Open the Mesh2.CATPart model from the from the Samples directory.

The original mesh had holes in it. Select a faulty portion and proceed to a new meshing on that portion.

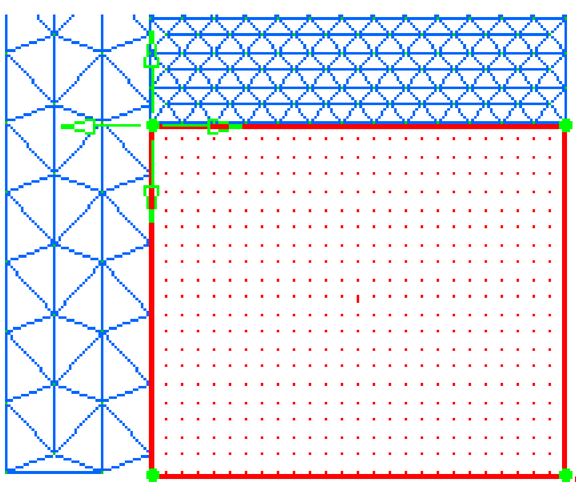
The original mesh is in red. The re-processed mesh is in blue.



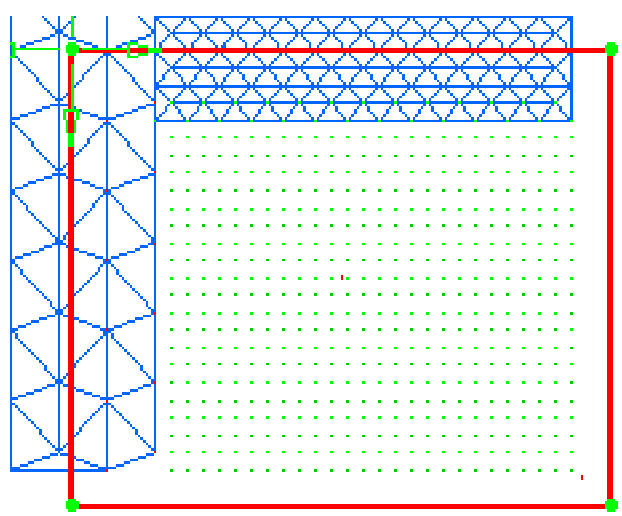
- connect several meshes:
  Open the Mesh1.CATPart model from the Samples directory. Draw two traps on that part and mesh them with different values, as shown below. Now activate the remaining square of points, as shown below.

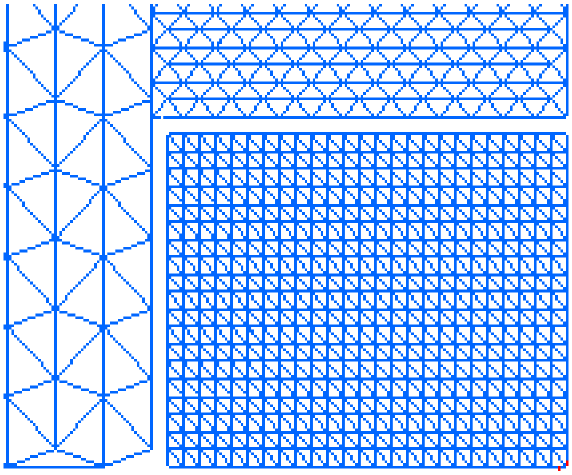
For quicker meshing performances, you can filter portions of the parts according to your successive needs.

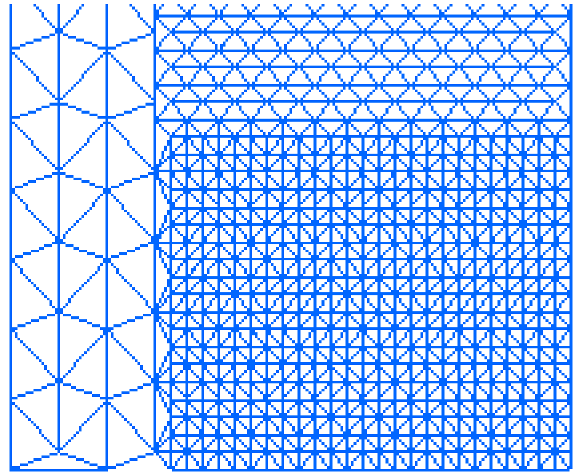
The mesh is unconstrained, the activation trap does not overlap the previous meshes. The resulting mesh is independent from the other two.



The mesh is constrained, the activation trap overlaps the previous meshes. The resulting mesh is connected to the other two. Facets of the existing mesh that were totally or partially inside the trap have been removed and recomputed to adjust to the additional mesh.

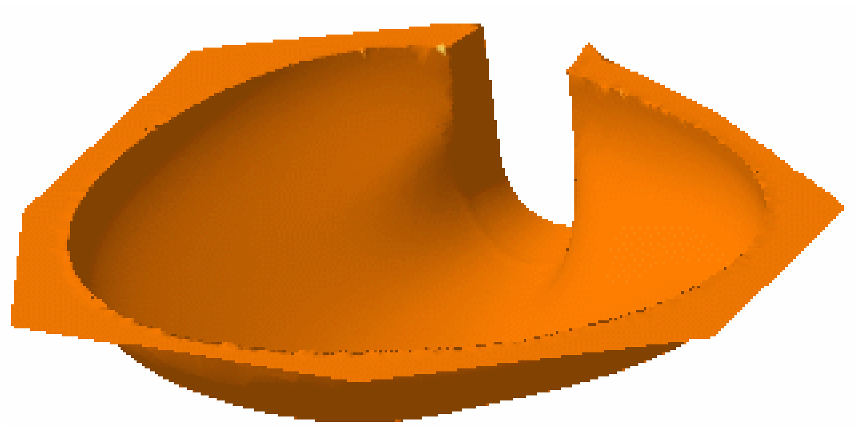






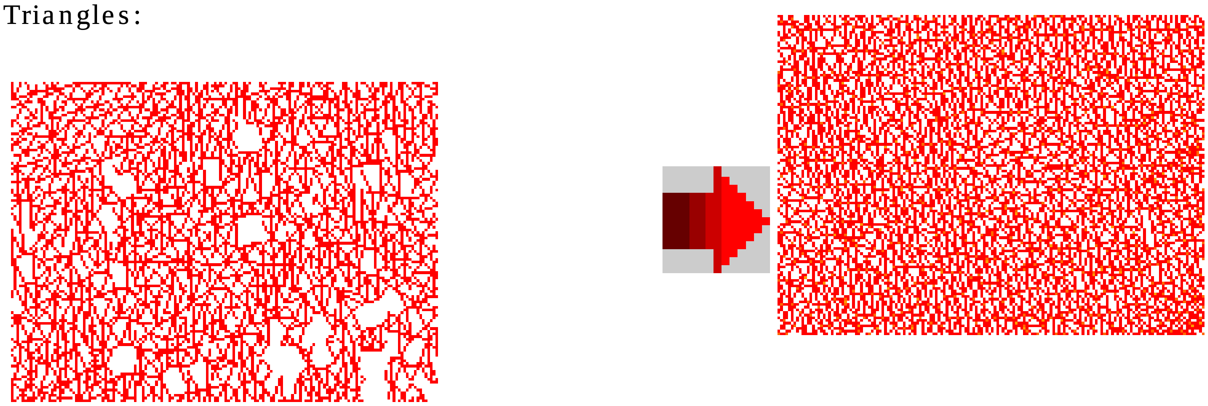
7. Click **Apply** to check or update the result. Then click **OK** to confirm the result and exit the action.

An element **Mesh Creation.x** is created in the specification tree.

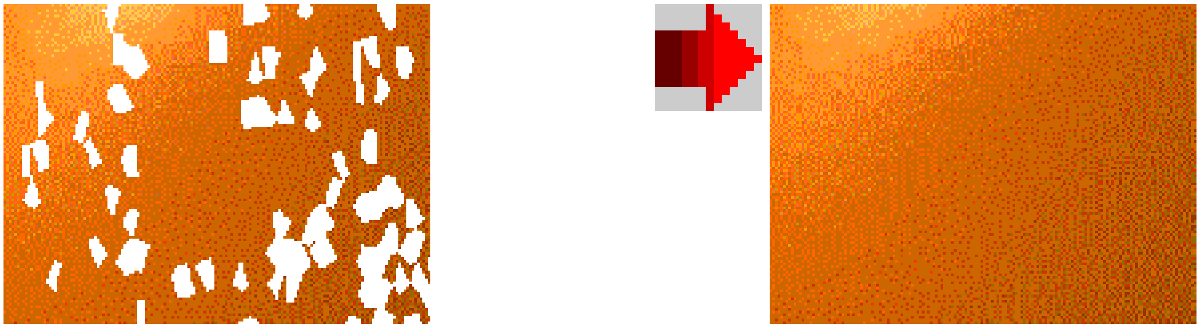


Increase the **Neighborhood** value to improve the mesh or reduce it to avoid filling holes that should remain clear:

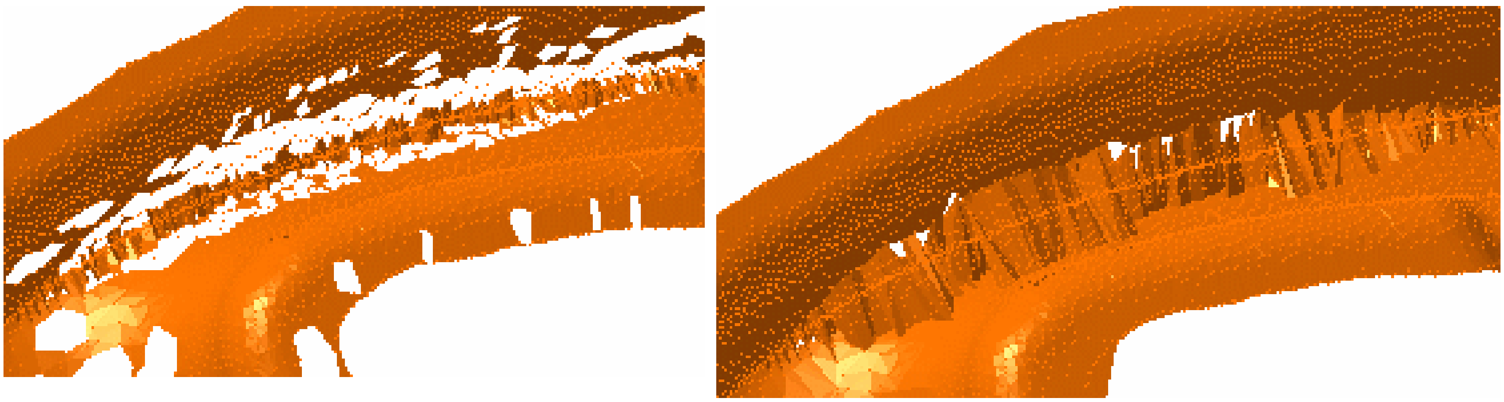
Triangles:



Shading:



- In some cases, it may be difficult to find a **Neighborhood** value that will fill unwanted holes, without creating unwanted triangles:



- Seams may appear on the mesh with the **Smooth** option, :
  - They indicate that the normals to the facets have different directions at this place.
  - In **2D** and **3D** mode, some triangles may overlap and mesh should be corrected.
  - In **Constrained** mode, they show the common boundaries of meshes.

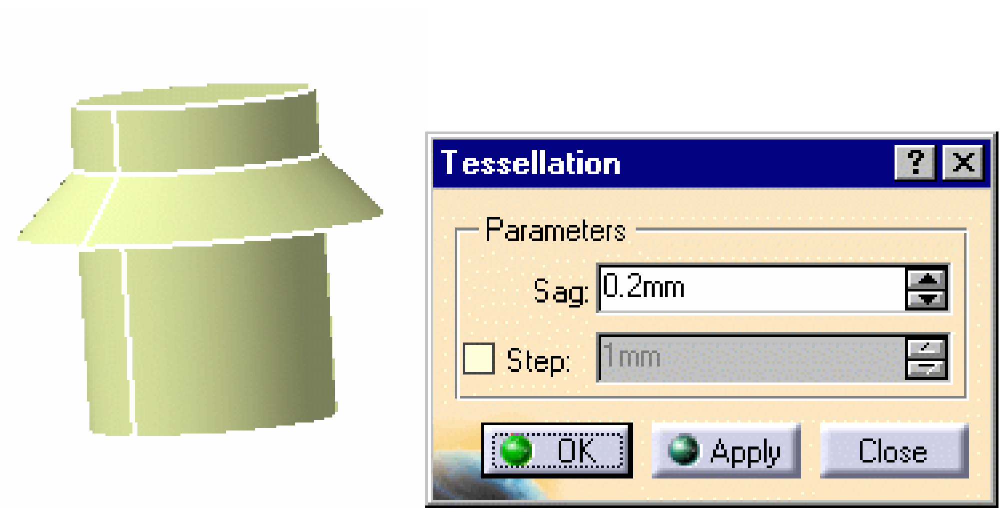
- When computing a constrained mesh, enter 0 as the Neighborhood value to check the boundaries of the mesh. If the boundaries are not satisfactory, modify the mesh plane to improve them.
- After the computation of a constrained mesh, two mesh elements are visible in the specification tree: the constrained mesh and the initial mesh. You can select one and then the other to make sure they are complementary.
- You can use the Meshes Merge action to obtain a single mesh.

Tessellating an Object

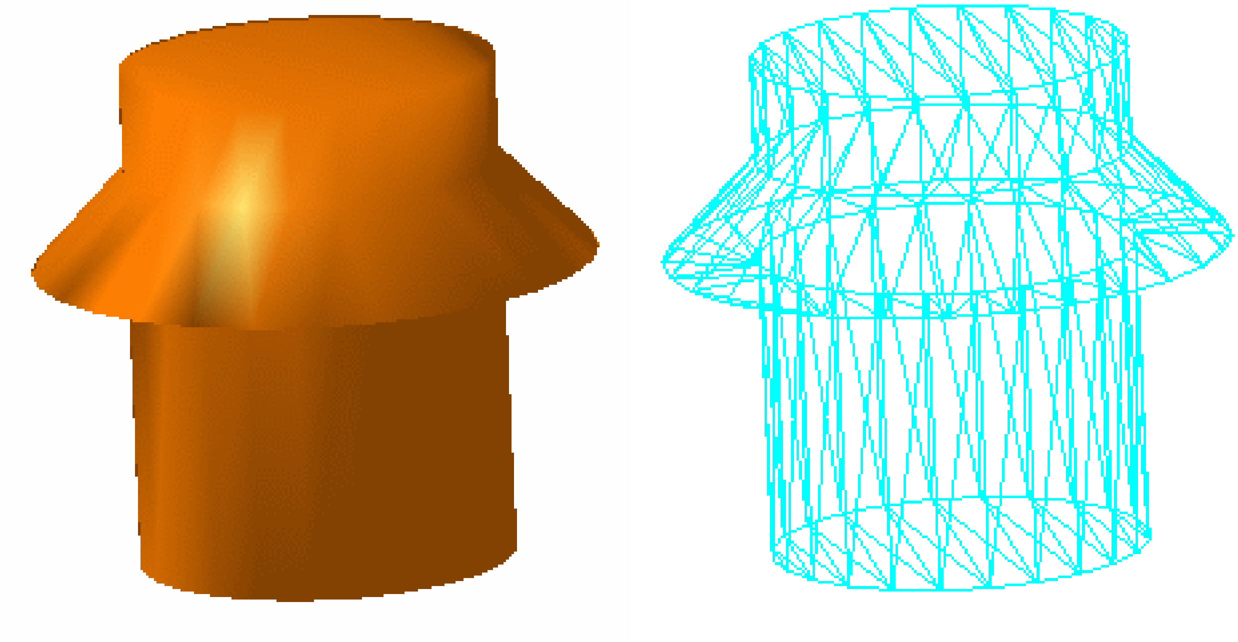
This task shows how to create a tessellation from a surface or a solid. Inside this action, you may specify the maximum size of the triangles and the maximum distance (sag) between the geometry and the triangles.

Open the SurfTessellation1.CATPart model from the samples directory.

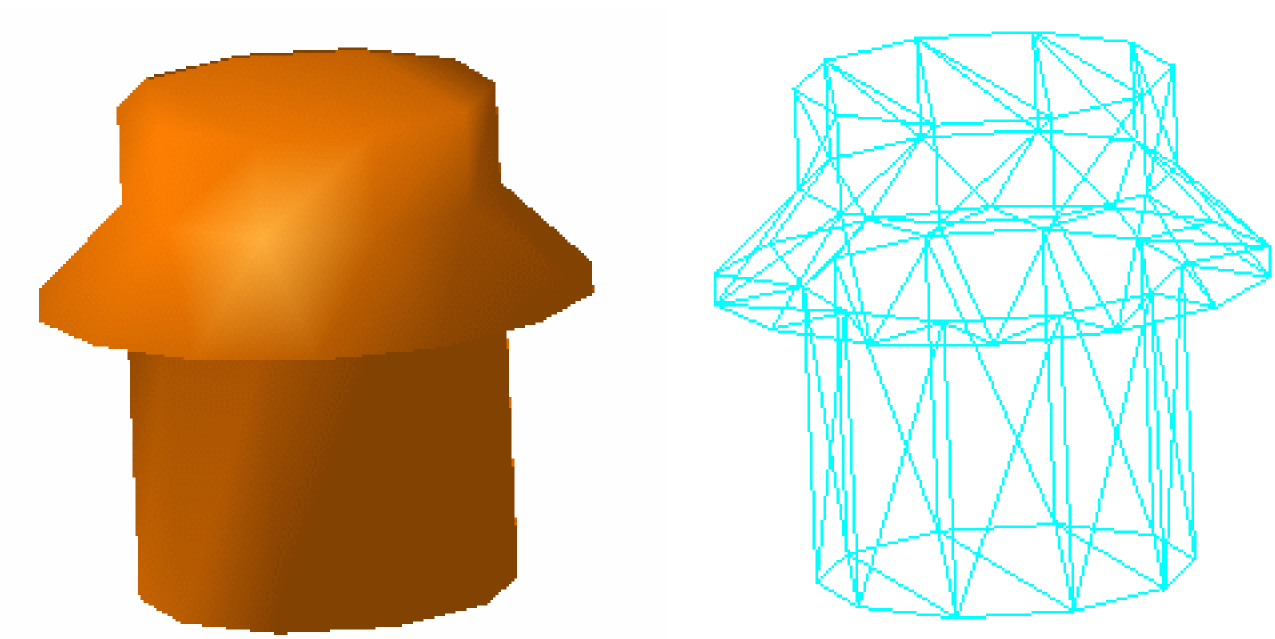
1. Click the **Surface Tessellation** icon . The **Tessellation** dialog box is displayed.



2. Select the surface or the solid to tessellate. Clik **Apply**.

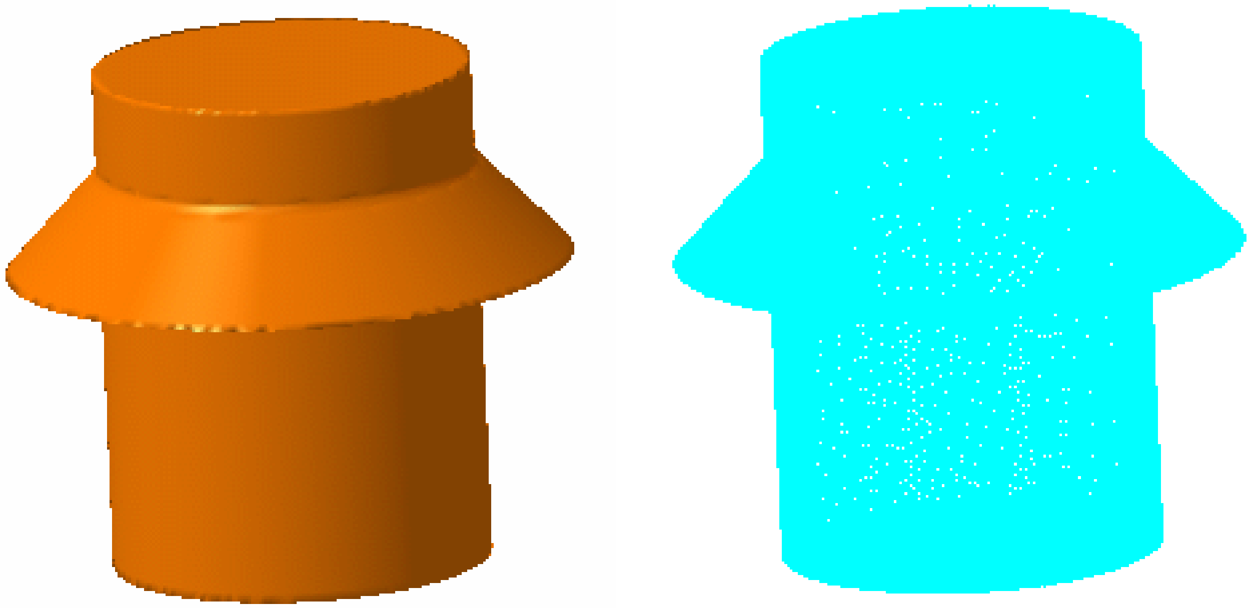


3. You can modify the **Sag** value, that is the maximum distance between the geometry and the triangles:



it has been increased to 1

4. You can also check the **Step** box to control the length of triangles:



5. Click OK to validate. A **Tesselation.x** element is created in the specification tree.

- The free edges of the tessellation are those of the surfaces or solids.
- To avoid free edges between the tessellations of several contiguous faces, you can join them with the **Join** action of Quick Surface Reconstruction or Generative Shape Design. They will then be processed as a single surface.

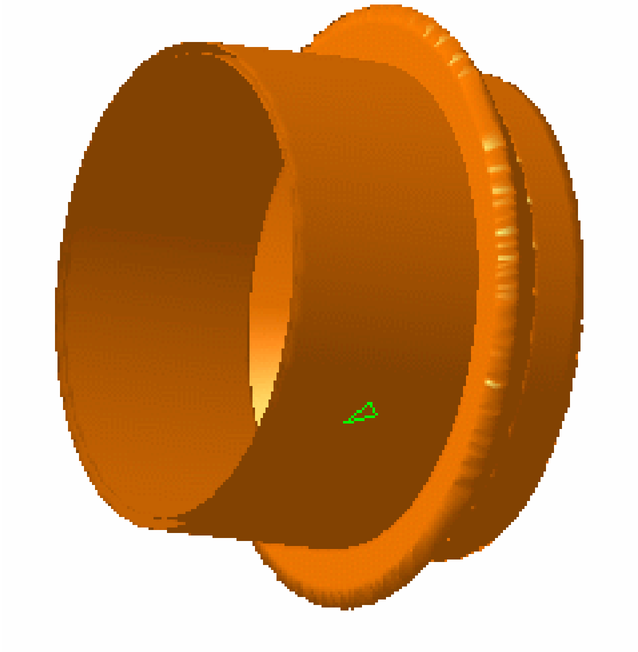
Offsetting a Mesh

This task shows how to offset a mesh and create a watertight mesh.

Open the Offset1.CATPart model from the samples directory.

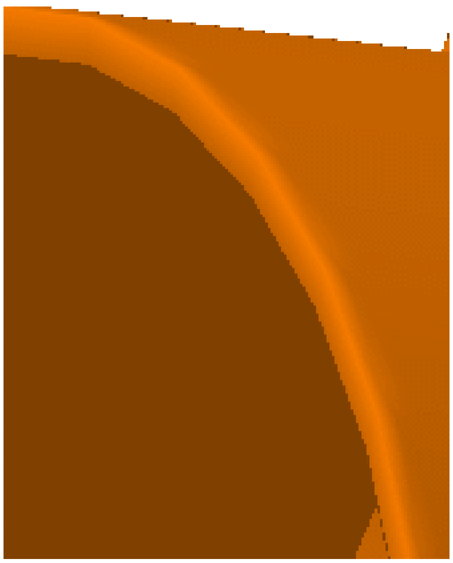
1. Select the **Offset** icon . The **Offset Mesh** dialog box is displayed.

2. Select the mesh.
3. Enter an **Offset value**. The offset mesh is displayed, together with a green vector representing the offset. The direction of the offset is given by the sign of the value.



4. Check the **Create shell** option to create a watertight offset::

Create shell unchecked



Create Shell checked

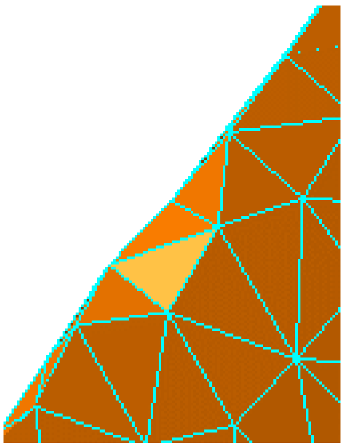
5. Click **Apply** to check or update the result. Then click **OK** to confirm the result and exit the action. The element **Mesh Offset.x** is created in the specification tree.

- The offset is computed in the direction of the weighted normals of the points.
- For better results, you should avoid to enter a high offset value.

Flip Edges

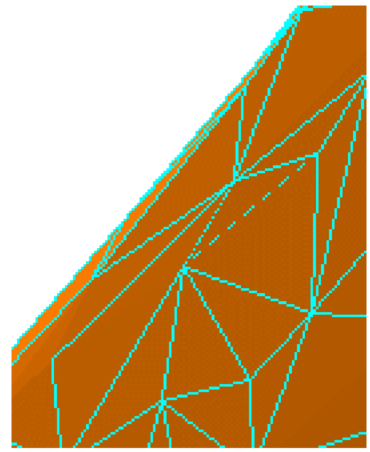
This task shows you how to flip edges of triangles of a mesh, for a better respect of sharp edges, by rotating the triangles common edges without modifying their vertices. The meshing may become less harmonious but will provide a better respect of the shape of the part because the triangles will be oriented in the direction of the shape, in particular for sharp edge fillets. This is particularly important for milling operations that may follow.

This action reorganizes the meshing without modifying the geometry because the vertices are not recomputed.



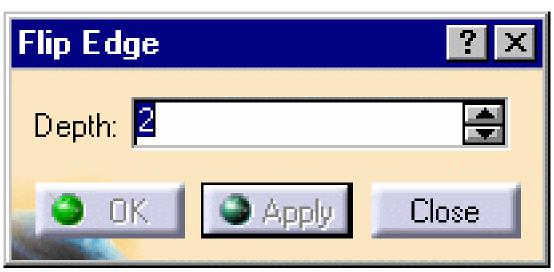
The first picture shows a blunted edge.

The second picture shows the same edge after the reorientation of the triangles.



Open the Cloud.CATPart model the from the samples directory. For a better understanding, use the **Cloud Display Options** icon to display the triangles of the mesh.

1. Click the **Flip Edges** icon and a mesh. The dialog box is displayed.



2. Sets the value of **Depth**, that determines the amplitude of the reorganization of the mesh:

- The value of **Depth** ranges from 0 to 10.
- The default value is 2.
- When the value of **Depth** is 0, the action processes a triangle and its direct neighbors.

- When the value of **Depth** is 1, the action processes a triangle, its direct neighbors and their direct neighbors, and so on as you increase the value of Depth.
- This may lead to a temporary degradation of the energy function of the mesh, but results in a final optimal solution.
- The computation time depends on the value of **Depth**, and on the size of the mesh to process.

3. Click **Apply** to start the first reorganization iteration. Click Apply again to start another iteration. You may repeat this step as many times as you wish.
4. Click OK to validate the result. A **Flip_Edge.x** element is created in the specification tree. The initial mesh is sent to the No Show.

- The new mesh inherits the graphic properties and display mode of the initial mesh.
- During the process, the mesh is displayed in Flat Shading mode.
- This action should not be used on meshes with non-manifold edges.
- Undo/Redo are available.
- You may create several Flip_Edge.x elements in the specification tree, corresponding to various degrees of reorganization of a given mesh. You may then delete any of them, according to your needs.

Smoothing Meshes

This task shows you how to smooth a mesh.

The cloud of points you import in Digitized Shape Editor may be noisy, for various reasons, mainly because of a poor digitalization accuracy on the edges of parts. This noise is found again on the meshes computed from these clouds of points or imported in STL format.

The consequences are:

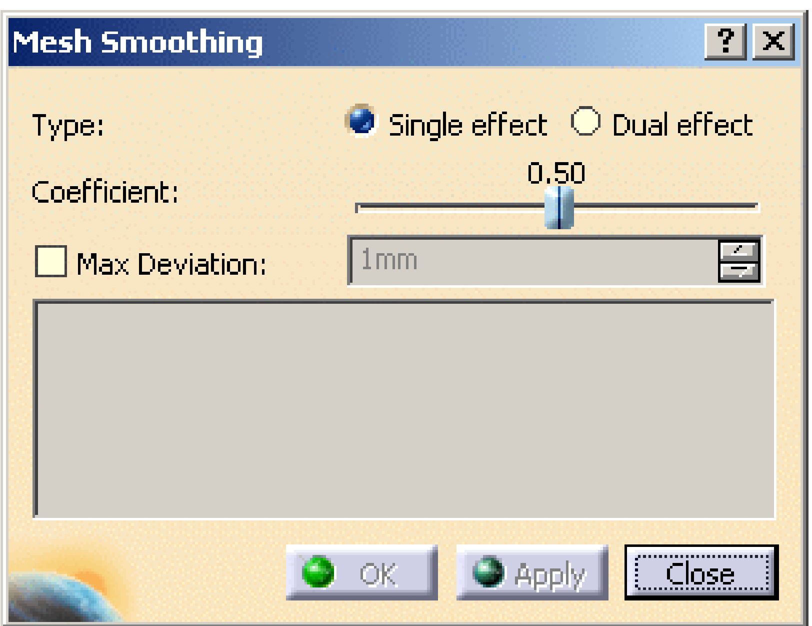
- very noisy scans produced with the **Planar Sections** or Segmentations actions or
- the reconstruction of wavy curves or surfaces and/or of very high order.

This can be partly avoided by smoothing the mesh.

- Since the volume of the part is reduced, some small facets may be inverted by the meshing. Therefore we recommend you alternate **Mesh Smoothing** and **Flip Edges** actions.
- Use the Activate function to process only a portion of a cloud.

Open the SmoothMesh01.CATPart from the samples directory

1. Click the **Mesh Smoothing** icon and a mesh.
2. The **Smoothing** dialog box is displayed.



3. Select the type of smoothing:

Choose **Single effect** if there is no sharp edge on the mesh to process.

- Small radii will be erased.
- The volume of the part will be reduced (contraction towards the center of gravity of the part).

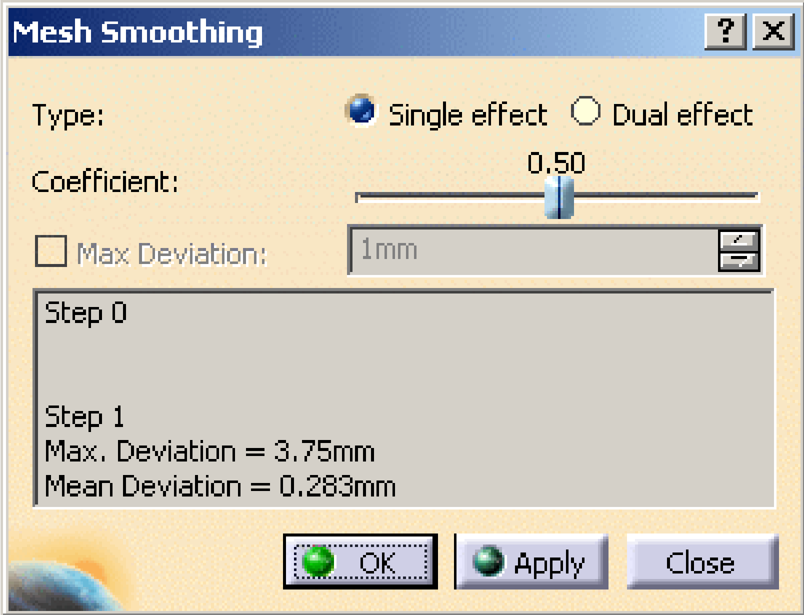
Choose **Dual effect** to reduce the distance between outliers and the surface, and reduce the erasing of small radii.

- The reduction of the volume of the part is smaller.
- A large displacement of one vertex inwards may cause the neighboring vertices to move outwards.

4. Two other controls are available:

- **Coefficient** : It balances the effect of the new theoretical position in comparison with the original position. It varies from 0 (the vertex is not moved) to 1 (the vertex is moved to the computed position).
- **Max Deviation**: Check this option to control the maximum deviation allowed (the displacement will remain under the value set.)

- The deviation is the distance between a vertex and its initial position (not between its current position and that of the previous iteration).
- Therefore, if you want to control the maximum deviation, you have to check the **Max Deviation** option **before** the first **Apply** (it is no longer available after the first Apply).
- For a better appreciation of the quality of the intermediate meshes, the meshes are displayed in Flat Shading within the action.
- In addition, for each step the maximum and the mean deviations (distances between a vertex and its initial position) are displayed in the dialog box.

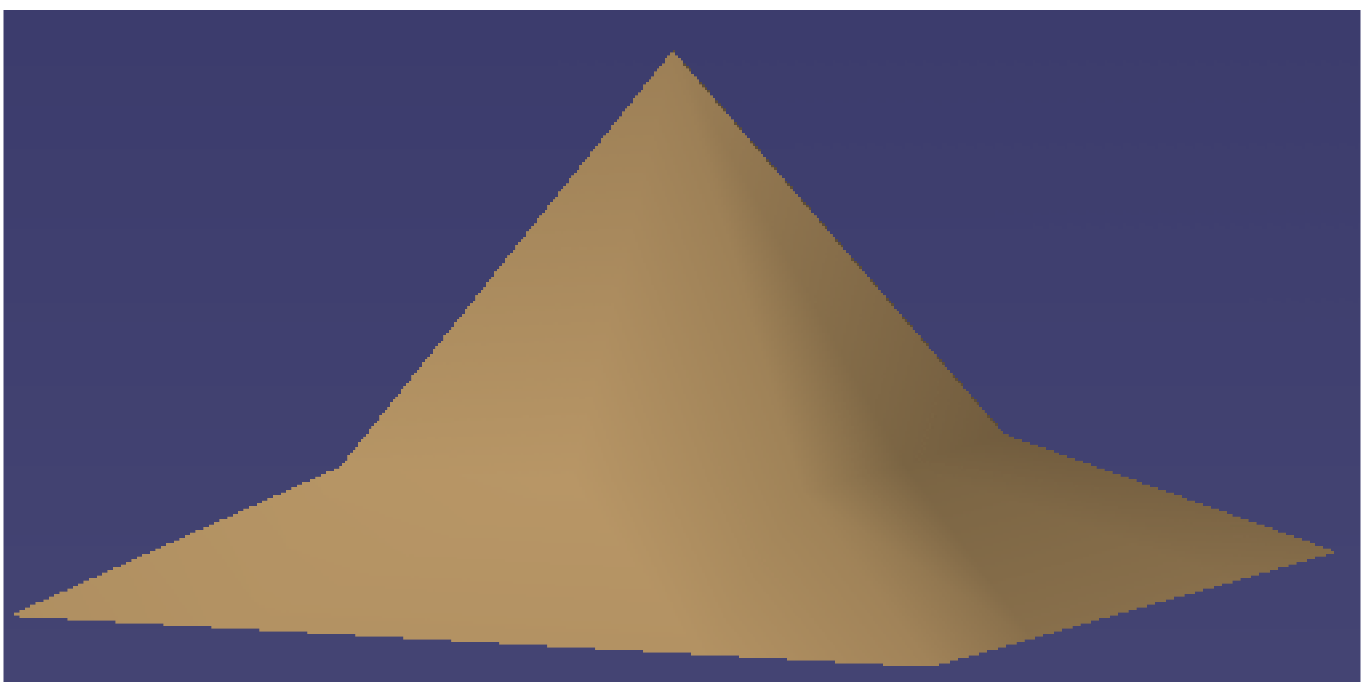


5. Click **Apply**: a new mesh is computed. This action is an iterative one: click **Apply** again to smooth the proposed mesh.

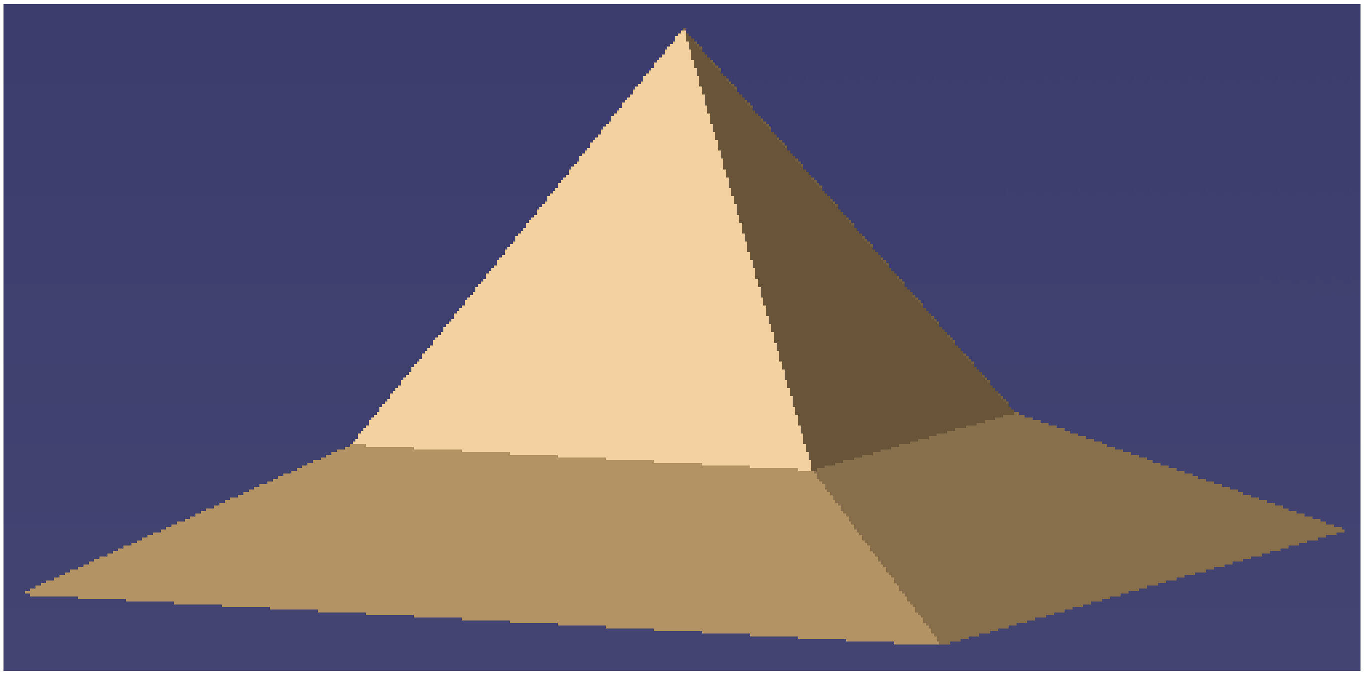
6. Click **OK** once you are satisfied. A **Smoothing.x** element is created in the specification tree, the original mesh is sent to the No Show.

Examples:

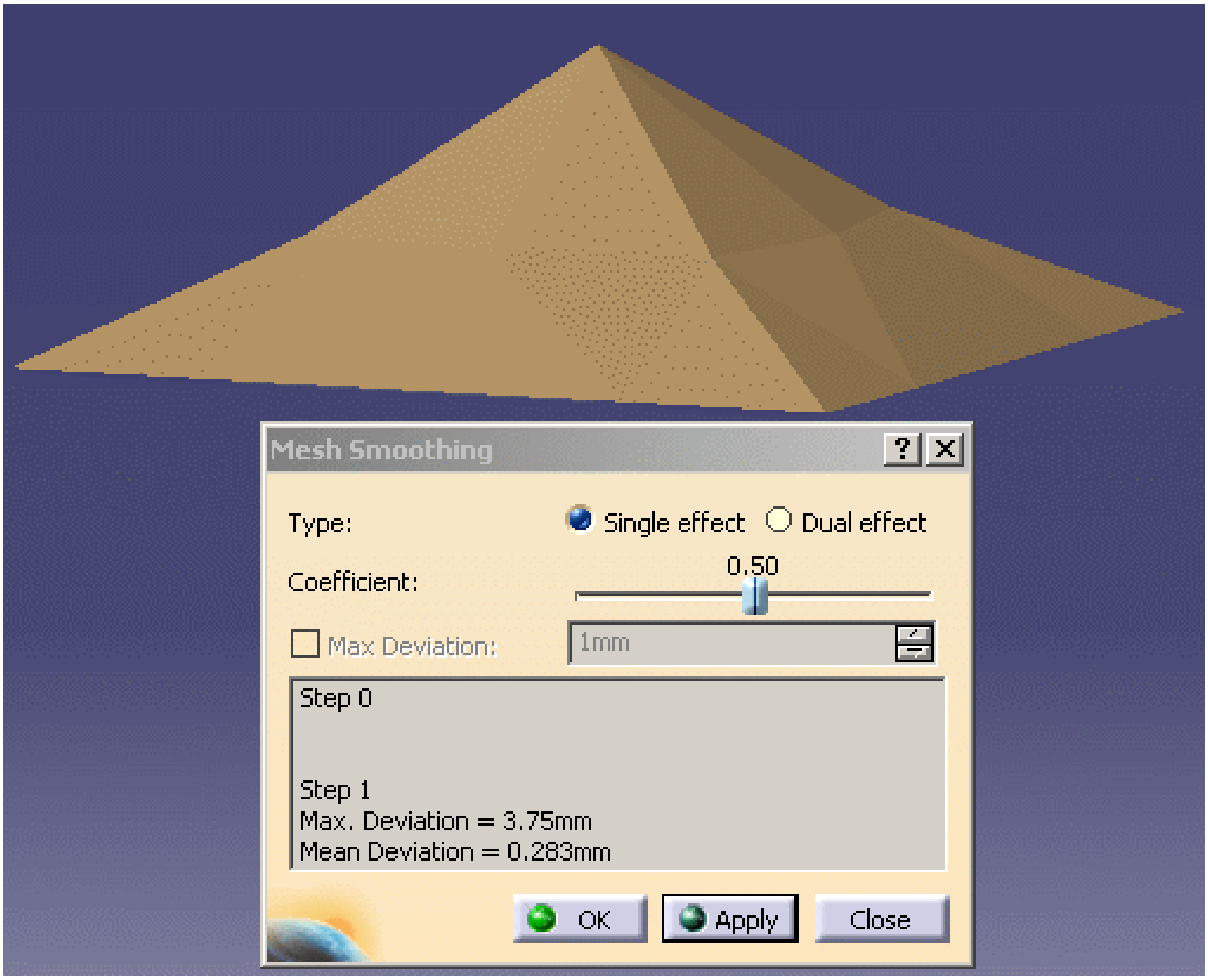
Original part, before entering the action, i.e. in Smooth Shading:



Original part as you enter the action, i.e. in Flat Shading:

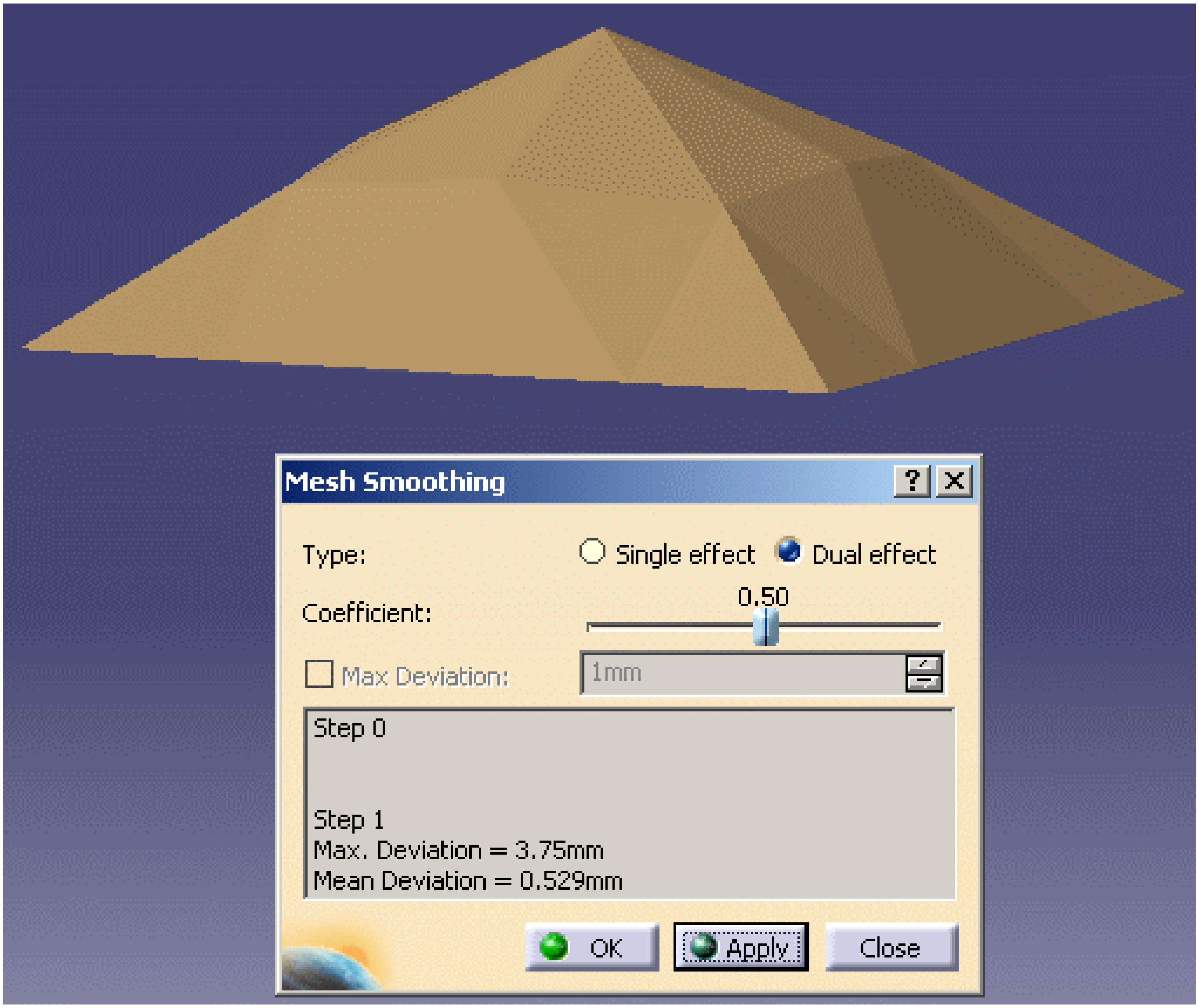


Single effect, in Flat Shading

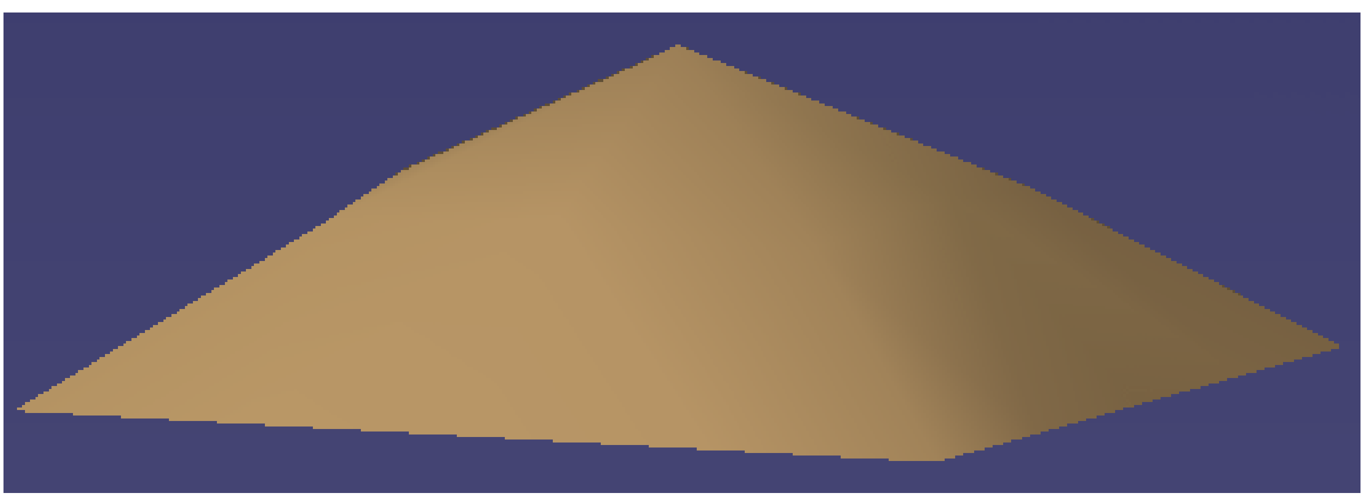


Single effect, in Smooth Shading (after exiting the action)

Dual effect, in Flat Shading



Dual effect, in Smooth Shading (after exiting the action)



Mesh Cleaner

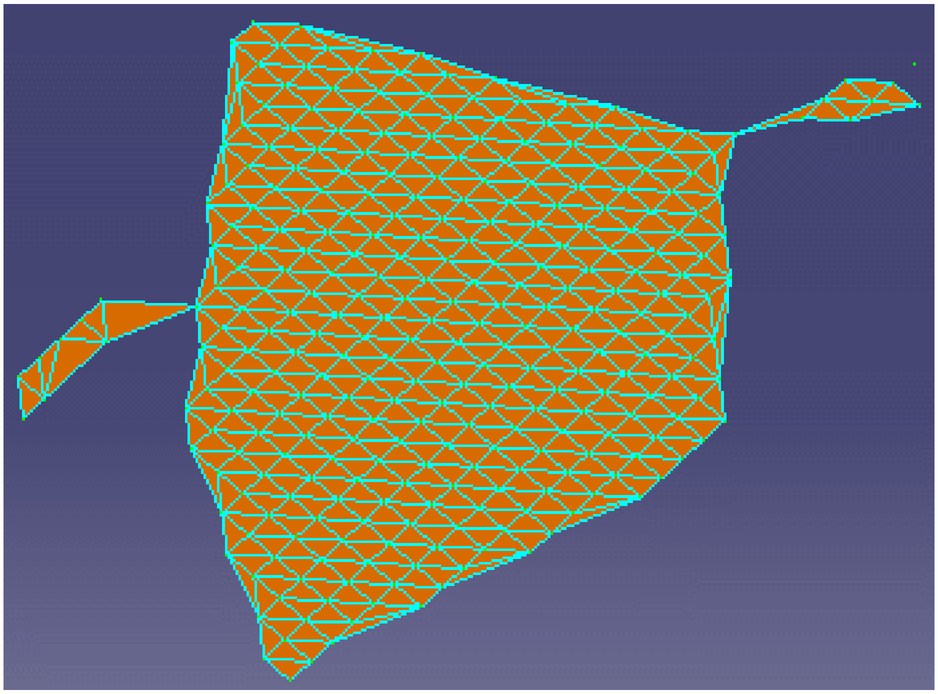
This task will show you how to clean a mesh.

Imported STL files or generated meshes may present some irregularities such as:

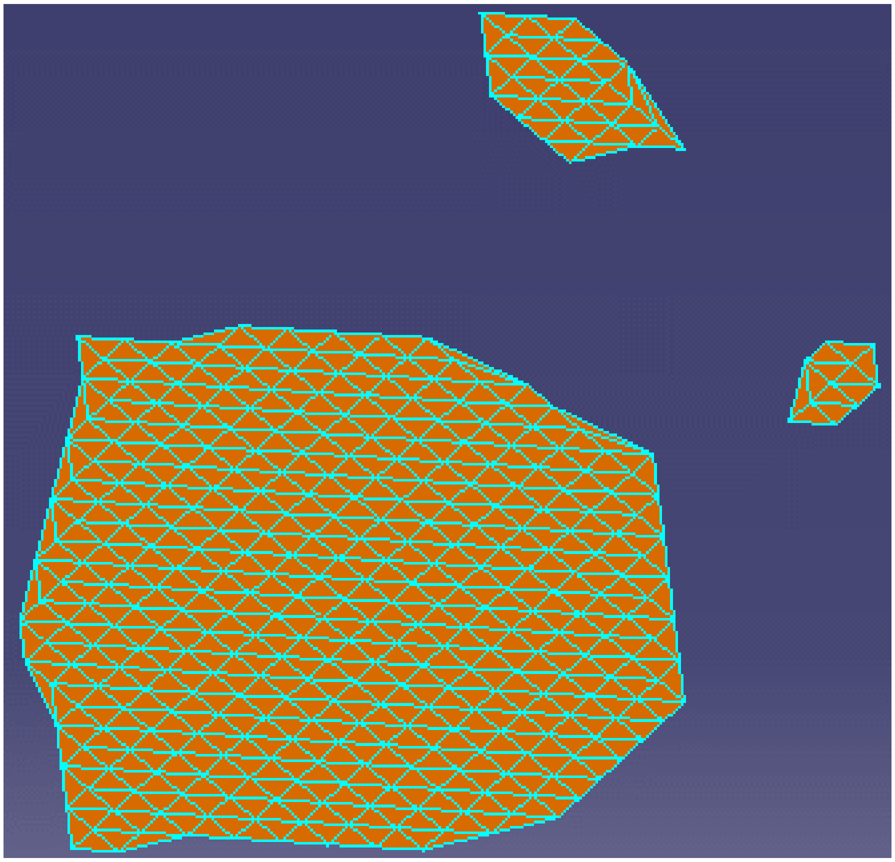
- Corrupted triangles, i.e. triangles that have the same vertex twice,
- Duplicated triangles, i.e. triangles that share the same three vertices,
- Inconsistent Orientation, i.e. triangles that can not be oriented consistently with respect to each other,
- Non-manifold edges, i.e. edges shared by more than two triangles,
- Non-manifold vertices, i.e. vertices shared by two or more connected shells.

A mesh may also present some structural problems such as:

- Orientation problems, i.e. all the triangles are not oriented in the same direction,
- Isolated triangles, i.e. triangles belonging to small connected areas of the mesh,



- Disconnected zones, i.e. the mesh is made of several disconnected zones,



- Triangles with long edges.

Mesh cleaner proposes two families of treatments on such meshes:

- Deletion, i.e. visualization and deletion of corrupted or duplicated triangles, of triangles with an inconsistent orientation, of non-manifold edges, or non-manifold vertices, of isolated triangles and triangles with long edges.

- Structure, i.e. e-orientation or split.

You can process simultaneously several types of problems in the Deletion tab. Structure problems must be processed separately.

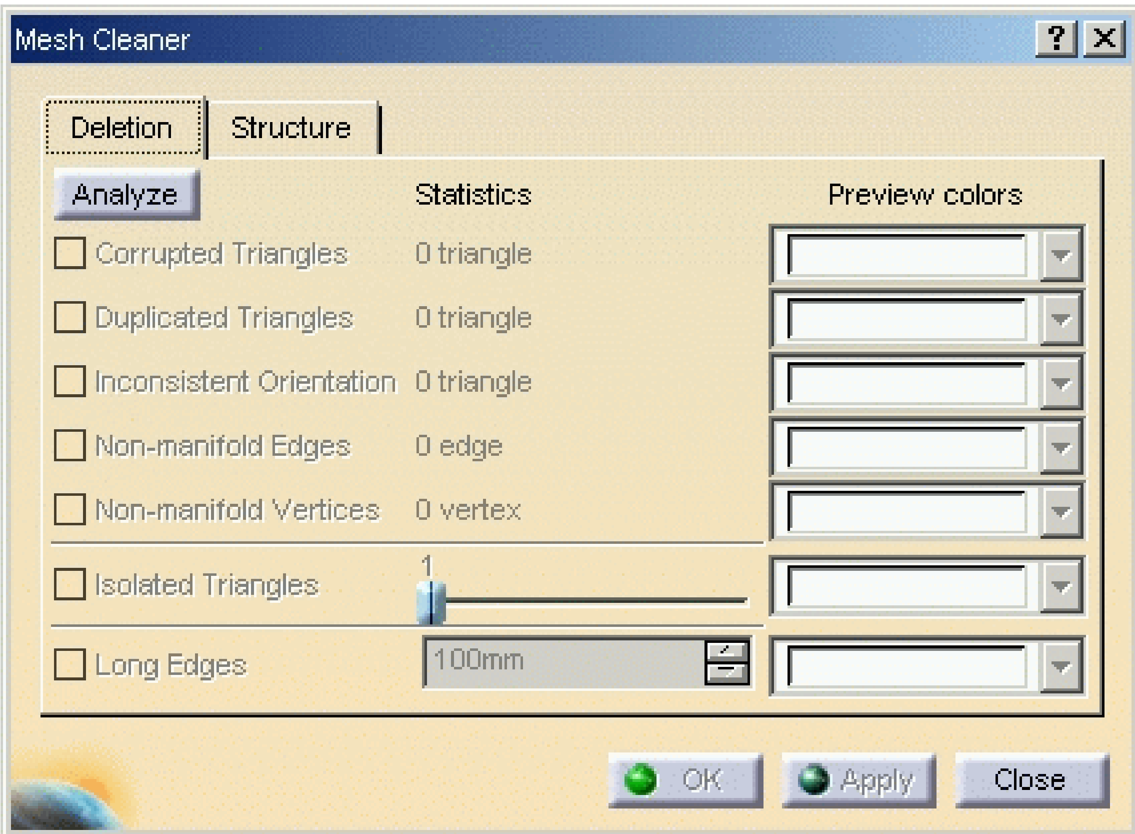
Open MeshCleaner1.CATPart the from the samples directory.

1. Click the **Mesh Cleaner** icon  and select the mesh to process. The dialog box is displayed.

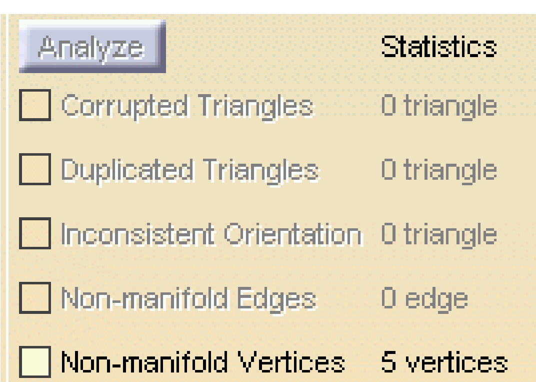
2. Go to the tab of the treatment you want to apply.

In the **Deletion** tab:

3. Push the **Analyze** button.

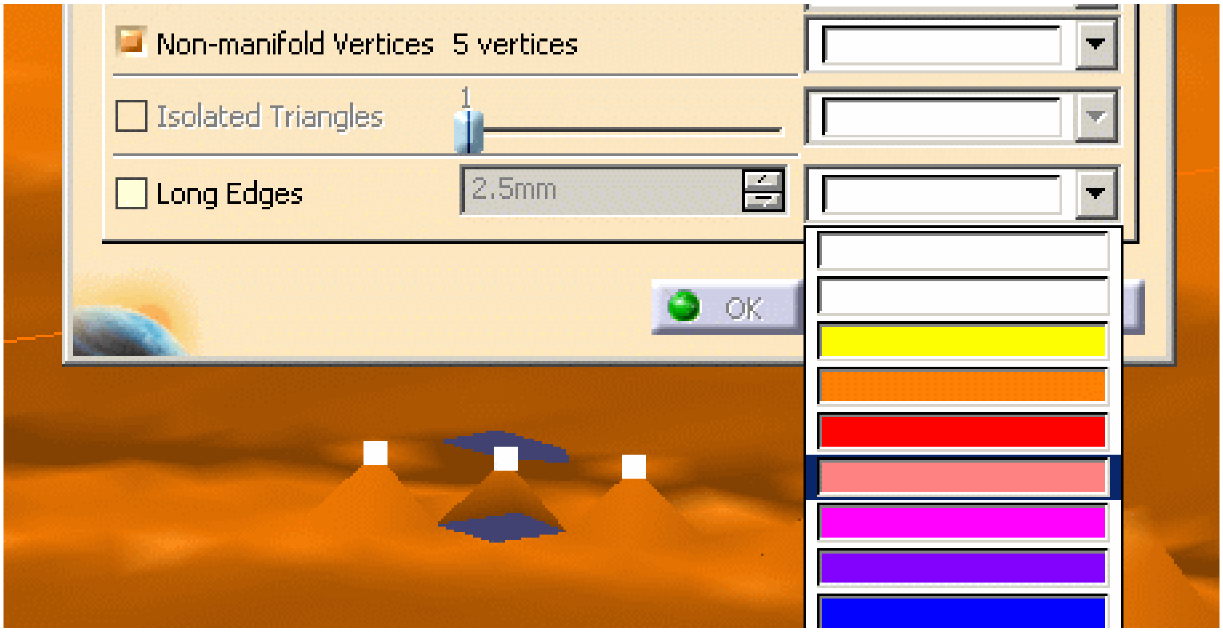


Once the analysis is completed, the line(s) corresponding to the problem(s) found become active. The **Statistics** column is updated with the number of cases found for the four first lines. Here, only non-manifold vertices have been found, and there are 5 such vertices.



4. You can visualize the problems found:

- Check the line(s) of the problem you want to visualize. By default, they are displayed in white. You can choose another color in the **Preview colors** combo box.

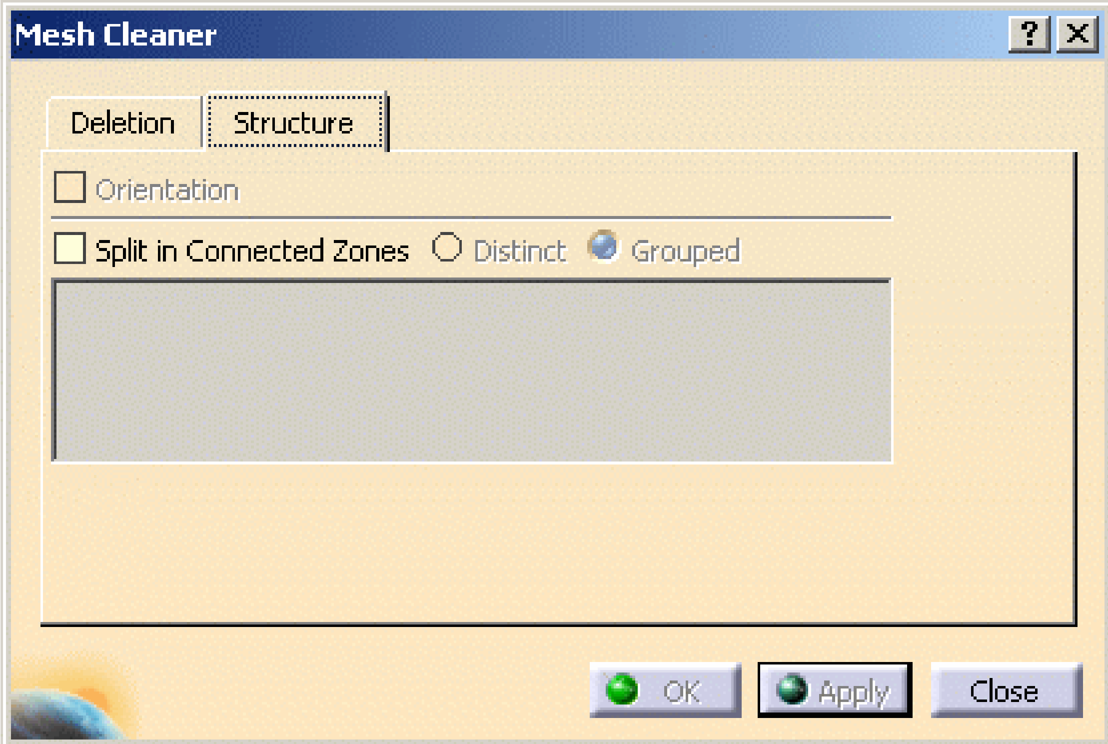


5. For **Isolated Triangles**, use the slider to define the maximum number of triangles that a disconnected area may contain. If you set the slider to n, all the areas containing between 1 and n triangles will be visualized, then deleted. The limit values of the slider are defined according to the mesh.

For **Long Edges**, use the spinner to define the maximum allowed length edge of triangles. All triangles with edges longer than this value will be deleted.

6. Click Apply to delete the unwanted elements and OK to exit the action and save the processed part.

In the **Structure** tab:



1. Check the **Orientation** line and click Apply to re-orient triangles, if that is possible.

or

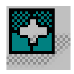
1. Check the **Split in Connected Zones**. The text box below is updated with the number of connected zones found. You can then choose to split them in **Distinct** zones or in one **Grouped** zone by checking the appropriate option and clicking Apply. **SubMesh.x** elements are then created in the specification tree.
2. Click OK to exit the action and save the processed part.

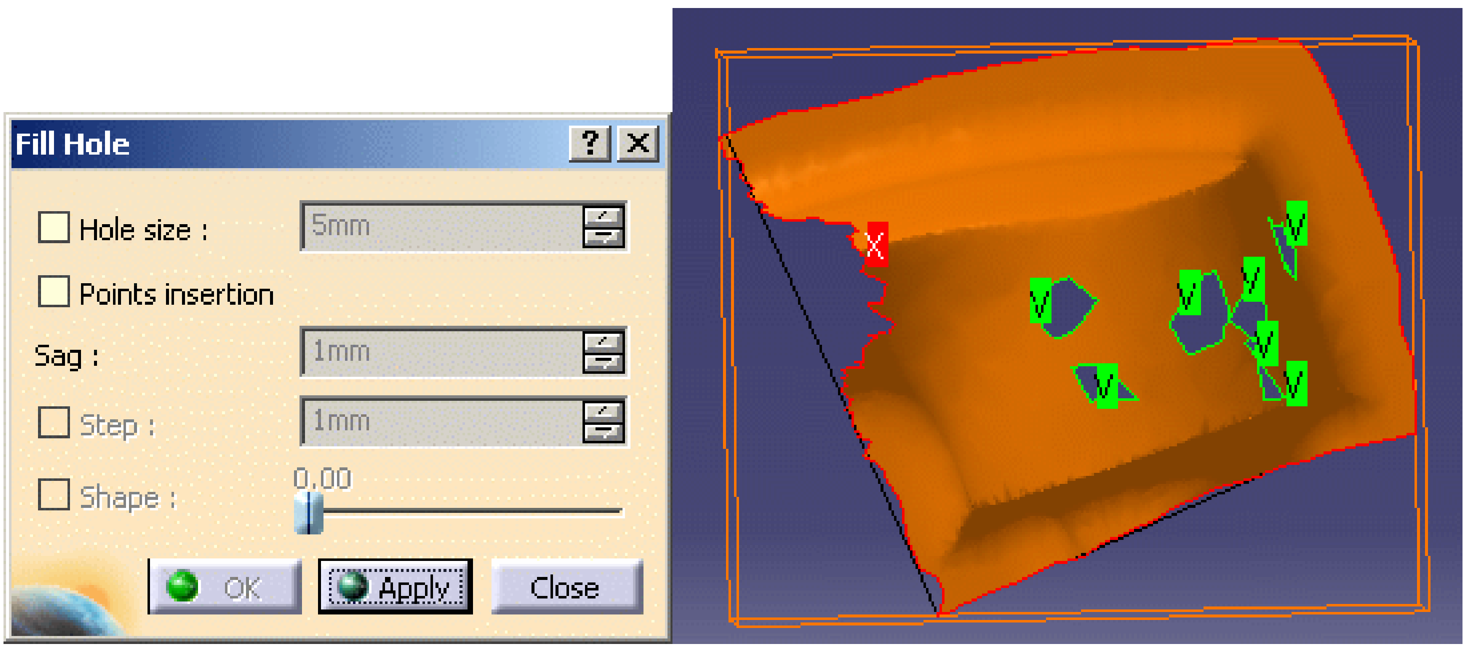
Filling Holes on Meshes

This task shows you how to fill holes on meshs with the following advantages:

- You can either select the holes manually or automatically.
- The filling can be basic (no point inserted, basic remeshing) or more sophisticated (points are inserted, and the meshing can be flat or curved).

Open the Fillhole1.CATPart from the samples directory.

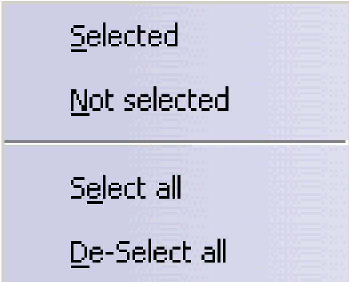
1. Select the **Fill Holes** icon  and Cloud Import.1.



2. The dialog box is displayed. A first recognition of the holes is done:

- X means the hole is not selected,
- V means the hole is selected,
- The biggest hole found is considered as "exterior". Since you usually do not want to fill the outside of the part, this hole is not selected, but you can change its status.

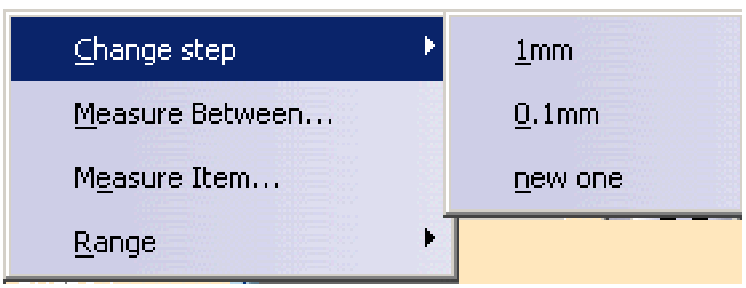
3. Click on the label to select or de-select a given hole or right-click on a label to call the contextual menu:



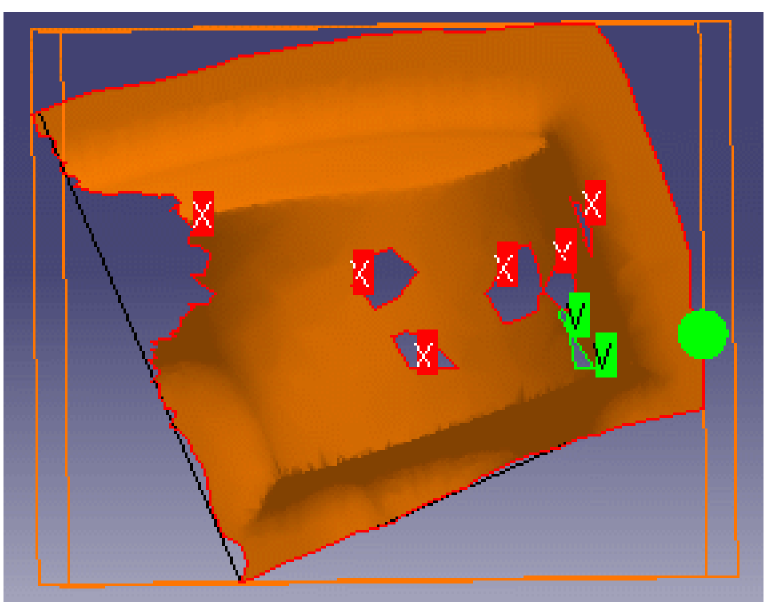
Use Selected/Not selected as a toggle on a single hole, and Select all/De-Select all as a toggle on all holes.

4. If you want to select the holes to fill automatically, check the Hole size option. A sphere is displayed. You can change its diameter in the box on the right:

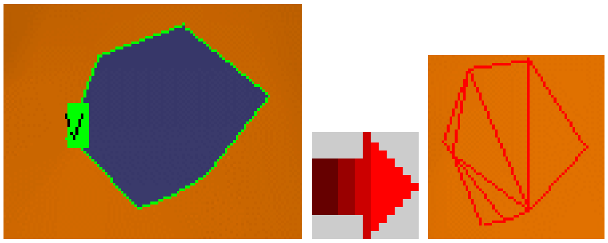
- either enter a value,
- or right-click to call the contextual menu:



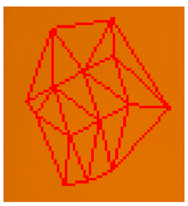
All the holes smaller than this sphere are selected.



5. Click Apply. A basic meshing is computed to fill the hole:

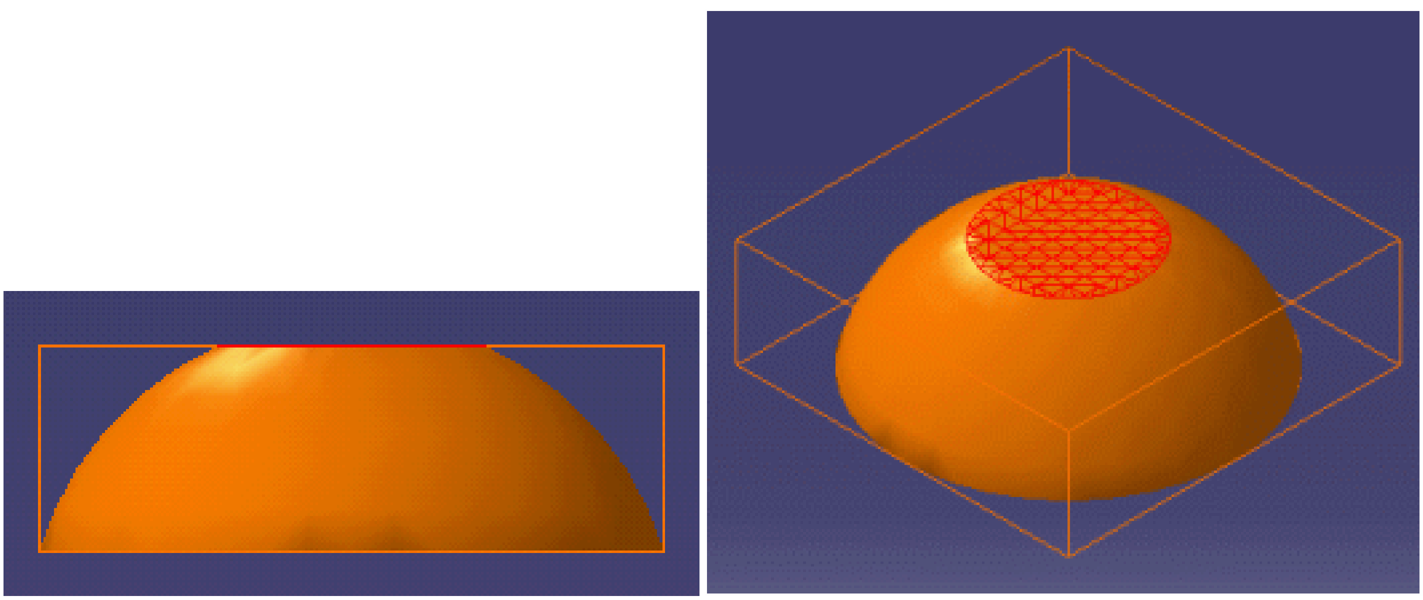


6. Check the Points insertion option and click Apply. A new meshing is computed, with more meshing points:

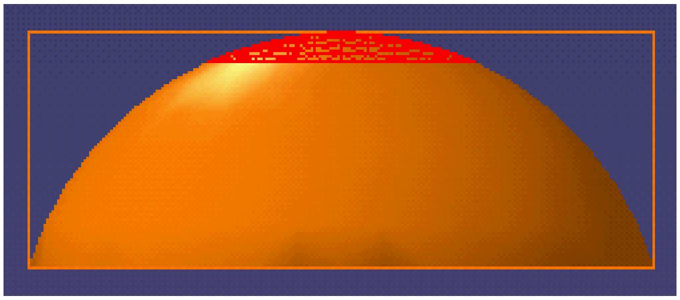
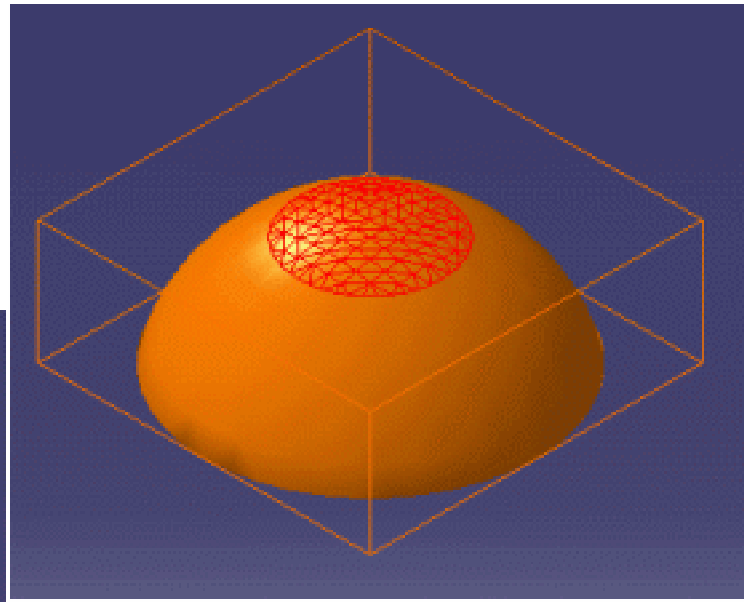


With this option, you can set a Sag and a Step (i.e. the maximum length of the mesh edges) value, either directly or with the contextual menu.

7. By default, the filling is flat:



Check the Shape option for a curved filling. The filling is computed on a virtual surface, the curvature coefficient of which is controlled by the slider on the right: increase this coefficient to increase the curvature of the filling.

8. Click Apply to visualize the filling with given parameters. If you are not satisfied, click Undo, modify the parameters and click Apply to refresh the filling. Once you are satisfied, click OK to validate and exit the action

- Undo is available within the action, not after you have exited the action.
- The holes to fill must be closed.
- When no coherent result can be computed, an error message is displayed.

Interactive Triangle Creation

This task will show you how to create mesh triangles interactively:

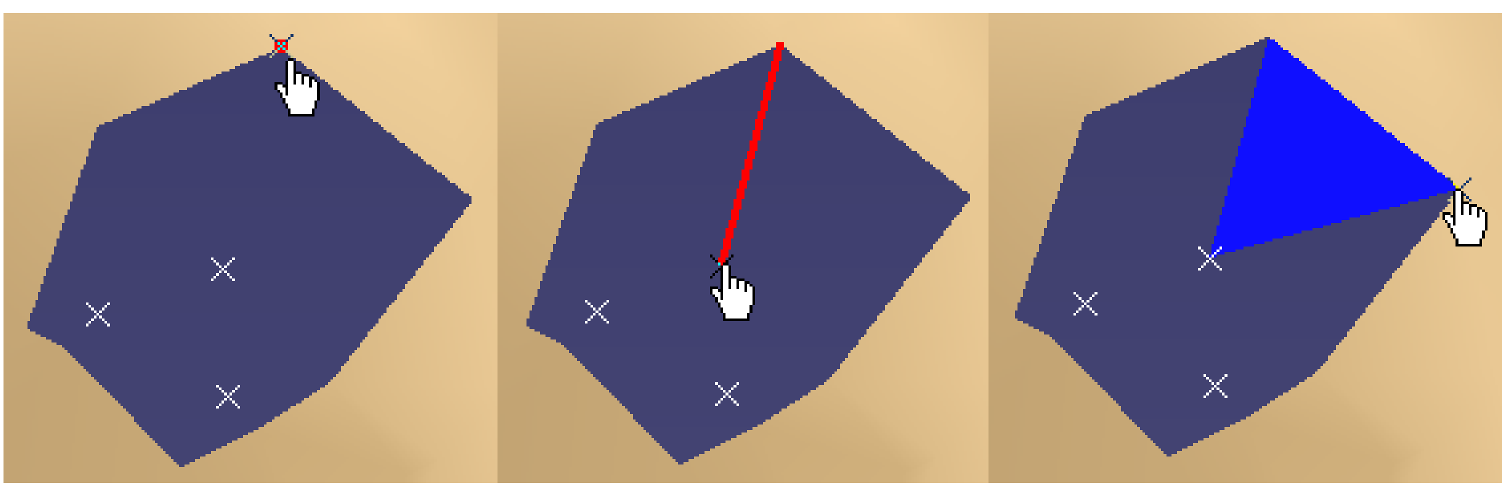
- to create or modify a mesh quickly or
- to simplify hole filing by creating bridges within a hole.

Open the Fillhole1.CATPart from the samples directory.

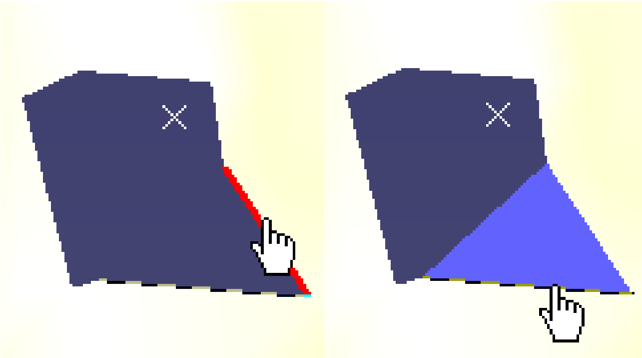
1. Click the **Interactive Triangle Creation** icon . The Interactive Triangle Creation dialog box is displayed.

2. To create a new mesh triangle, you can input:

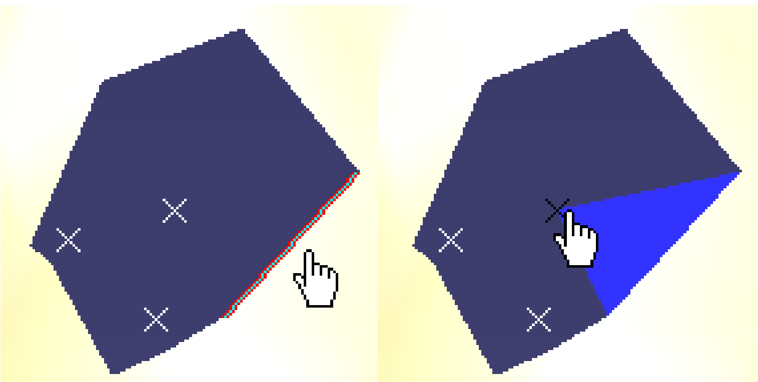
   - three points (vertices of an existing mesh or not), or



   - two neighboring edges of a mesh (with a vertex in common), or

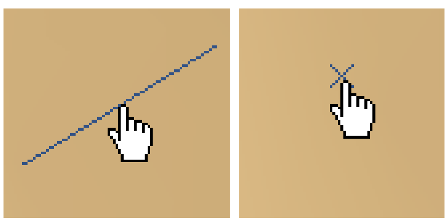


- an edge of an existing mesh and a point (vertex of an existing mesh or not).

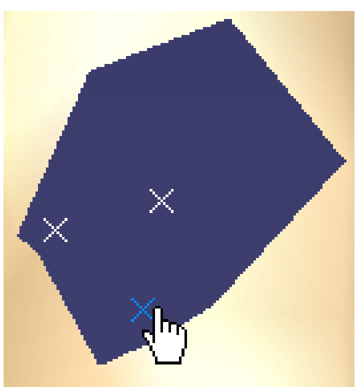


To make the selections easier:

- As you pass the cursor over a mesh, the edge or the vertex under the cursor are highlighted:



- As you pass the cursor over a point, it is highlighted:



When the first selection is a point, it is displayed in red.

- If the next selection is another point, a red line is displayed between those two points. At the next point selection, a triangle is proposed and displayed in blue.
- If the next selection is an edge, a triangle is proposed and displayed in blue.

When the first selection is an edge, it is displayed in red. At the next selection, a triangle is proposed and displayed in blue.

3. Once a triangle is proposed:

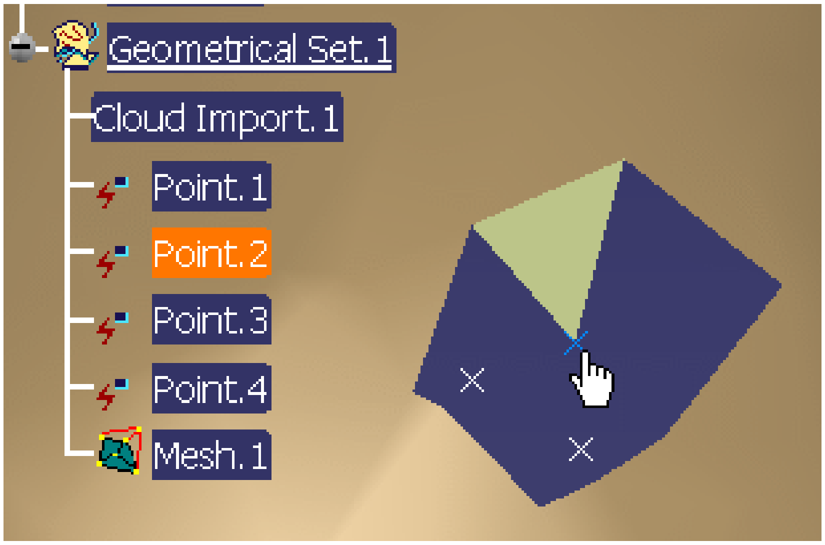
   - you can select other elements to define more triangles. They will be proposed and displayed in blue, or
   - click Apply. The triangle(s) displayed in blue is(are) created temporarily. You can create further triangles.

4. Click OK to validate the creation of triangles:

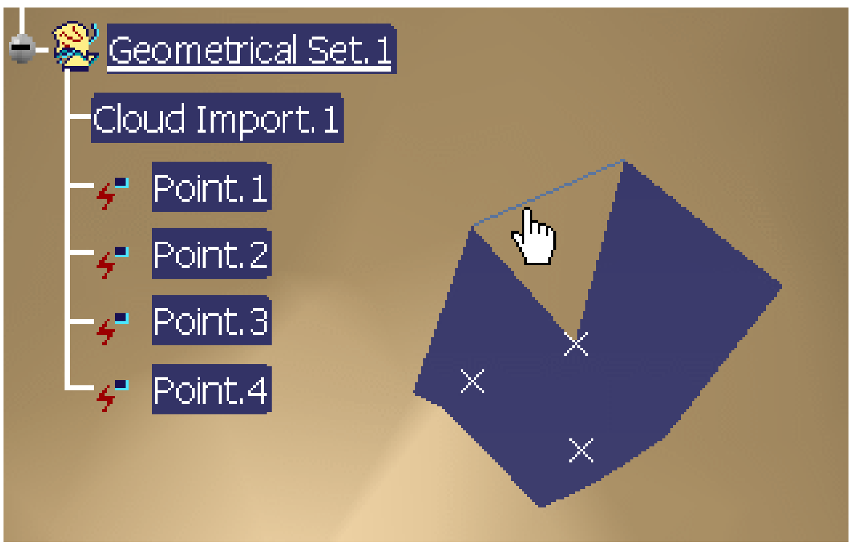
   - if the first element picked was a point that did not belong to a mesh, a new Mesh.x element is created (even if the other elements selected belong to a mesh),
   - if the first element picked was a vertex or an edge of an existing mesh, this mesh is modified and no other mesh is created.

   or Cancel to exit the action without creating any triangles.

The first element picked is **Point.2**. A new mesh is created.



The first element picked is an edge of **Cloud Import.1**. **Cloud Import.1** is modified, no new mesh is created.



Decimating Meshes

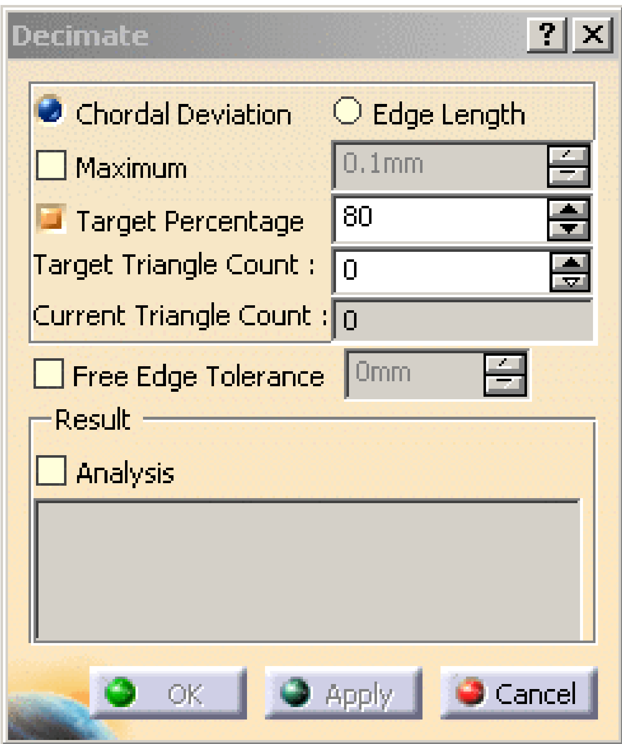
This task will show you how to decimate a mesh.

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.

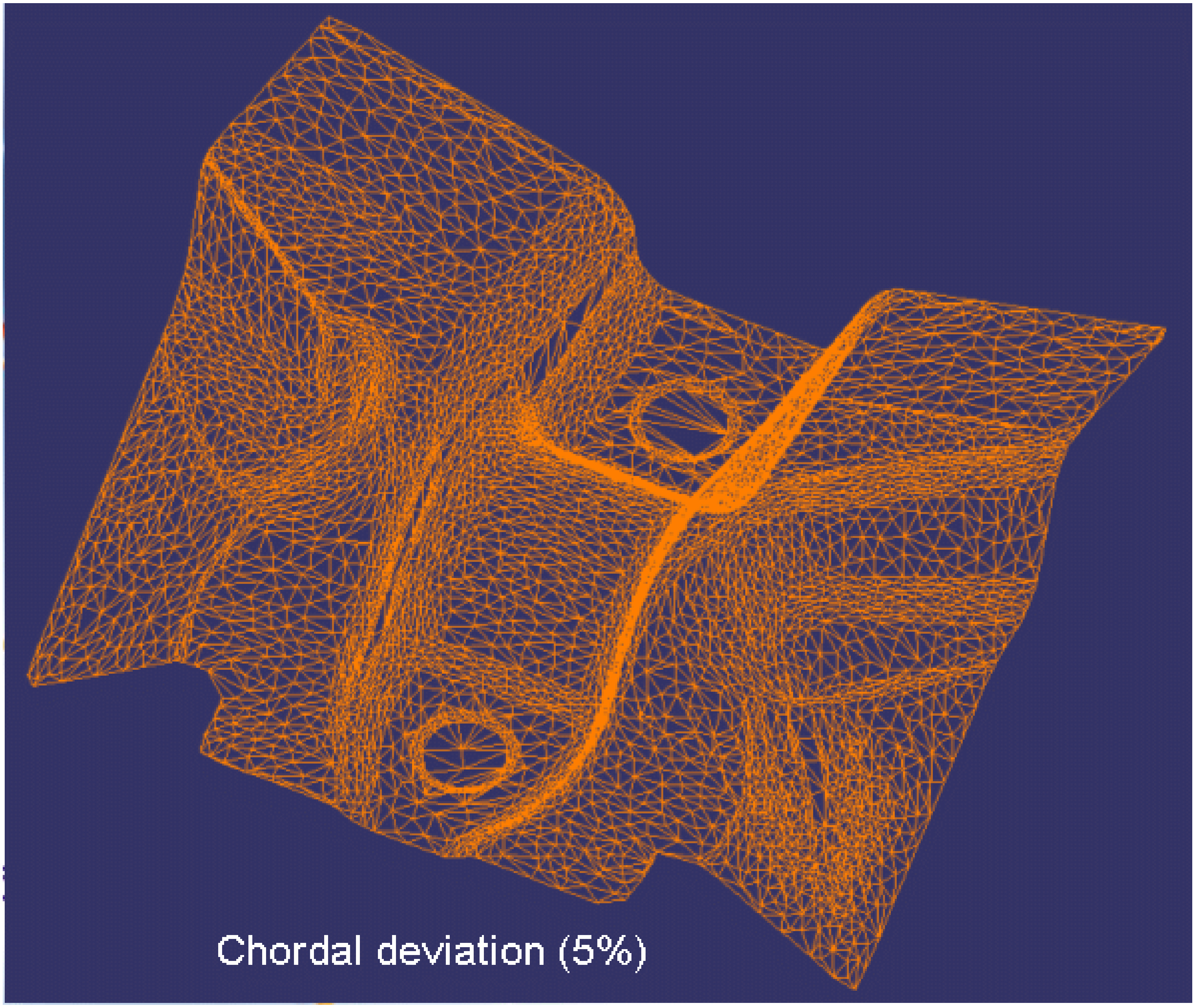
3. Then, decide how you want the decimation to stop:

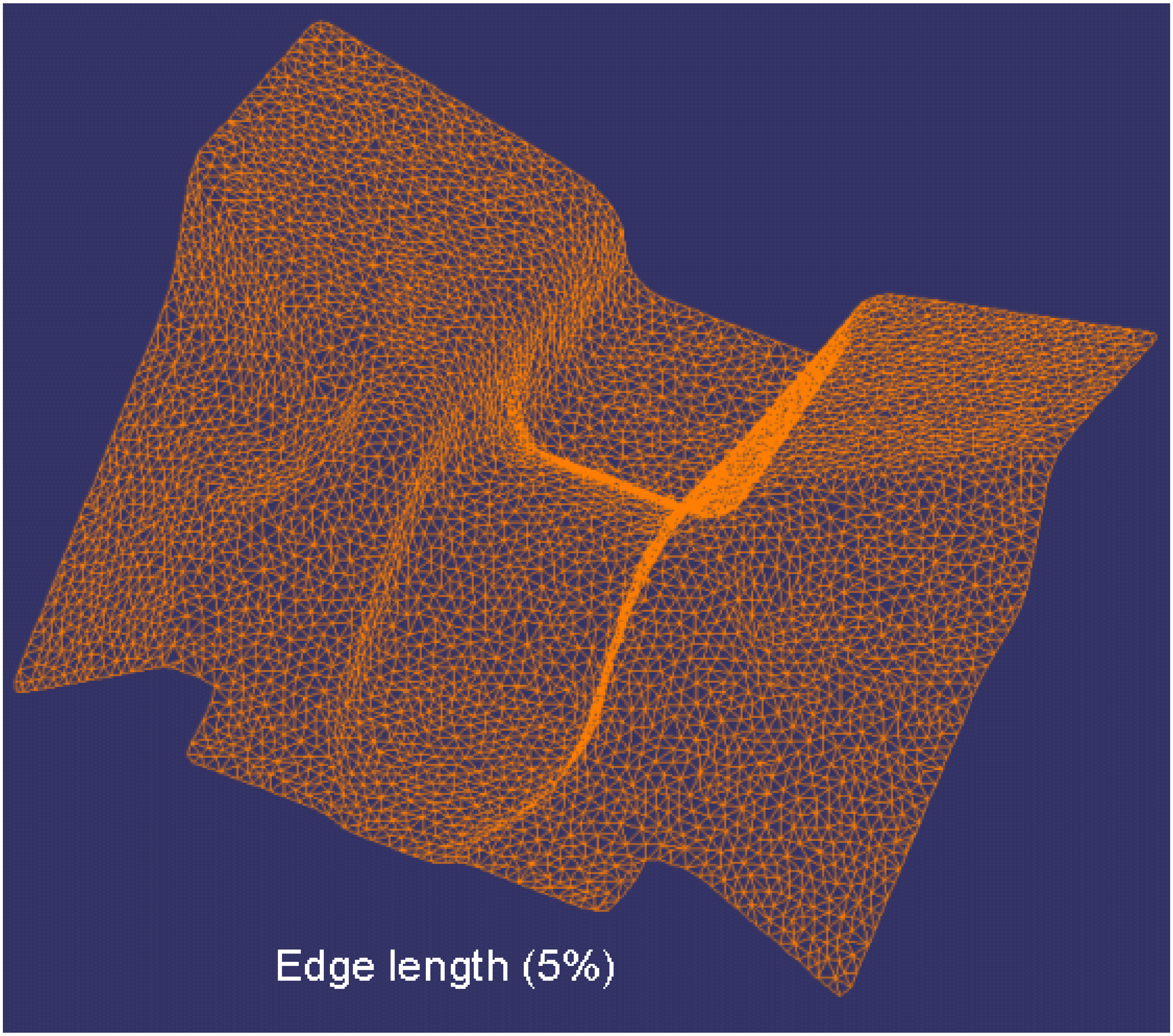
- For a decimation by **Chordal Deviation**, you can check **Maximum** and enter a value. It is the chordal deviation that should not be exceeded during decimation. Decimation stops when the chordal deviation limit has been reached.

- For a decimation by **Edge Length**, you can check **Minimum** and enter a value. The command stops when further decimation could collapse edges of length greater than the value entered.

- For both types of decimation, 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.

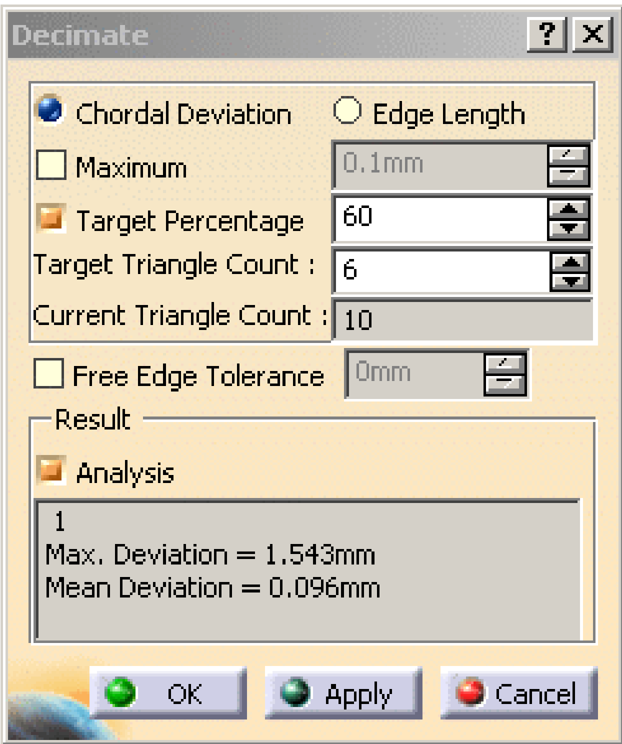

Chordal deviation (5%)



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.

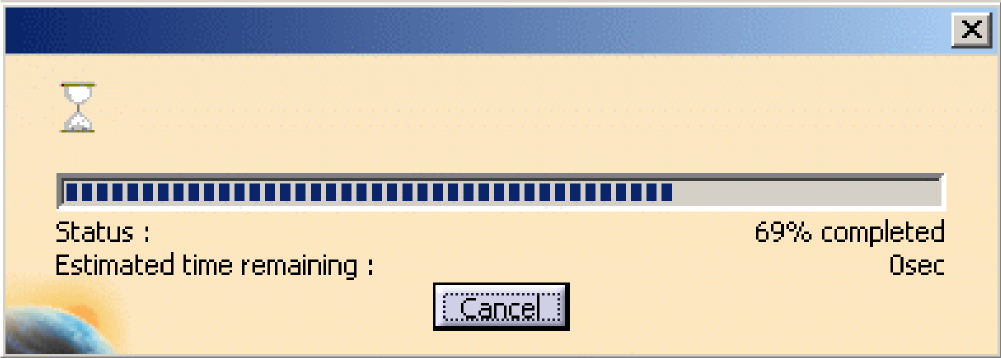
5. The chordal deviation that can be used as a stopping criterion is an approximation of the chordal deviation between the original mesh prior to decimation and the decimated mesh. Therefore it may be useful to know the maximum distance and the mean distance between the original mesh and the result mesh at the end of the command.

To do so, check **Analysis** before clicking **Apply**. At the end of the decimation, the maximum and the mean deviation will be reported as shown below. Unlike the value entered in the **Maximum** field, they are the true deviations between the original mesh and the result mesh.



- However, **Analysis** may be time consuming, especially for large models. we recommend that you turn it off when you do not need it.
- The meshes can be decimated in several steps (click **Apply**= one step). The deviations displayed are those between the original model and the last result (not those between the previous and the current results). For this reason, the deviations will be increasing in value after each **Apply**.
- Any selection change resets the original model for the analysis.
- **Undo** is not taken into account by the **Analysis**.

At each **Apply**, a progress bar is displayed. A **Cancel** button is available to stop the decimation.



6. Click **OK** to confirm the decimation once you are satisfied with the result.

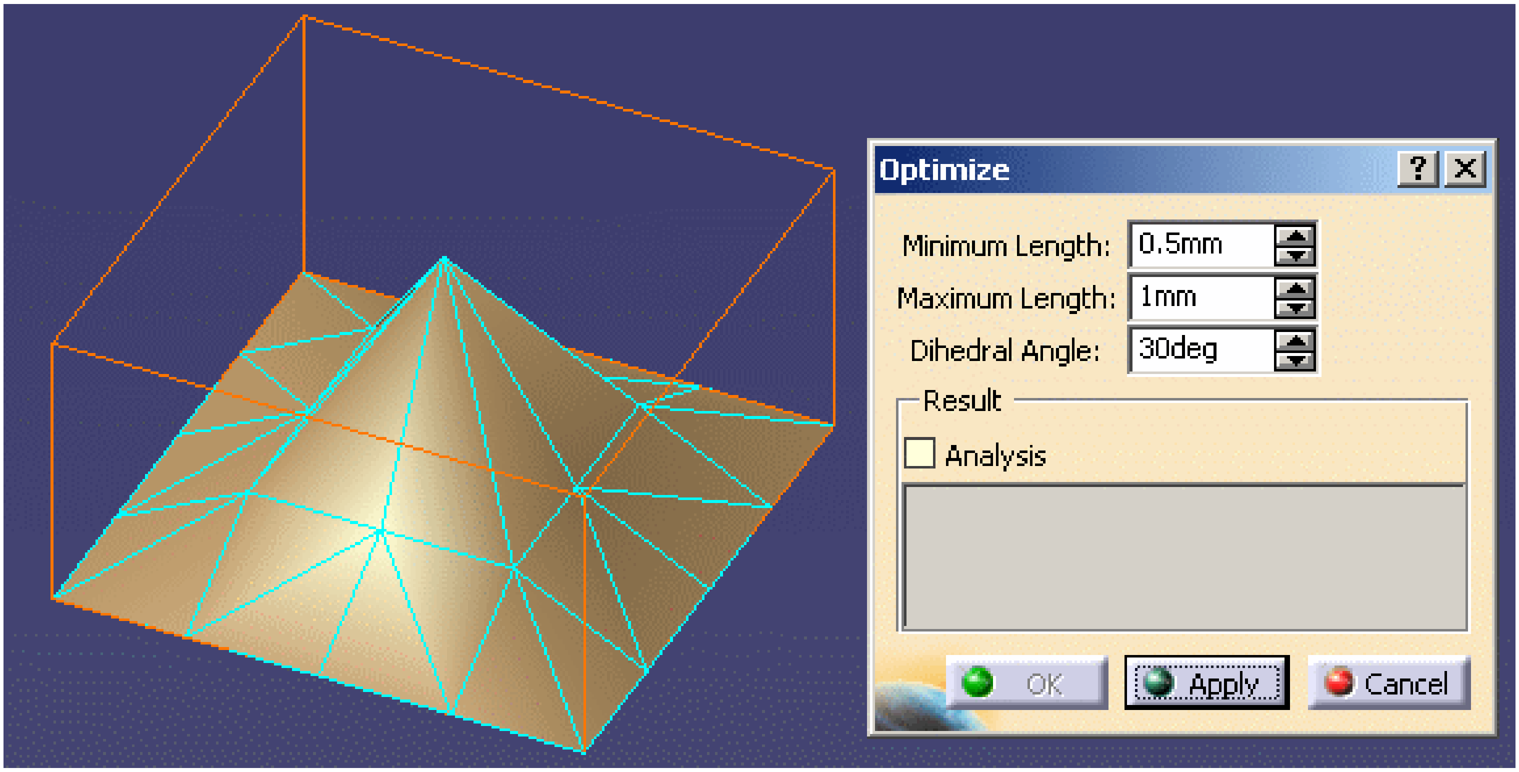
Optimizing Meshes

This task will show you how to optimize an existing mesh, i.e. redistribute and reshape the triangles within the mesh. This way you will obtain a more homogeneous mesh, e.g. for analysis purpose.

- Be aware that this optimization tends to modify the shape of the model!
- Non-manifold meshes can not be processed. You should use the Mesh Cleaner.
- Mesh Optimize is an iterative operator using edge splits and edge collapses. If an edge is too long, it is split in two. If an edge is too short it is collapsed. Consequently, there is a minimum ratio between the minimum and maximum edge length; otherwise an edge that has just been split could be collapsed at the next step and then split again and so on.

Open the SmoothMesh01.CATPart from the samples directory. You may want to display the triangles of the mesh, using the **Cloud Display** command.

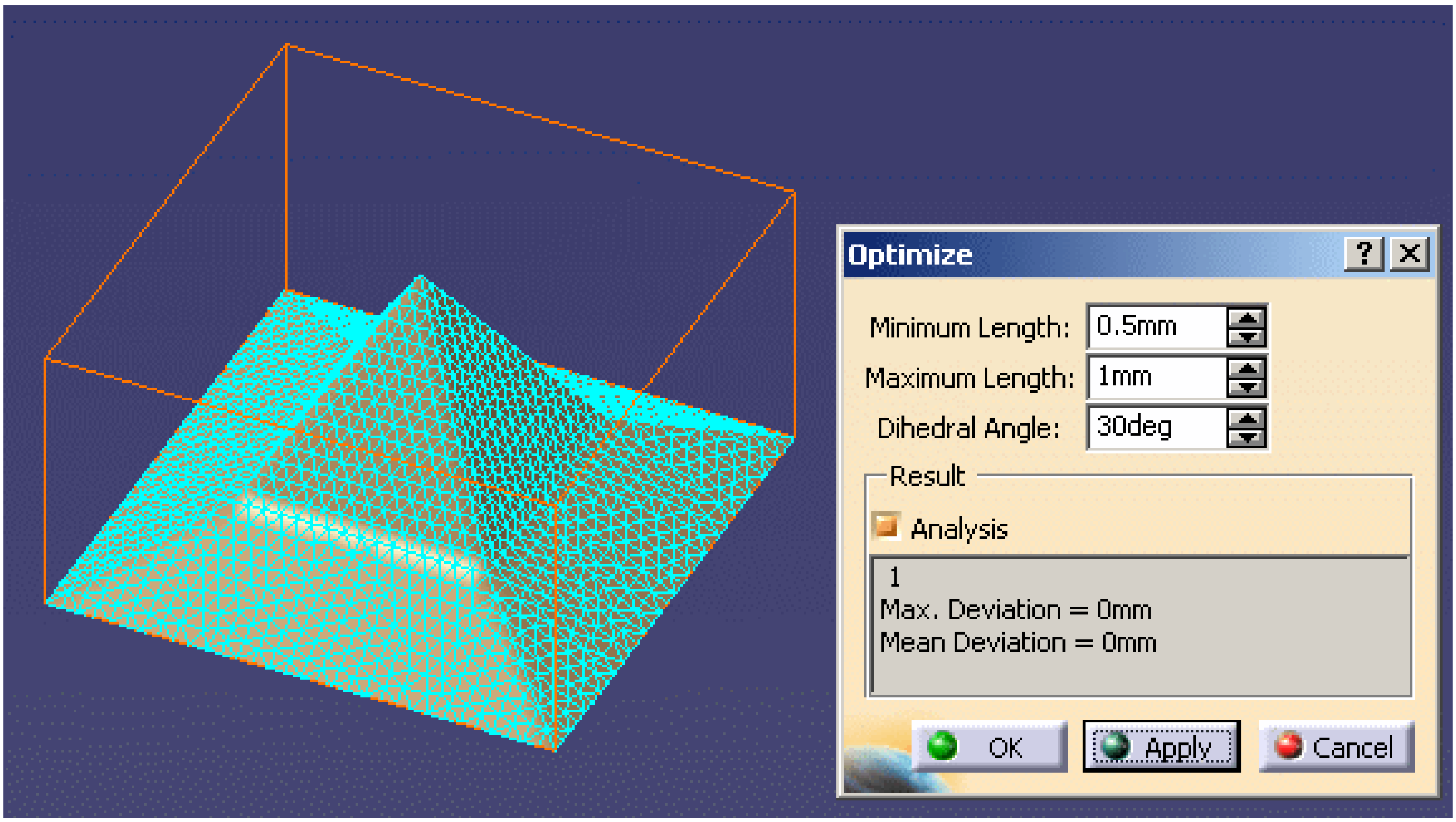
1. Click the **Optimize** icon and select the mesh.



2. Set the **Minimum Length**, **Maximum Length** and **Dihedral Angle** according to your needs.

- **Minimum Length** and **Maximum Length** refer to the length of the edges of the triangles.
- The **Minimum Length** must be less than or equal to the half of the **Maximum Length**.
- **Dihedral Angle** is the angle between two triangles.
- All the triangles with edges shorter than the **Minimum Length** will be collapsed so that their edges reach this **Minimum Length**.
- All the triangles with edges larger than the **Maximum Length** will be refined so that their edges reach this **Maximum Length**.
- Al the triangles with an **Dihedral Angle** lower than the value given will be flipped.
- Make sure the chosen values are coherent with your model.

3. Check the **Analysis** option to display the maximum and the mean deviations (distances between a vertex and its initial position) in the dialog box.
4. Click Apply.



5. Click OK to validate the optimization and modify the initial mesh, or Cancel to exit the action without any modification.

Operations

This chapter deals with the operations on meshes.

Merging Meshes

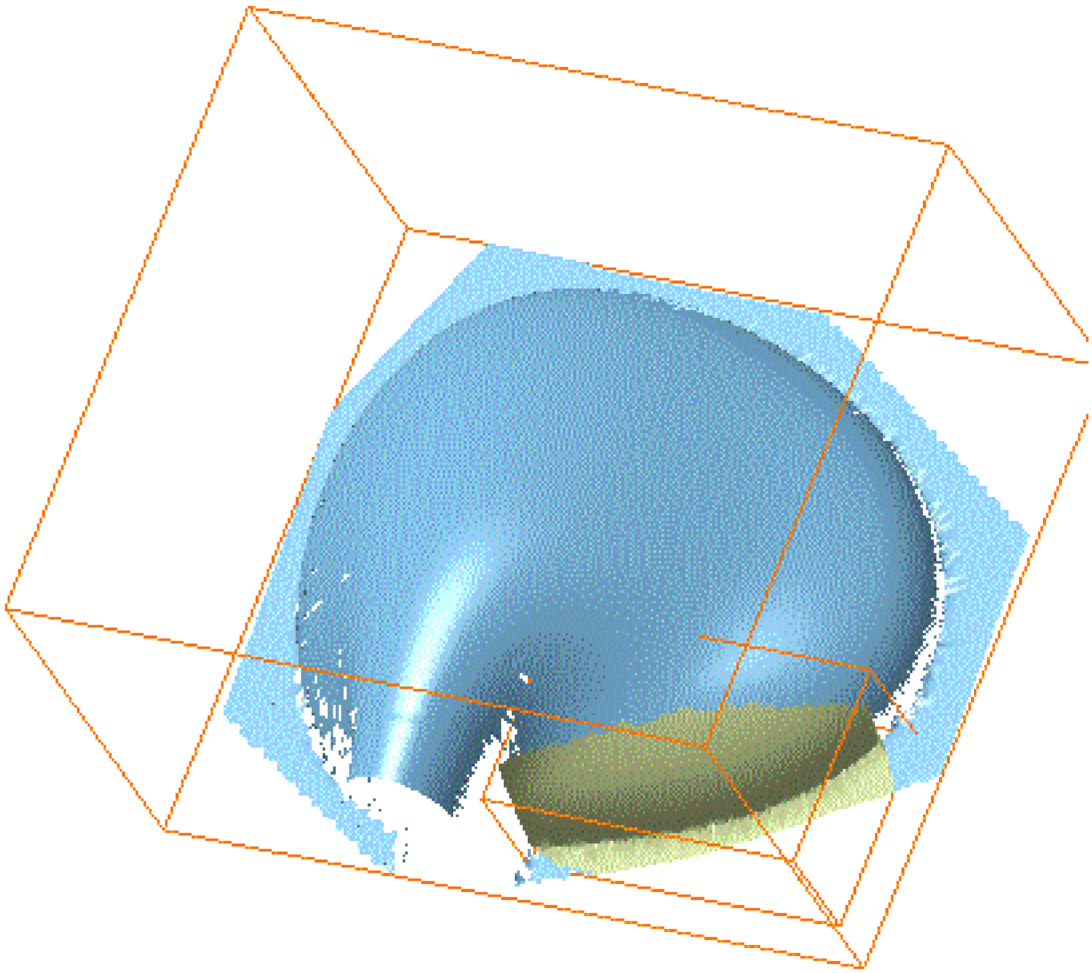
P2

This task shows how to merge two meshes. This action is useful especially for meshes resulting from a constrained mesh: it removes the seam created by the constrained mesh.

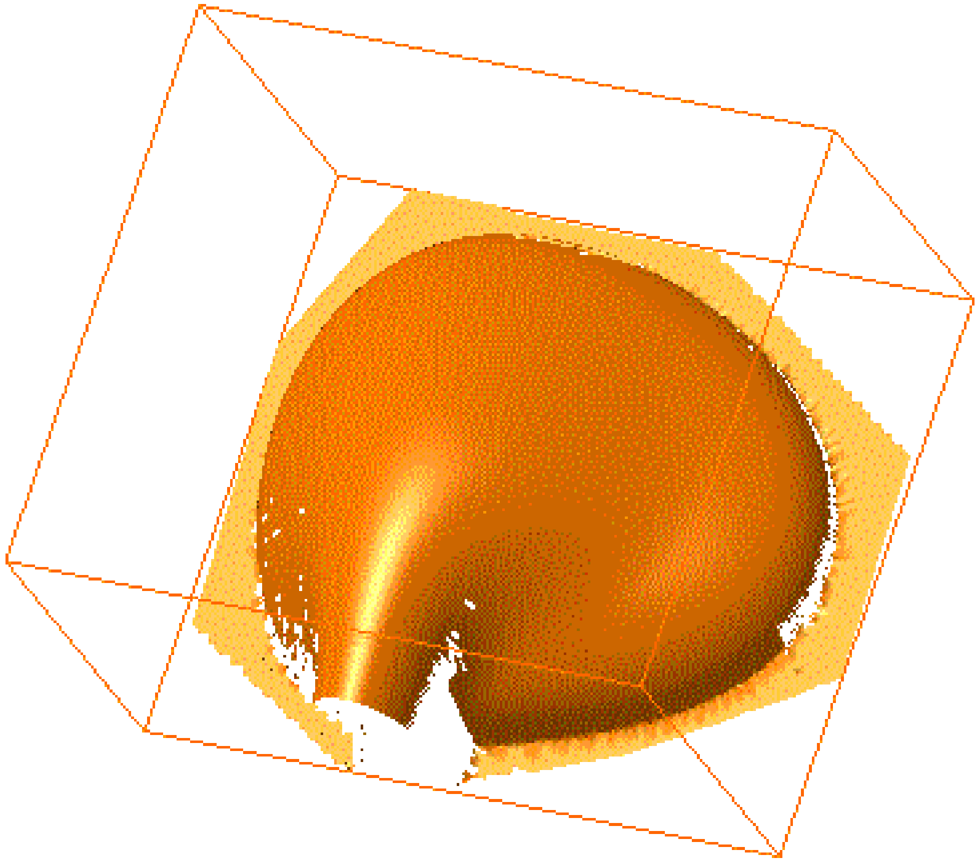
This action does not modify the triangles of the meshes (holes are not filled, no management of overlaps, ...). For a good result, repair the meshes first, using constrained meshing, fill hole action, ...

Open the MergeMeshes1.CATPart model from the samples directory. It consists of two meshes, the smallest resulting from a constrained meshing.

1. Select the **Merge Meshes** icon . The **Meshes Merge** dialog box is displayed.
2. Select the meshes you want to merge. The list in the dialog box is updated.
3. To remove a mesh from the list of meshes to merge, make its name active in the list and check **Remove**.
4. To replace a mesh in the list of meshes to merge, remove it, then select a new mesh.
5. Once you have selected all meshes to be merged, click **OK**. A new single-cell mesh is created in the specification tree under the name **Meshes Merge.x.**




Meshes Merge
Meshes to merge
Mesh Creation.1
Mesh Creation.2
Remove
OK
Cancel



Splitting Meshes

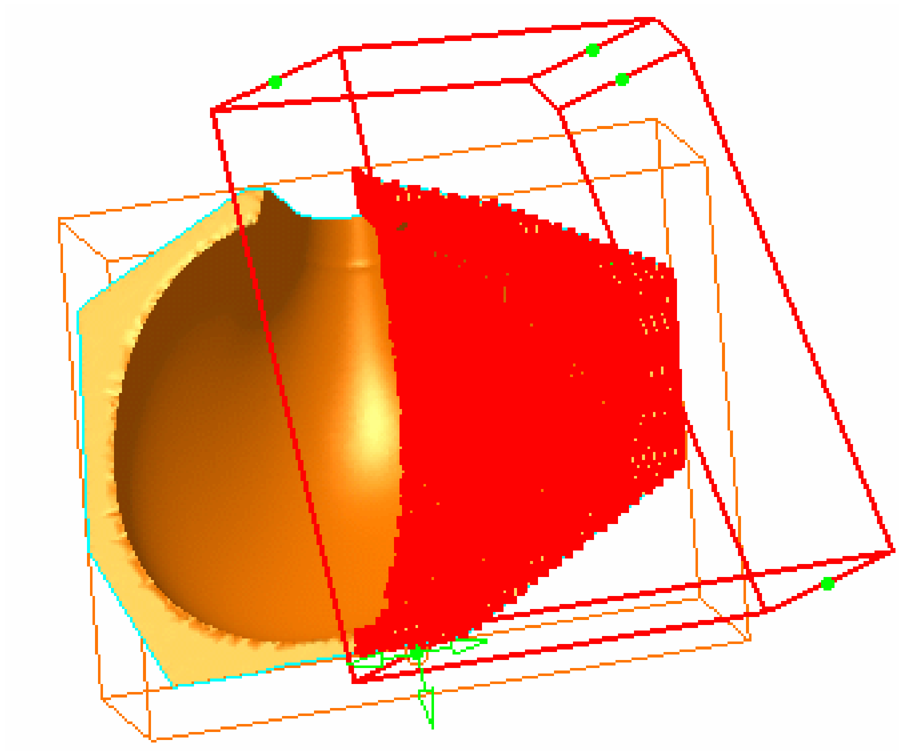
This task shows how to split a mesh or a cloud.

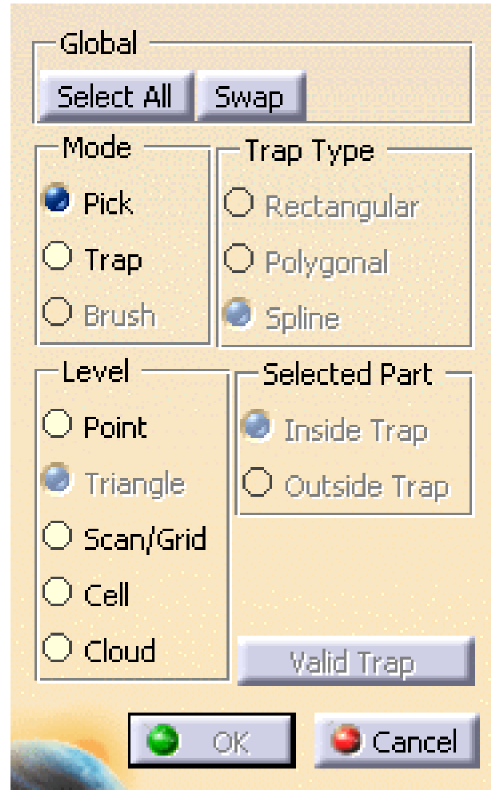
Open the Cloud.CATPart model the from the samples directory.

Click here for more information on the dialog box.

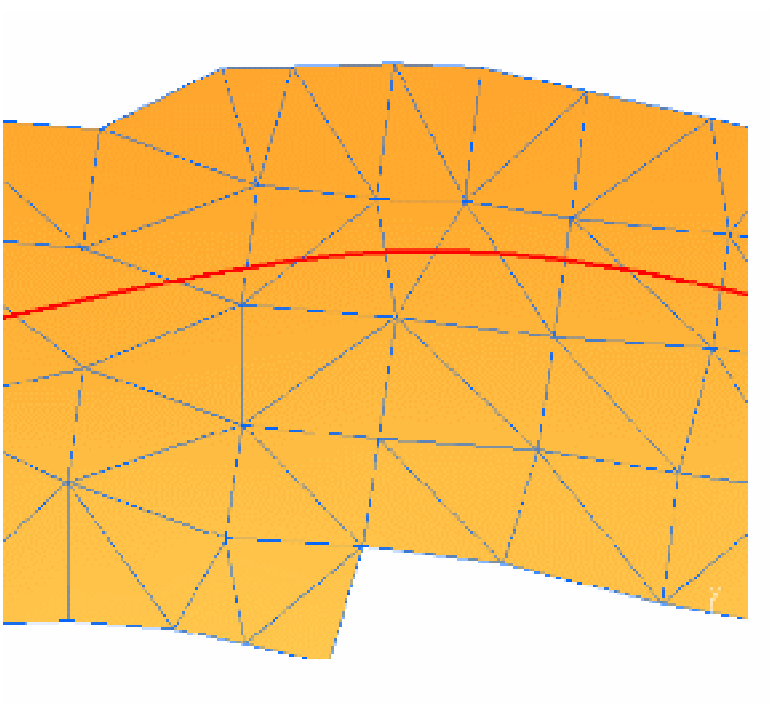
1. Select the **Split** icon . The **Split** dialog box is displayed.
2. Select the mesh or the cloud to split.
3. Select a portion of the element according to the Activation operating mode.
4. Once the selection is done, click OK to validate.
5. The input element is hidden. Two elements are created in the specification tree:

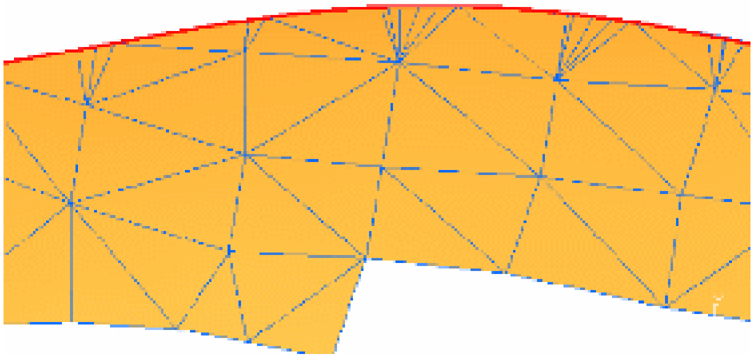
- **SplitCloud.1** and **SplitCloud.2** if you the input element was a cloud. **SplitCloud.1** corresponds to the remaining portion of the original cloud, **SplitCloud.2** is the split cloud (portion selected).
- **SplitMesh.1** and **SplitMesh.2** if you the input element was a mesh **SplitMesh.1** corresponds to the remaining portion of the original mesh, **SplitMesh.2** is the split mesh (portion selected).
- The output element indexes are increased if further splits occur.





- If the selection is empty, or if the whole input element is selected, no split element is created.
- To retrieve the original input element, recall it from the No Show, or merge the two split elements.
- When you split a mesh using the **Trap** option, the triangles are smoothly cut by the trap line.

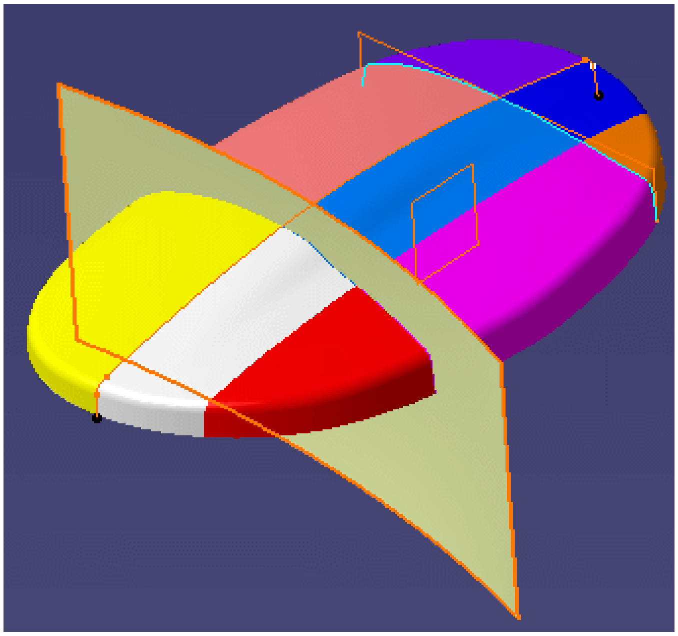




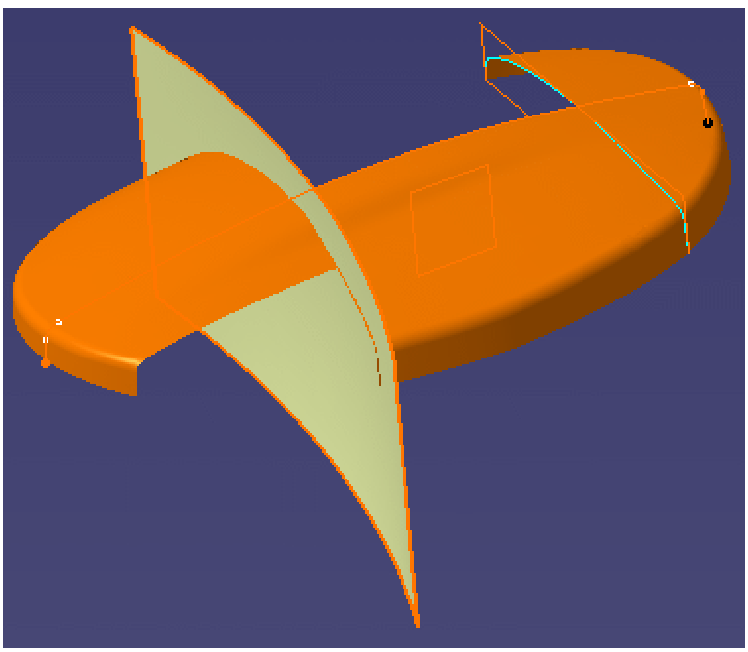
Trim/Split

This task will show you how:

- to split a mesh in several meshes (displayed in different colors below):

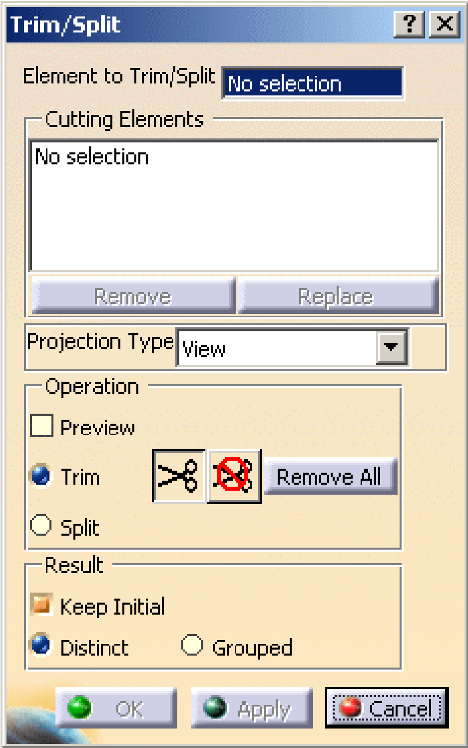


- and/or trim portions of the mesh delimited by curves, planes, surfaces or other meshes:



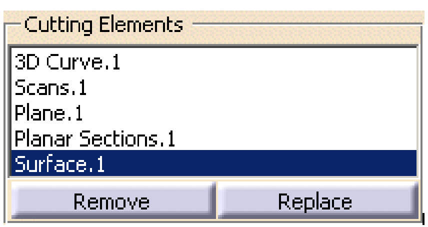
Open TrimSplit1.CATPart from the samples directory.

1. Click the **Trim / Split** icon . The dialog box is displayed.



2. Select the mesh to trim or split. It can consists of several cells.

3. Select the **Cutting elements** .



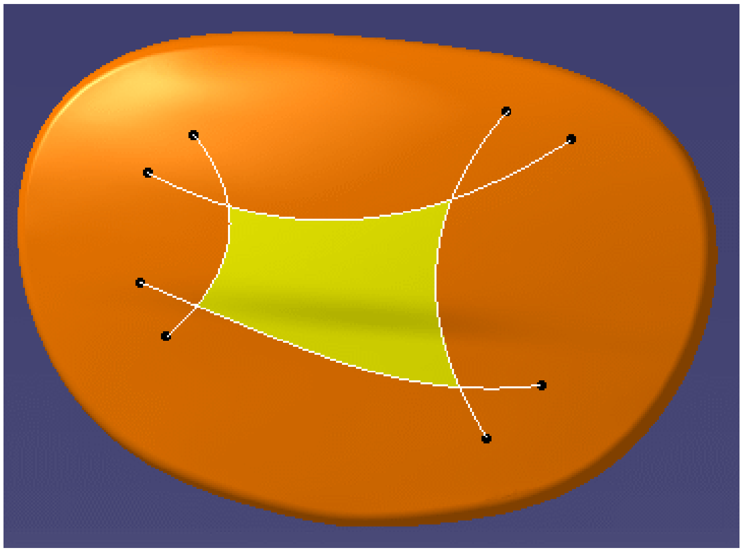
They can be:

- scans,
- curves,
- planes,
- surfaces,
- meshes.

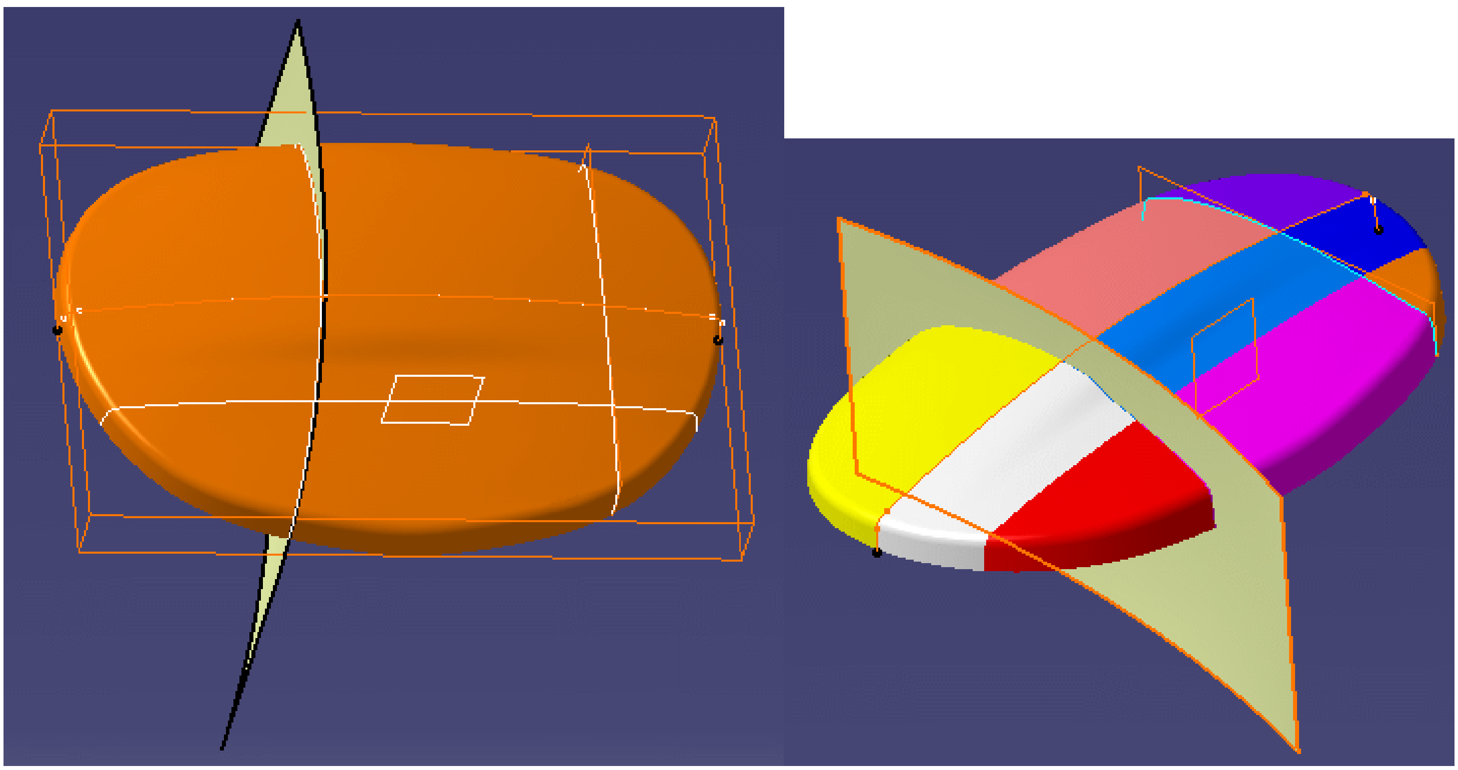
Those input elements are highlighted in the graphic area and listed in the dialog box. To remove a cutting element, select it in the dialog box list and push the **Remove** button. To replace a cutting element with another, select is in the dialog box list, push the **Replace** button and select the new cutting element.

To define an area, the projection curves of the cutting elements must intersect each other or intersect free edges.

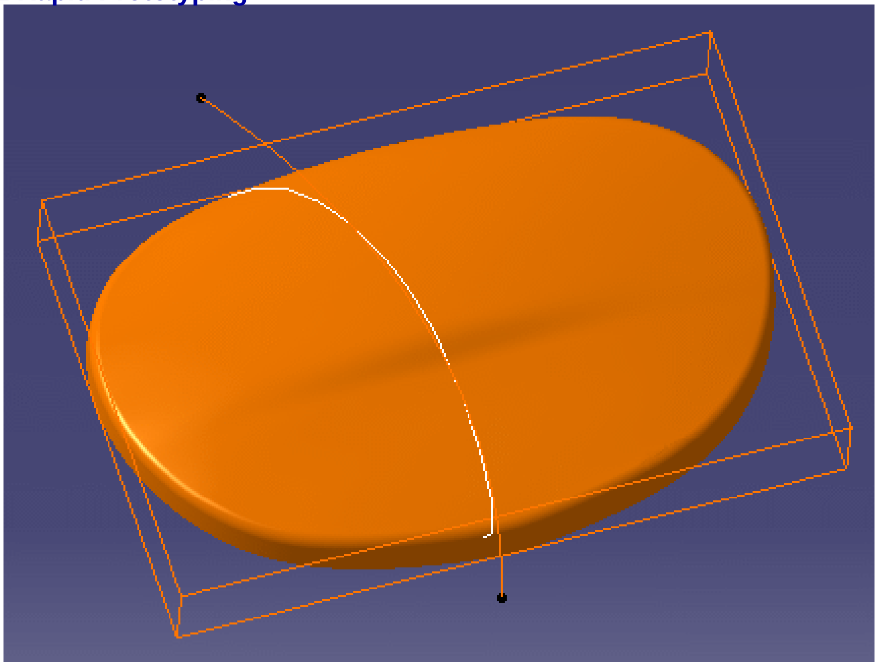
This case defines two areas:



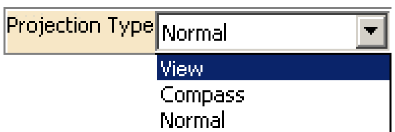
This case defines nine areas:



This case is not valid: the projection curve does not intersect any free edge. In some cases, changing the **Projection type** may solve the problem.



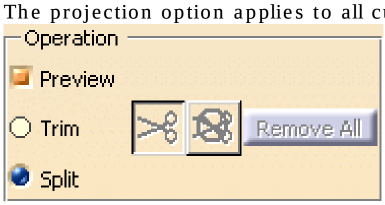
4. Select the **Projection type**



When scans or curves are used as cutting elements, those entities are close to the mesh but not on the mesh. To compute the intersection, it is necessary to project those scans or curves on the mesh to create intersection curves. Three projection options are proposed:

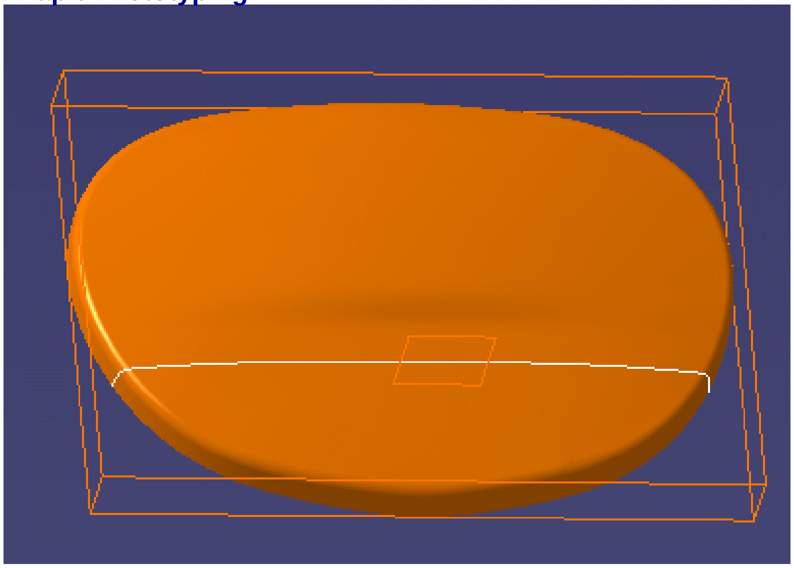
- **View**,
- **Compass**,
- **Normal**.

The projection option applies to all cutting elements.



5. Check **Preview** if you want to see the projection of the cutting elements on the mesh.

   This is an example of the Preview of a plane.



For better performances you should not activate the preview unless absolutely necessary.

6. Decide whether you want to trim or split the mesh:

If you want to split the mesh:

7. Make sure the **Split** option is checked. This makes the **Apply** button available.

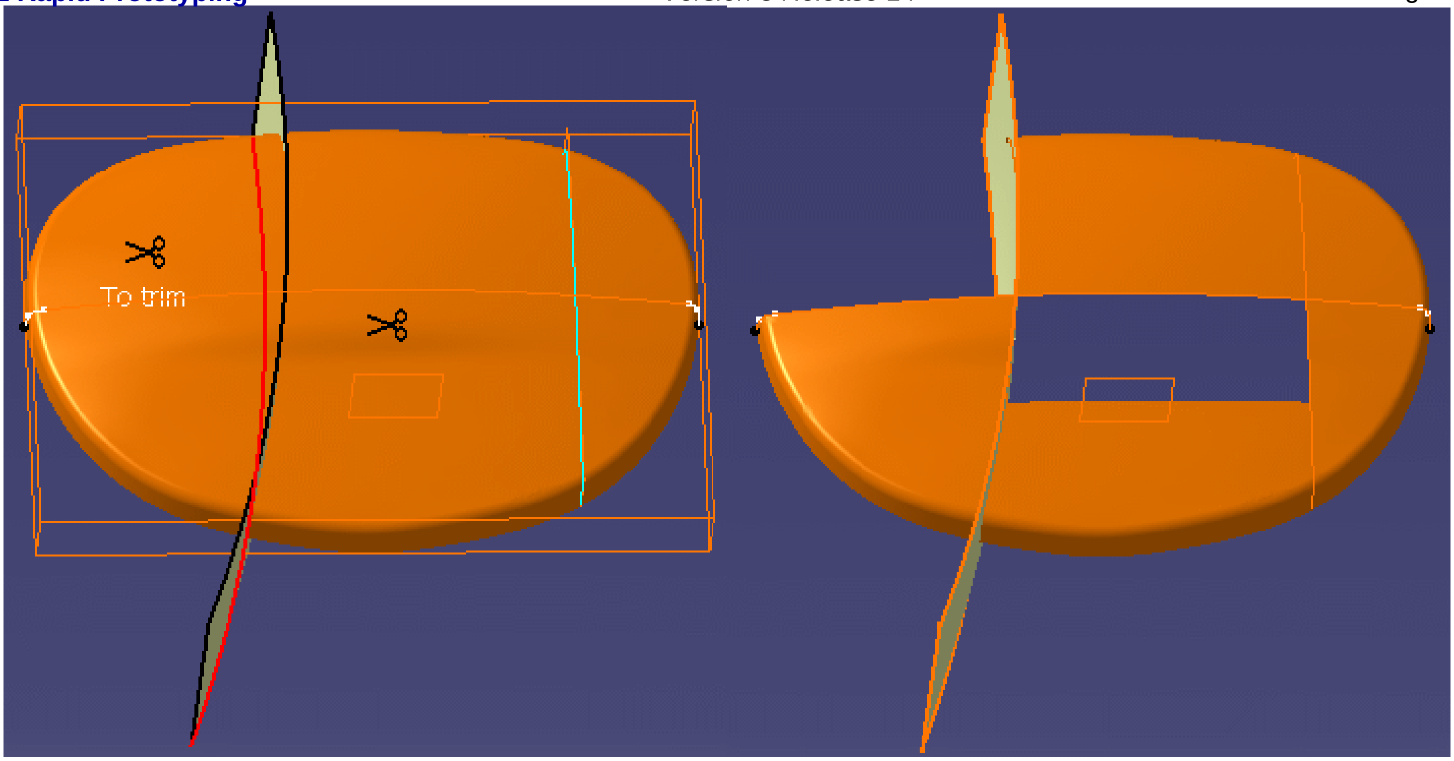
8. Click **Apply**. The action creates several new meshes.

If you want to trim the mesh:

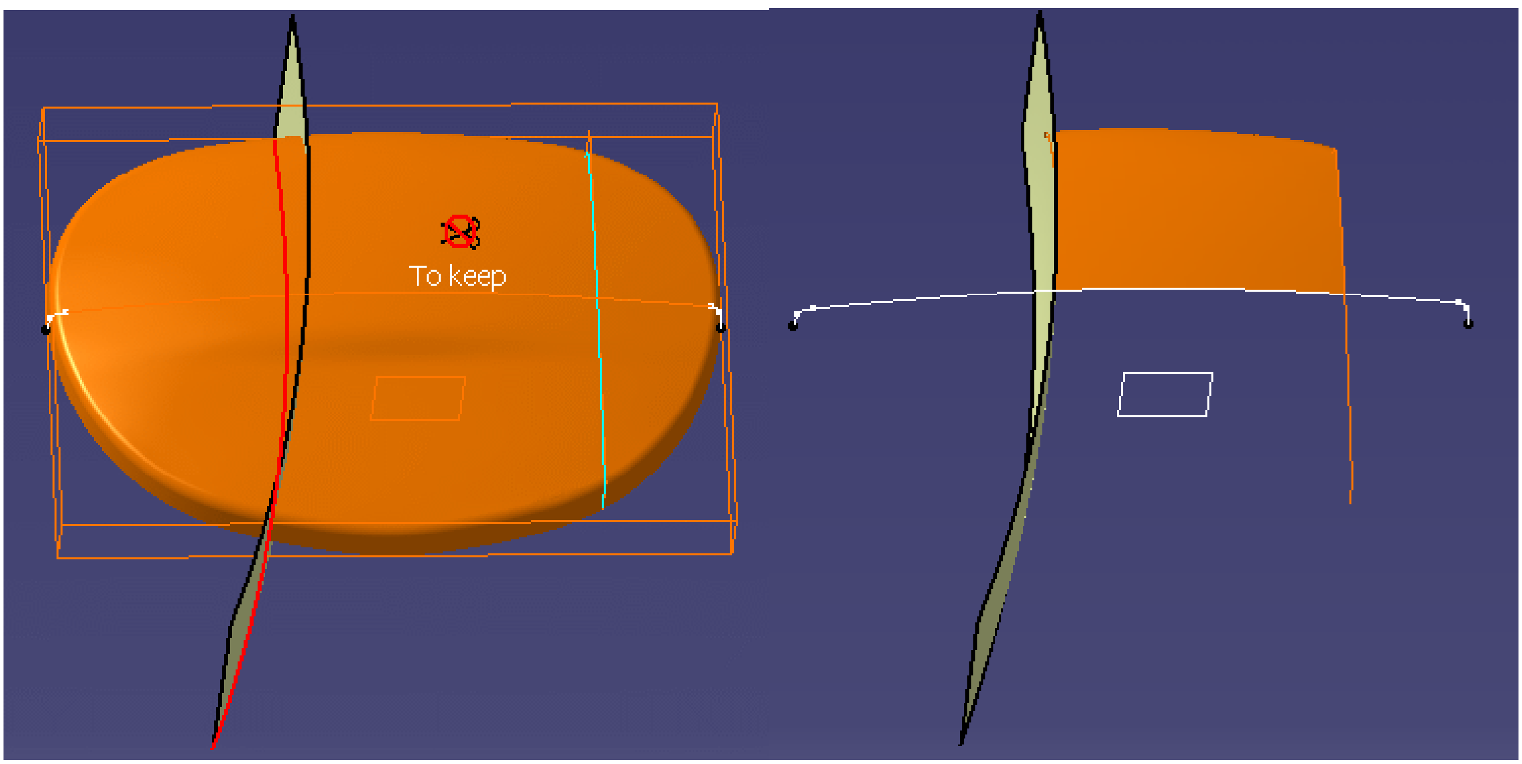
7. Make sure the **Trim** option is checked. This makes the scissors and crossed-scissors available.

8. Use the scissors or crossed-scissors to define the portions to be kept or removed:

Push the scissors button and pick the area(s) you want to remove, or



push the crossed-scissors button and pick the area(s) you want to keep.



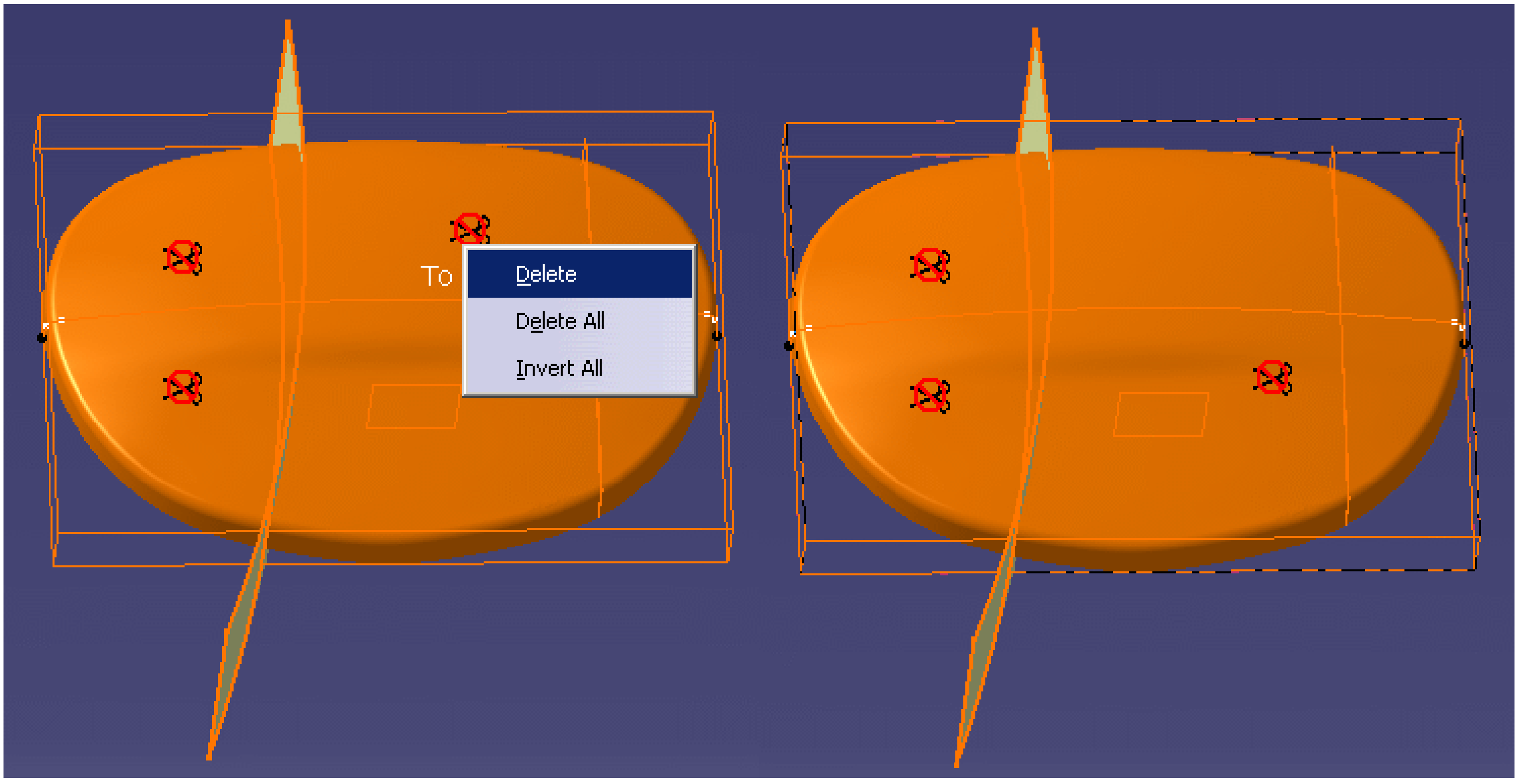
Click a scissors or crossed-scissors icon to delete one occurrence, or use the contextual menu.

Push the **Remove All** button to delete all occurrences, or use the contextual menu.

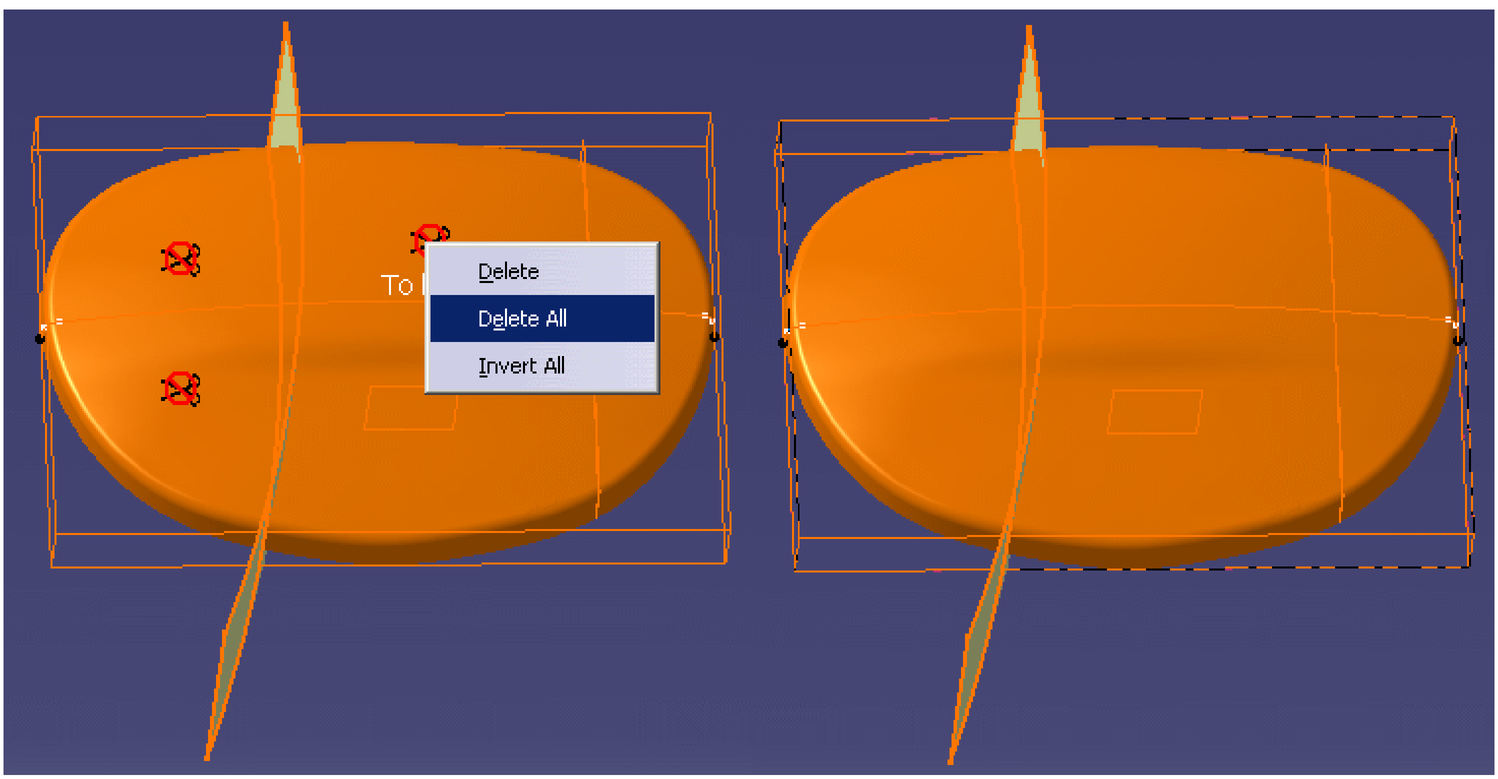
- Unselected areas have the opposite status.
- You can not mix instances of scissors and crossed-scissors.

A contextual menu is available on scissors and crossed-scissors:

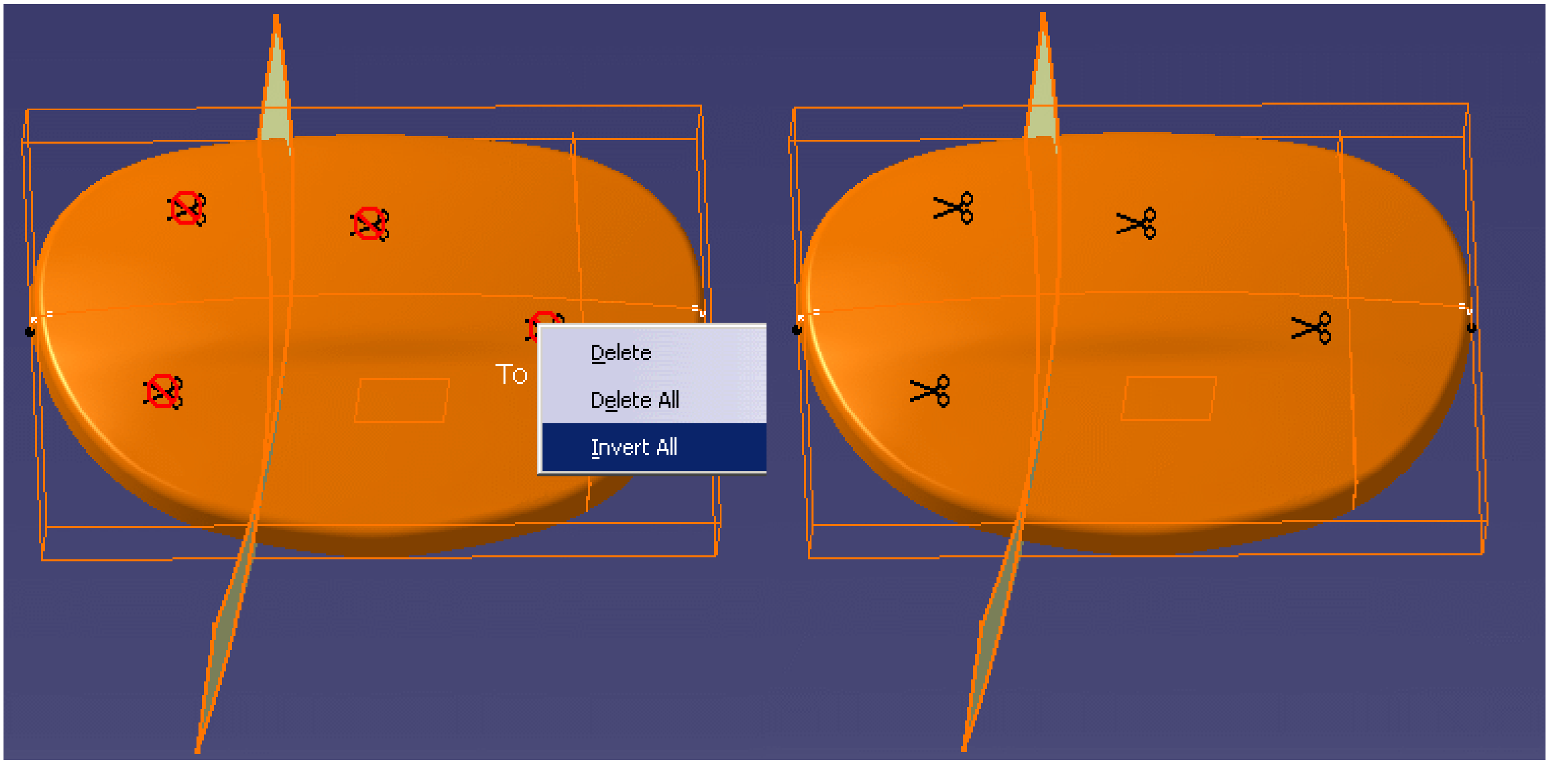
- **Delete**: Deletes the occurrence,



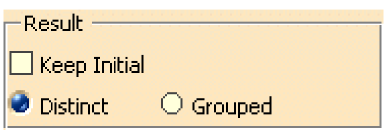
- **Delete All**: Deletes all occurrences,



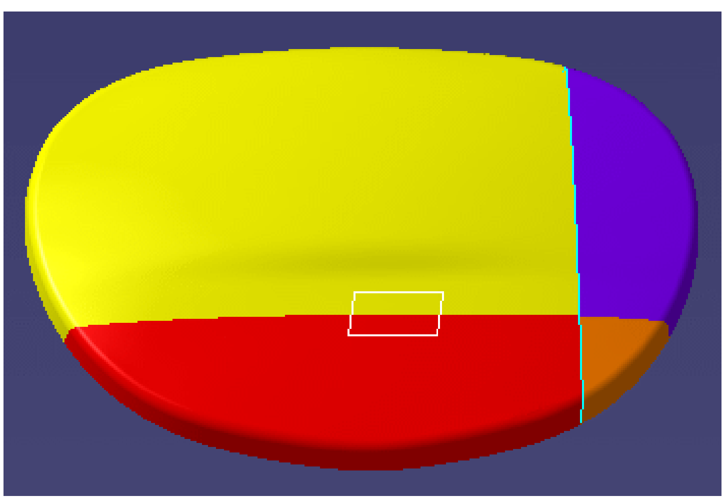
- **Invert All**: Replaces all occurrences of scissors by crossed-scissors and vice-versa.



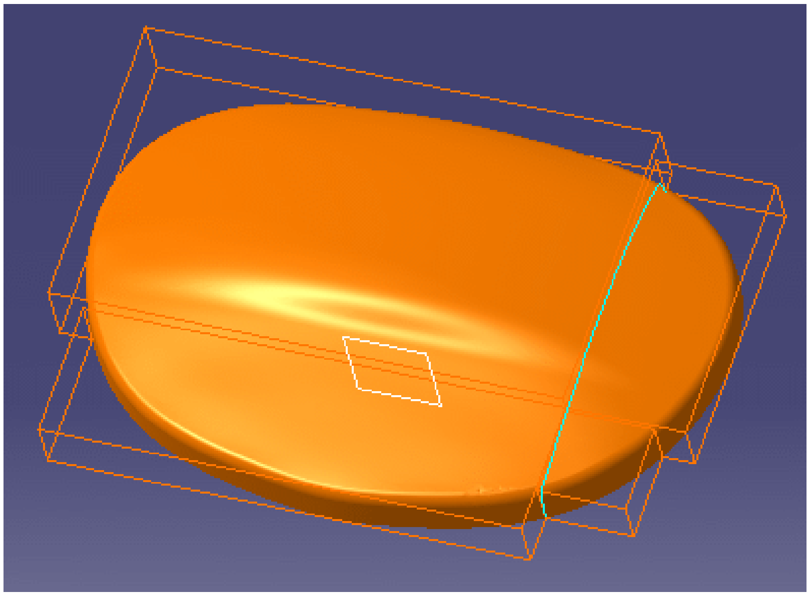
9. Select a Result option:



- if you check **Distinct**, the output meshes will be distinct elements,



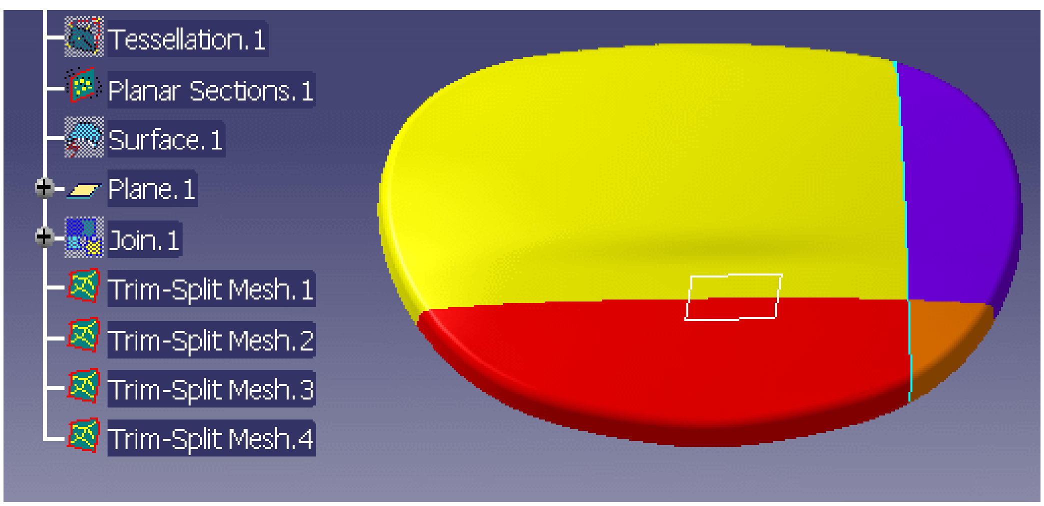
- If you check **Grouped**, the new meshes will be cells grouped in a single body.



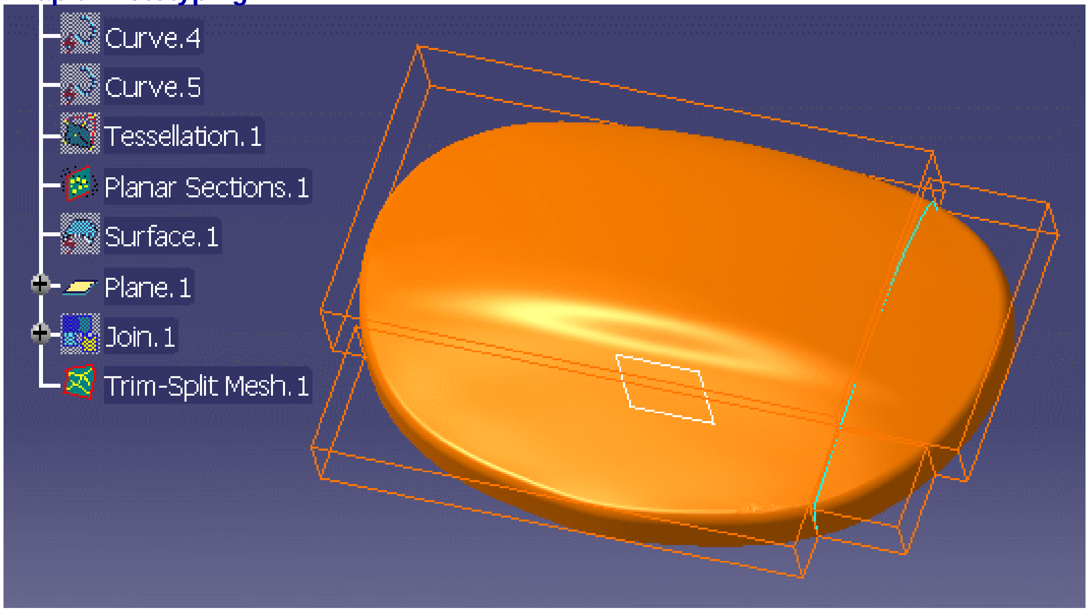
Keep Initial controls whether a new mesh is created in the specification tree or if the input mesh is replaced by another when the command is executed.

If **Keep Initial** is checked:

- the input mesh is sent to the NoShow and remains in the specification tree,
- the output meshes are created in the specification tree and the graphic area:
  - **Distinct** is checked

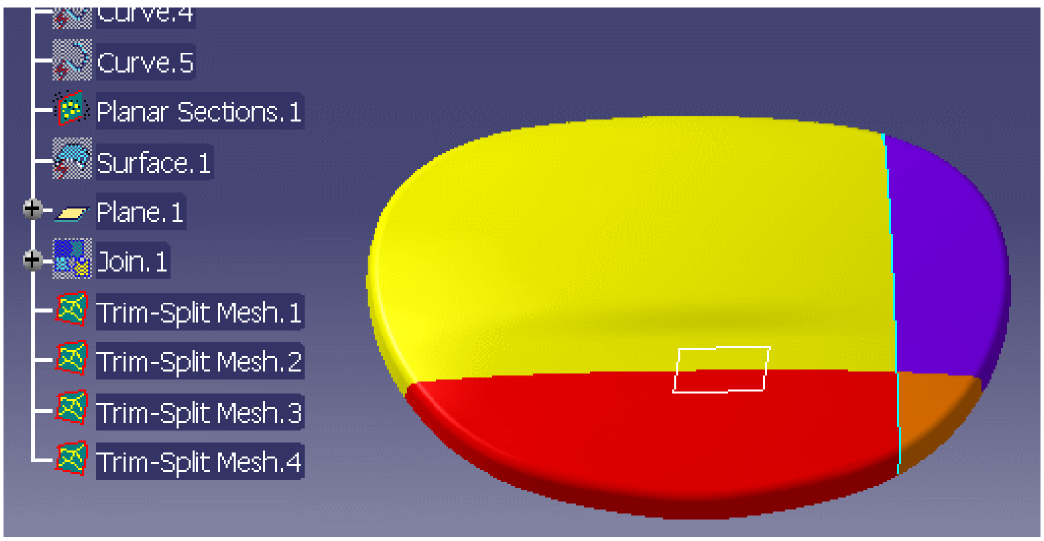


- **Grouped** is checked

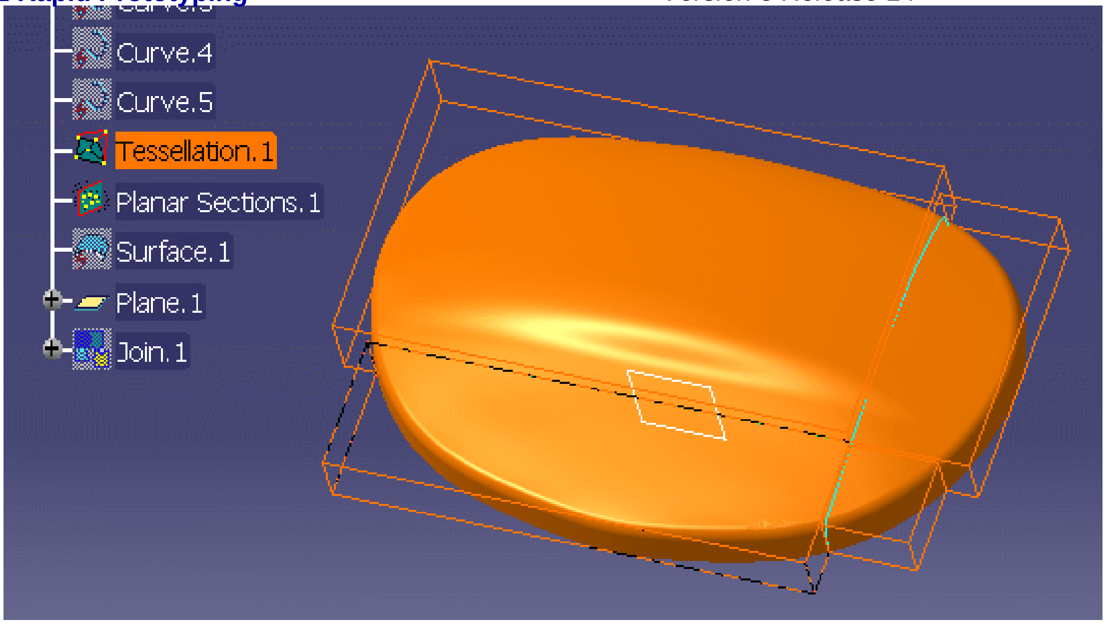


If **Keep Initial** is not checked:

- if **Distinct** is checked, the input mesh is removed from the specification tree, the output meshes are created in the specification tree and the graphic area.



- if **Grouped** is checked, the result multi-cells mesh replaces the input mesh:
  - the input mesh is removed from the graphic area but its name remains in the specification tree,
  - the result multi-cells mesh is created in the graphic area and under the name of the input mesh in the specification tree.



We recommend that you do not keep initial large meshes.

10. Click **Apply** to preview the result.

11. Click **OK** to validate and exit the action.

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.

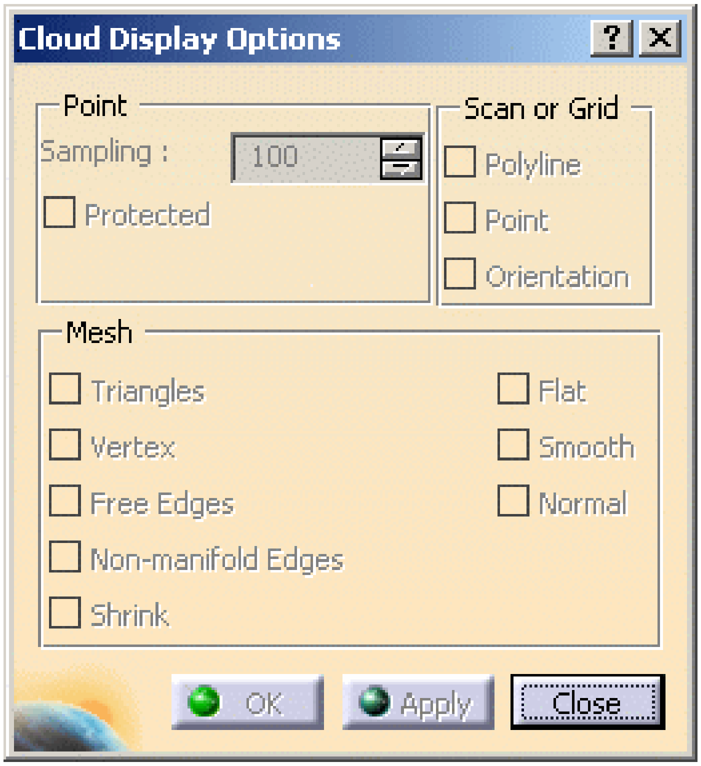
From the **Edit/Properties** menu, you can:

- 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.

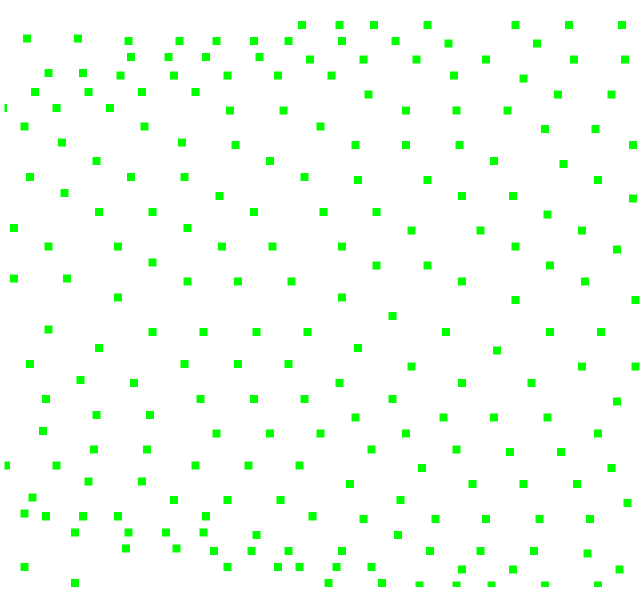


2. Select the cloud to modify. Display options are proposed according to the type of the cloud selected:

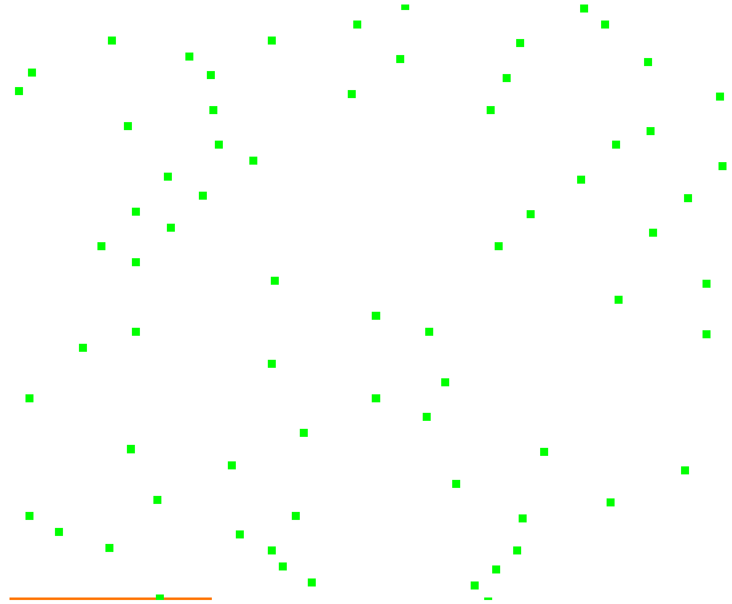
Following options are not yet available:

- 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.



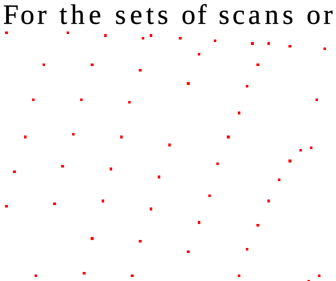
Sampling=100

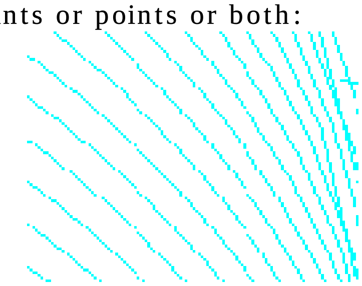


Sampling=25

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

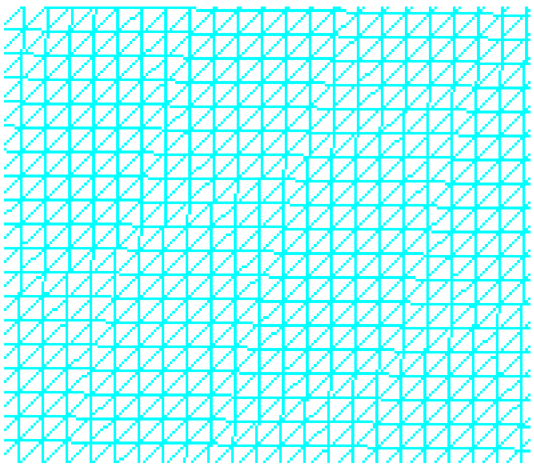
For the sets of scans or grids, you can display them as line of points or points or both:



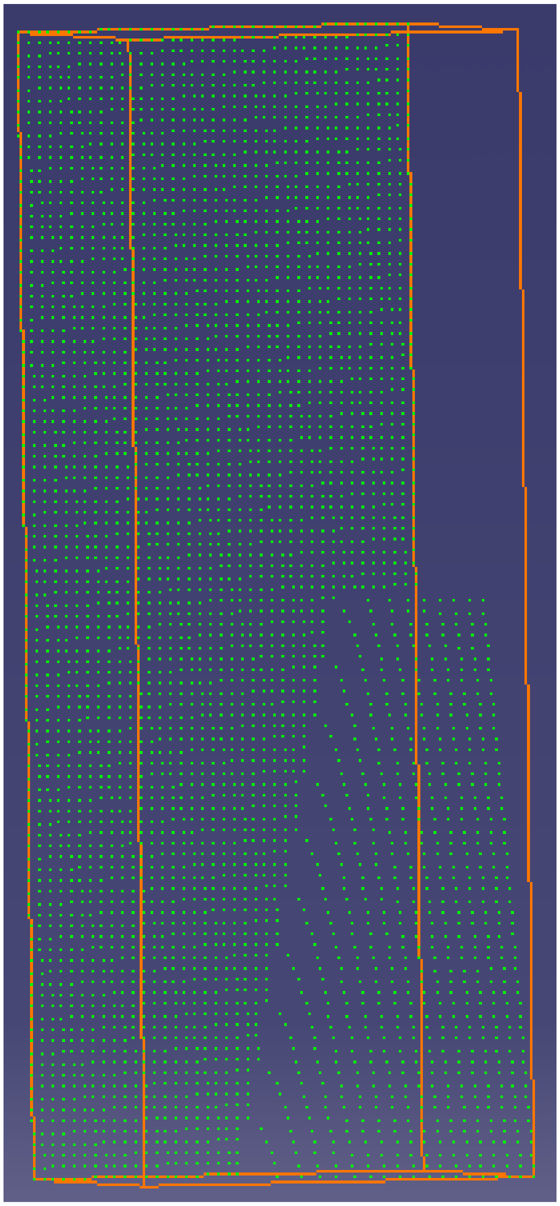
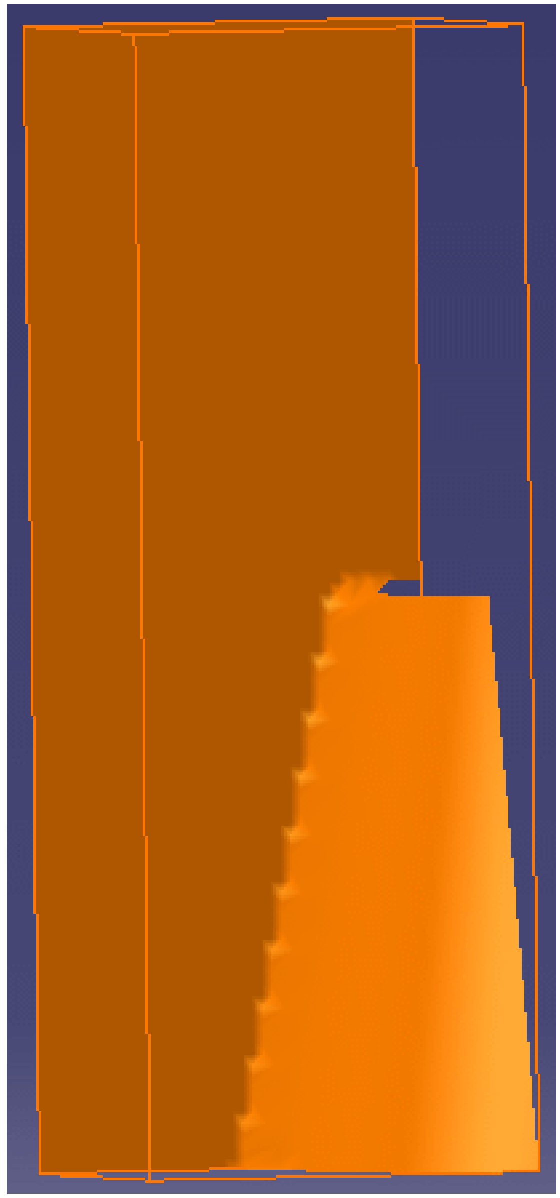


For the mesh, you can:

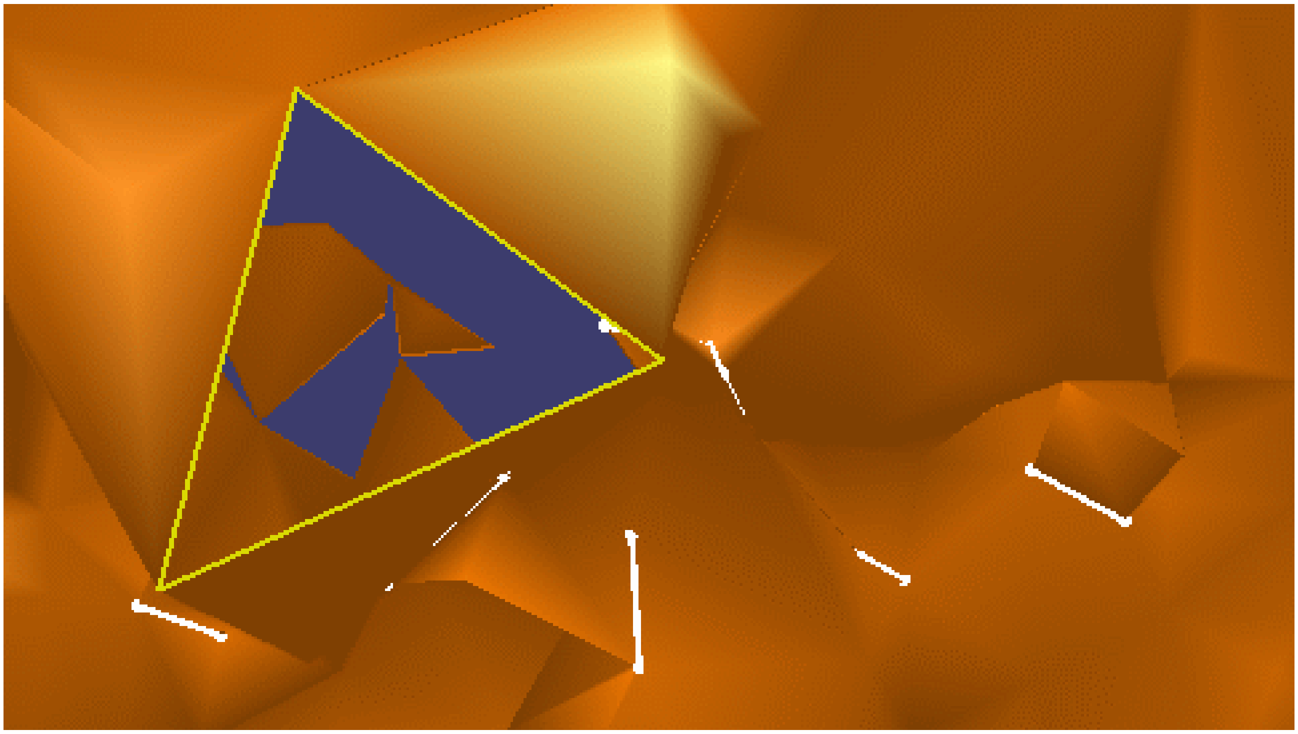
- display the triangles,



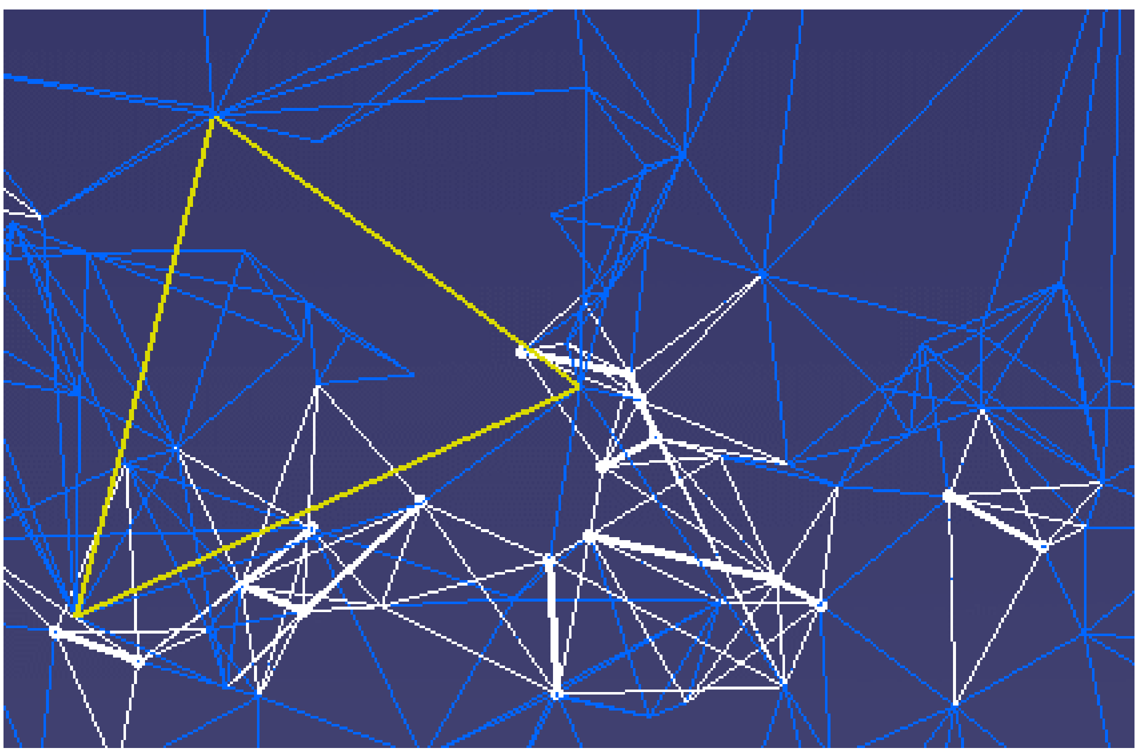
- visualize only the vertices for a lighter display (do not forget to de-activate the Smooth, Flat or Triangles options)

 instead of 

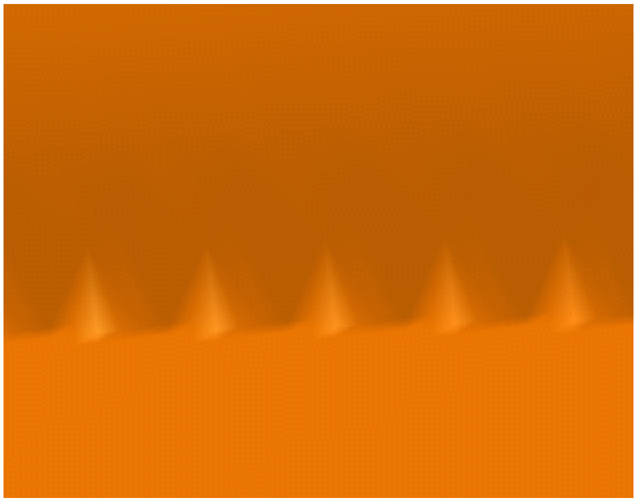
- the free edges in yellow,
- the non-manifold facets and their vertices in bold white lines.



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

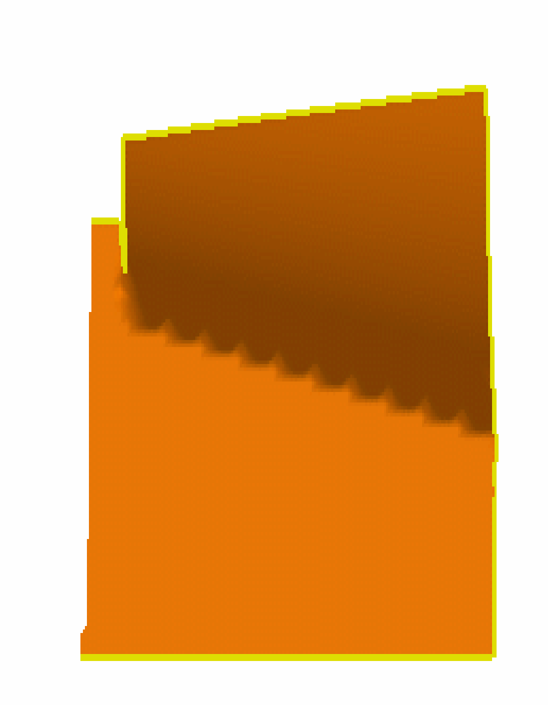


- display the mesh as a smooth or a flat mesh.

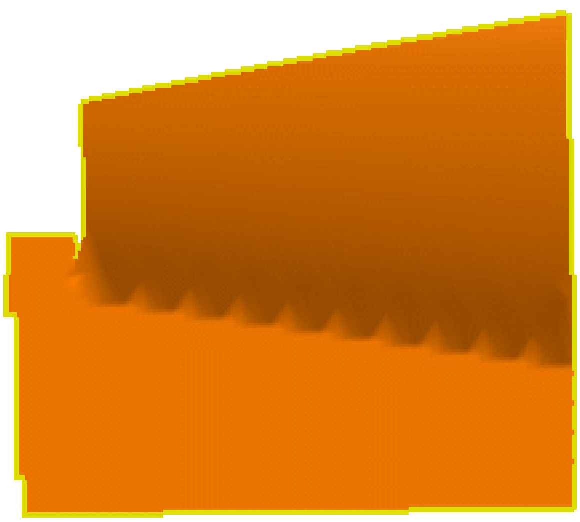
 or 

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.



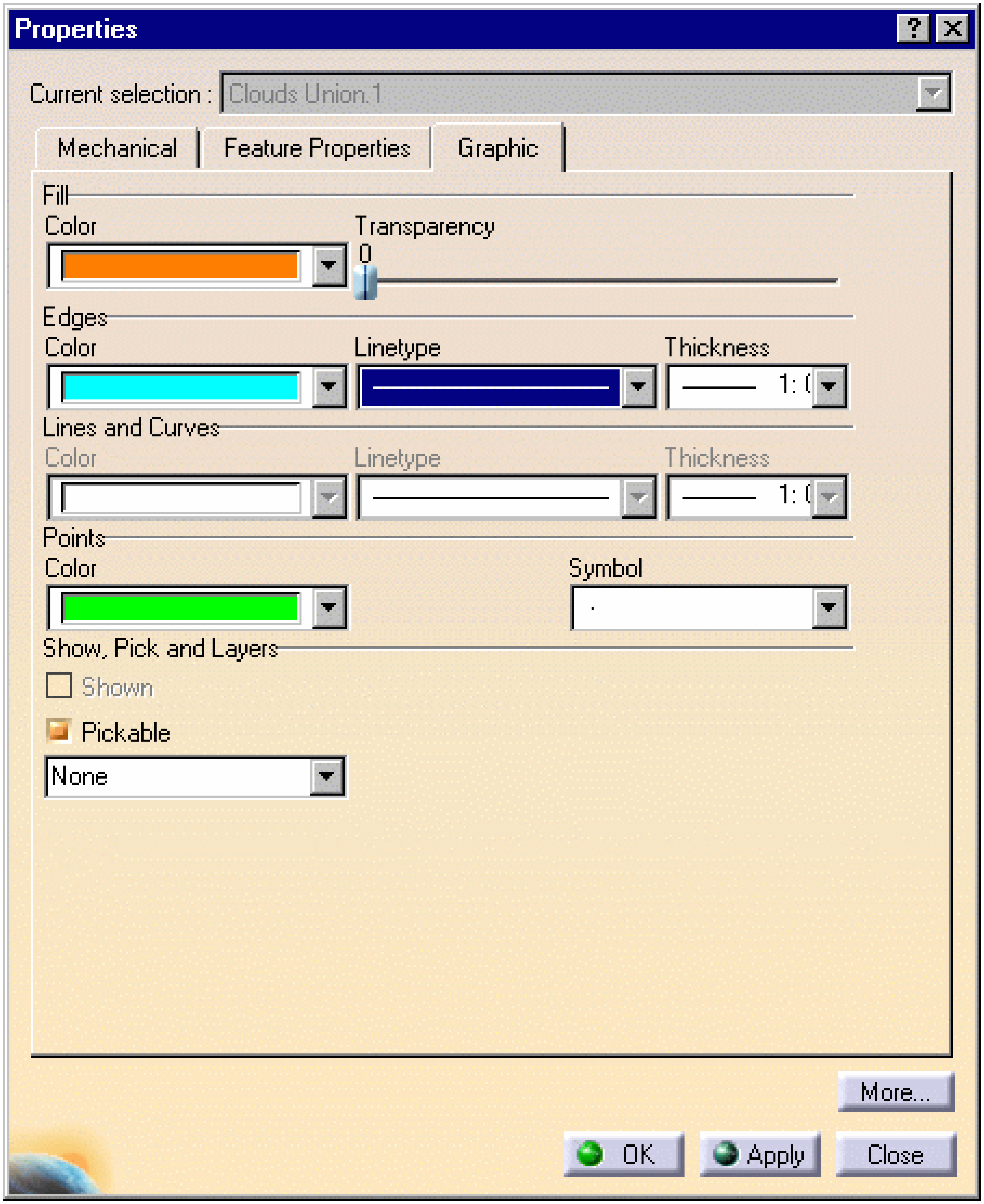
- If you move a cloud of points or a mesh, its graphic display options (not the graphic properties) are lost.
- The display options are not saved in the CATPart while the graphic properties are.

Edit/Properties menu (Graphic tab)

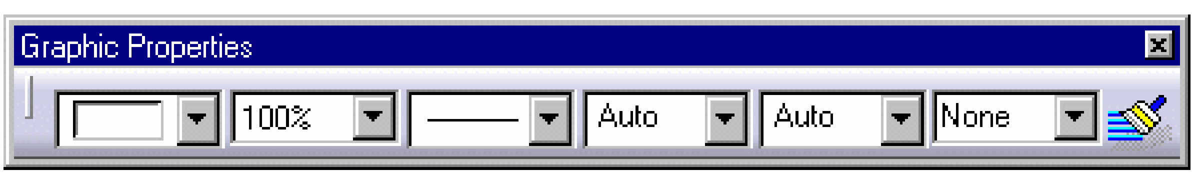
For more information about this menu, please refer to the Displaying and Editing Graphic Properties chapter in the CATIA Infrastructure user's guide.

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**).

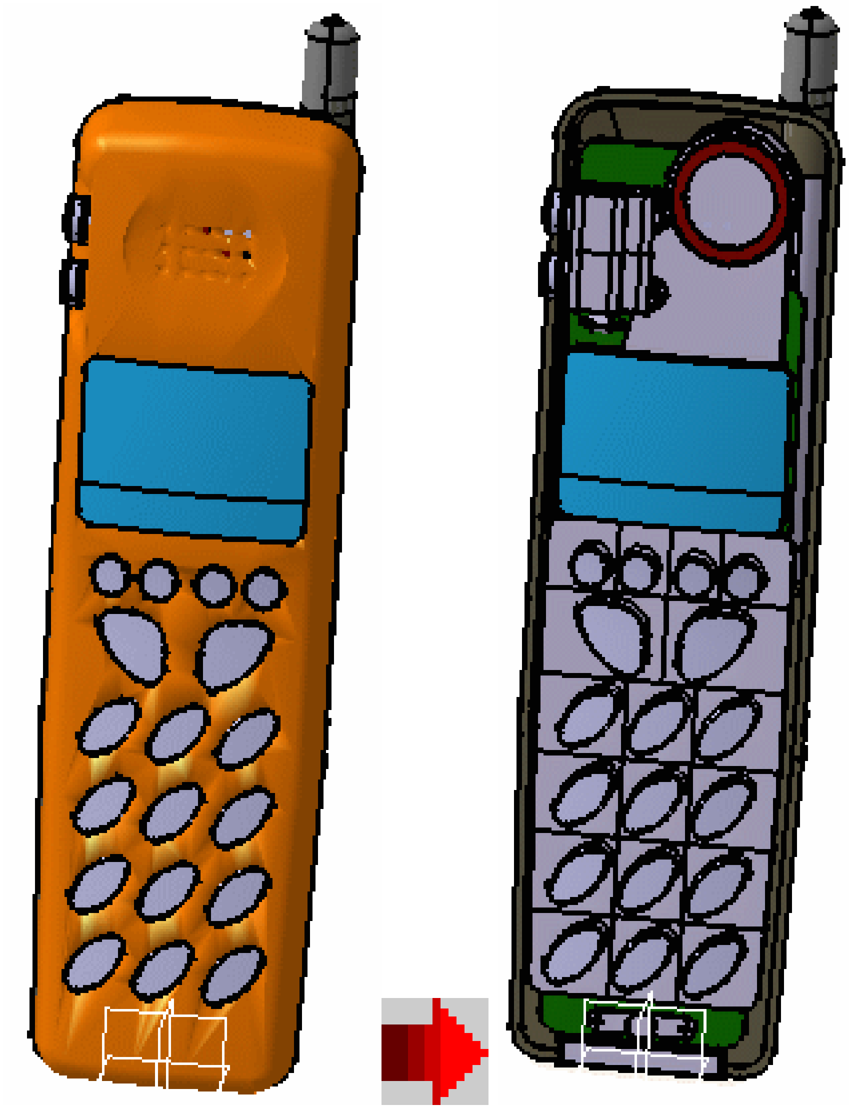


or



- The color displayed in the **Graphic Properties** toolbar applies to meshes only.
- The graphic properties are saved in the CATPart.

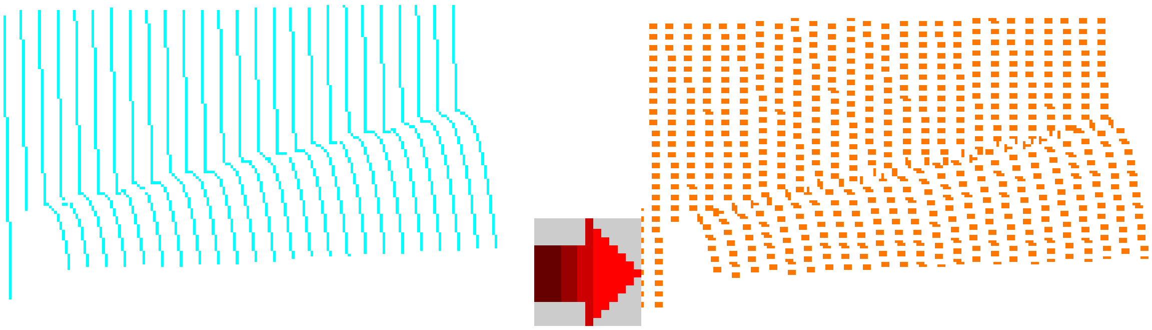
- Use **Fill/Color** and **Transparency** to modify the color and transparency of meshs:



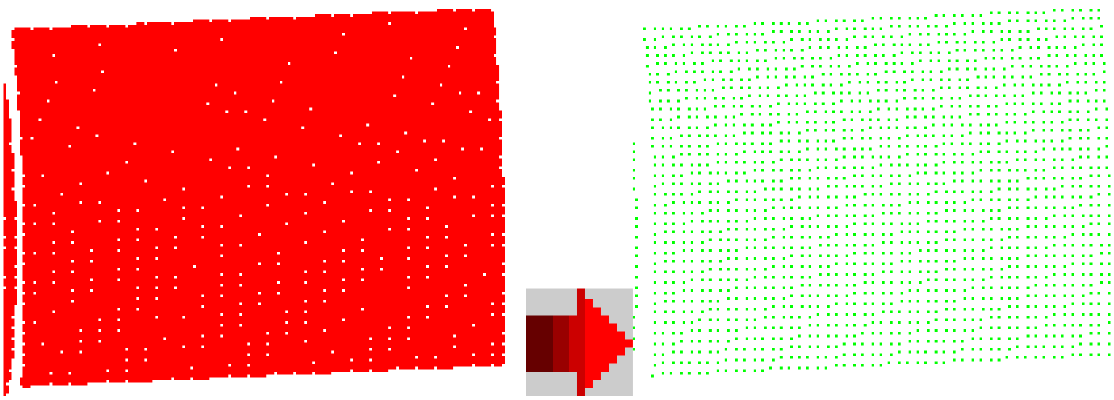
Please note that :
- the color of mesh free edges is yellow, and is not editable,
- the color of non-manifold edges is white, and is not editable,
- the default color of scans has changed to cyan.

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:



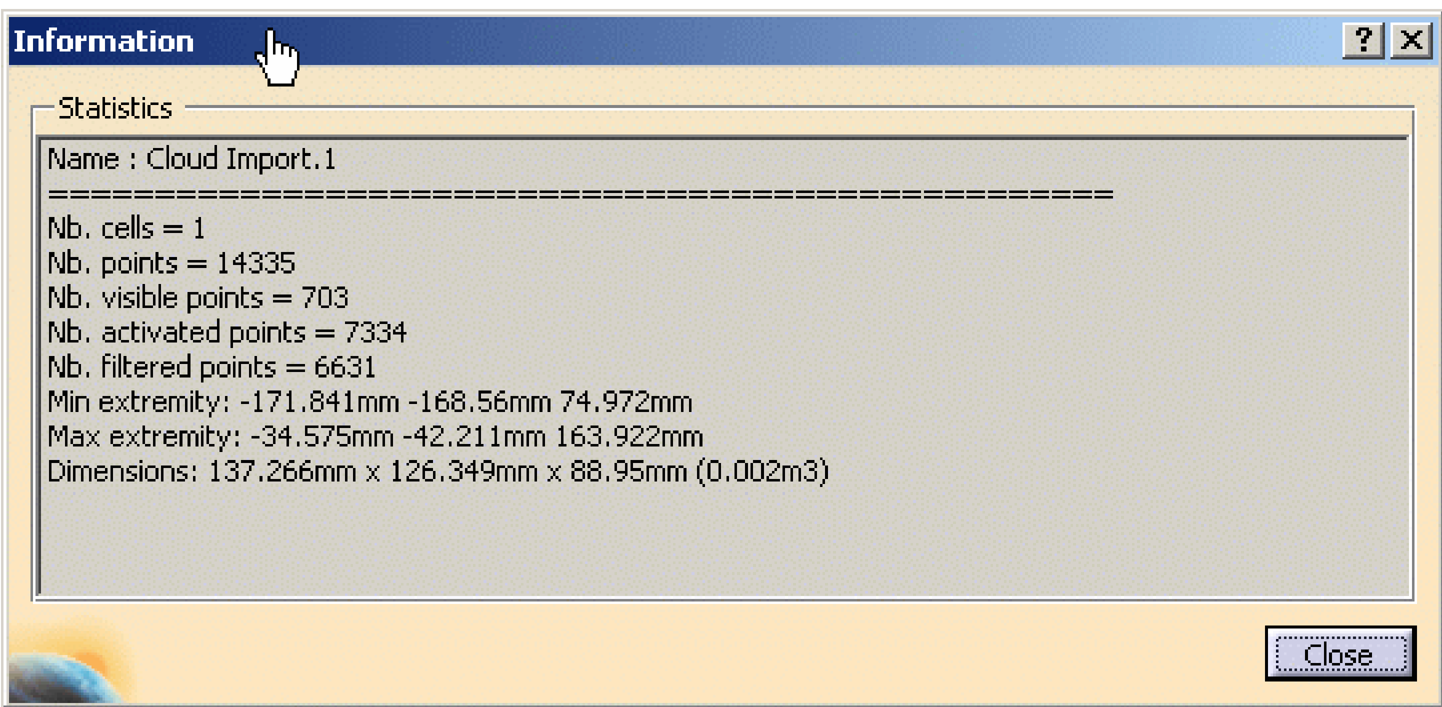
- Use the **Pickable** check box to make an element pickable or not, and choose the pick option in the list below.

Information

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

Open Info1.CATPart from the samples directory.

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



Interoperability

STL Rapid Prototyping complies with the following CATIA V5 standards:

Updating Your Design
Using the Historical Graph
Creating Datums
Points in Generative Shape Design

Updating Your Design

This task explains how and when you should update your design.

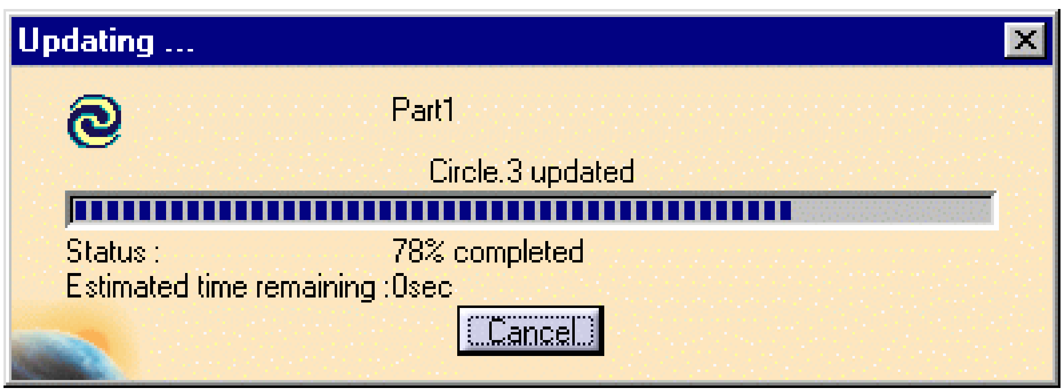
The point of updating your design is to make the application take your last operation into account. Indeed some changes to geometry or a constraint may require rebuilding the part. To warn you that an update is needed, CATIA displays the update symbol next to the part name and displays the corresponding geometry in bright red.
To update a part, the application provides two update modes:

- **automatic update**, available in **Tools -> Options -> Mechanical Design -> Assembly Design -> General** tab. If checked, this option lets the application update the part when needed.
- **manual update**, available in **Tools -> Options -> Mechanical Design -> Assembly Design -> General** tab, it lets you control the updates of your part. You simply need to click the **Update** icon whenever you wish to integrate modifications.

Non-updated wireframe and surface elements are displayed in red.

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

A progression bar indicates the evolution of the operation.



You can cancel the undergoing update by clicking the Cancel button available in the Updating... dialog box.

- Keep in mind that some operations such as confirming the creation of features (clicking **OK**) do not require you to use the update command. By default, the application automatically updates the operation.
- The Update capability is also available via **Edit -> Update** and the **Update** contextual command.
- To update the feature of your choice, just select that feature and use the **Local Update** contextual command.

- Besides the update modes, you can also choose to visualize the update on the geometry as it is happening by checking the **Activate Local Visualization** option from the **Tools -> Options -> Infrastructure -> Part Infrastructure**, **General** tab.

  In this case, as soon as you have clicked the **Update** icon :

  1. the geometry disappears from the screen
  2. each element is displayed as it is updated, including elements in No Show mode. Once they have been updated, they remain in No Show mode.

Interrupting Updates

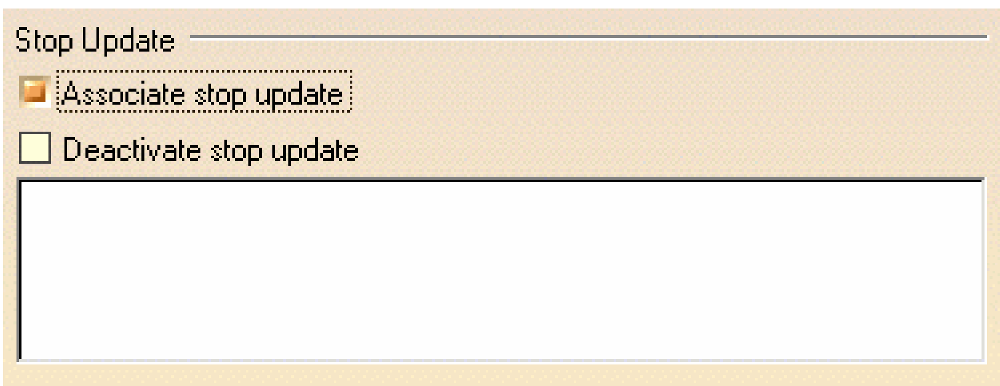
This task explains how to update a part and interrupt the update operation on a given feature by means of a useful message you previously defined.

Open any document containing geometric elements.

1. Right-click an element from the specification tree and choose the Properties contextual menu item.

The Properties dialog box is displayed.

2. From the **Mechanical** tab, check the **Associate stop update** option.



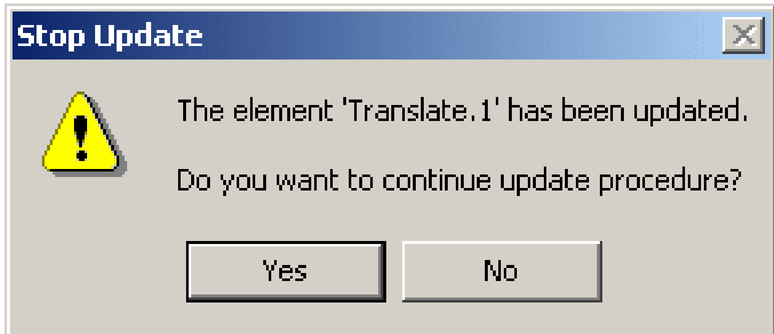
3. Enter the text to be displayed when the updating process will stop when reaching this element.

4. Click OK to confirm and close the dialog box.

The Stop Update.1 feature is displayed in the specification tree, below the element for which it was defined.

5. Whenever it is needed, click the **Update** icon to update the whole part.

The updating process stops after having updated the element selected above, and issues the message as has been defined earlier:



6. Click Yes or No, depending on what you intend to do with the geometry created based on the selected element.

Would you no longer need this capability, you can:

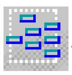
- right-click the element for which the stop was defined, choose the Properties contextual command and check the Deactivate stop update option from the Mechanical tab: the update will no longer at this element.
  You notice that when the capability is deactivated, the Stop Update icon changes to: in the specification tree.
- right-click Stop Update.1 from the specification tree, and choose the **Delete** contextual command.

Using the Historical Graph

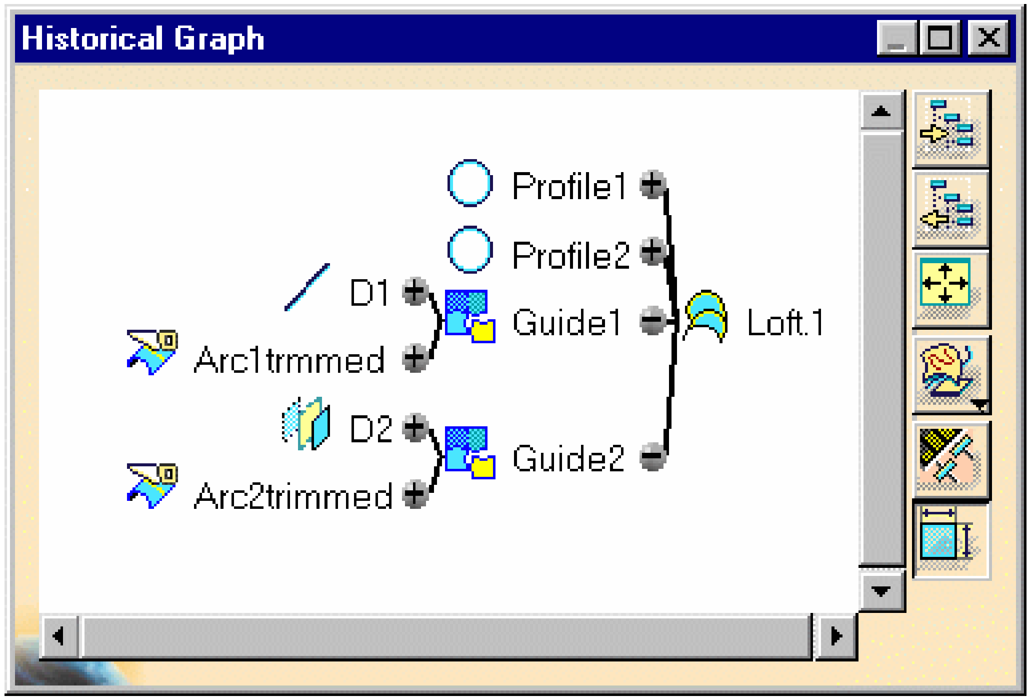
This command is only available with the Generative Shape Design 2 product.

This task shows how to use the Historical Graph.

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

2. Click the **Show Historical Graph** icon .

The Historical Graph dialog box appears.



The following icon commands are available.

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

3. Just close the dialog box to exit this mode.

Creating Datums

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

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

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

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

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

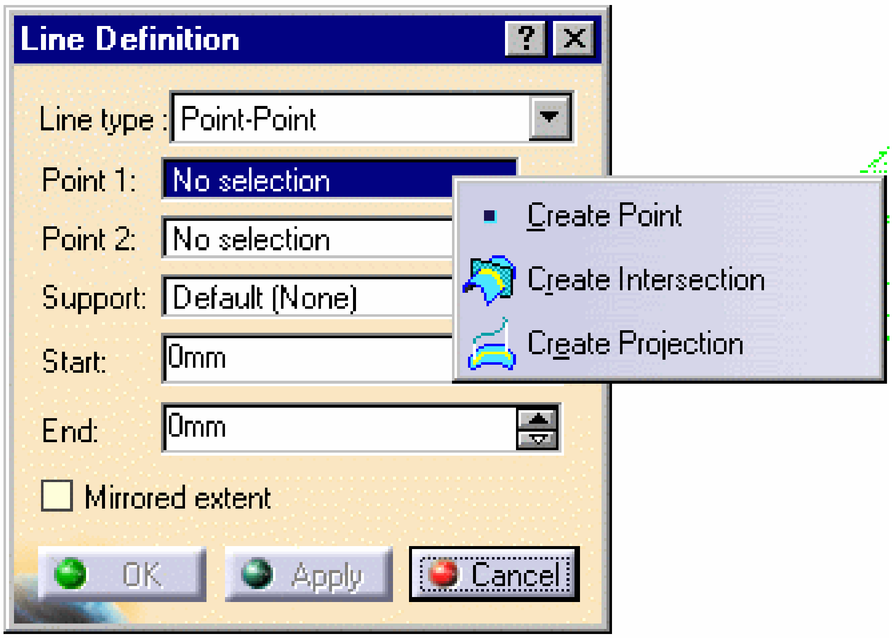
The History mode (active or inactive) will remain fixed from one session to another: it is in fact a setting.

Points in Generative Shape Design

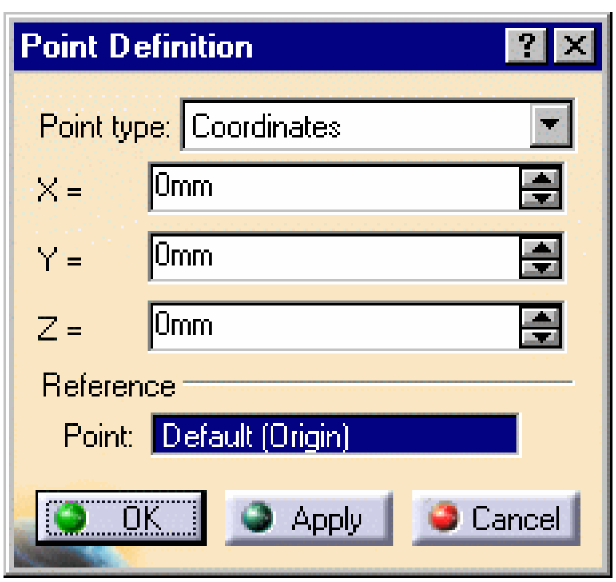
This task shows you how to use points from a cloud of points in Generative Shape Design.

Open the Cloud1.CATPart from the samples directory. We used the **Line Definition** for our example, but the operating mode is the same for all creation action requiring points.

1. Select the type of creation from the combo list of the dialog box.
2. Go to the next Point field and choose **Create Point** from the contextual menu.



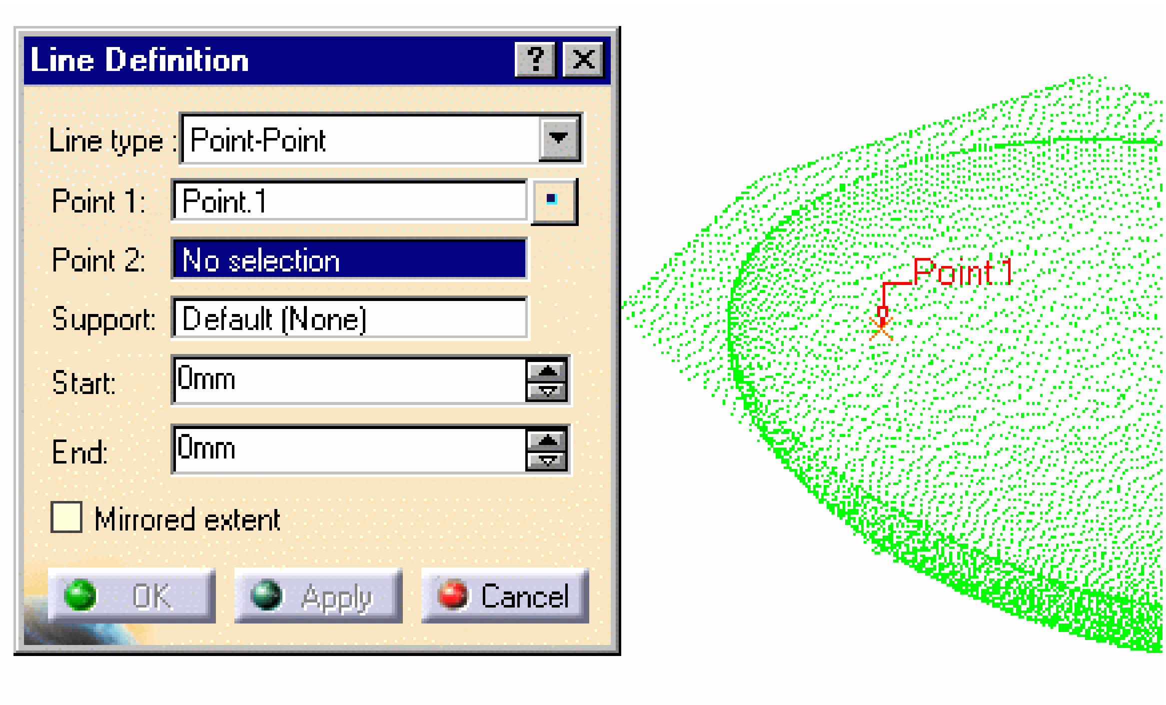
3. The **Point Definition** dialog box is displayed.



4. Click a point on the cloud. Its coordinates are displayed in the **Point Definition** dialog box. Click **OK** to confirm the creation of this point.

5. In the main dialog box, go to the next **Point** field and repeat the above steps as many times as necessary.

6. If necessary, push the **Point** icon on the right of the Point field to modify the point you have created: the **Point Definition** dialog box is displayed and updated according to your pick on the cloud.



Managing Geometrical Sets

Geometrical sets enable to gather various features in a same set or sub-set and organize the specification tree when it becomes too complex or too long. You can put any element you wish in the geometrical set, it does not have to be structured in a logical way. The order of these elements is not meaningful as their access as well as their visualization is managed independently and without any rule.

This task shows how to manage geometrical sets within the specification tree. This involves:

- inserting a geometrical set
- removing a geometrical set
- changing body
- sorting the contents of a geometrical set
- reordering components

You will find other useful information in the Managing Groups and Hiding/Showing chapters.

- You can insert and manipulate geometrical sets in the specification tree in much the same way as you manage files in folders.
- These management functions have no impact on the part geometry.
- You should refer to the Copying and Pasting section for information about how geometrical sets can be used in a part edition context.
- When loading the Generative Shape Design workbench, a Geometrical Set automatically becomes the current body.
  This also means that only the results of the Hybrid Body, i.e. the result of all the operations performed on geometry, is visible and not any intermediate state of the Hybrid Body.
- You can define the Generative Shape Design feature that is to be seen when working with another application, such as Generative Structural Analysis for example.

To do this, while in the Generative Shape Design workbench:

1. Choose the **Tools -> External View...** menu item
   The External View dialog box is displayed.
2. Select the element belonging to a Geometrical Set that should always been seen as the current element when working with an external application.
3. Click OK in the dialog box.

The selected element will be the visible element in other applications, even if other elements are created later in the .CATPart document, chronologically speaking.

To check whether an external view element has already been specified, choose the **Tools -> External View...** menu item again. The dialog box will display the name of the currently selected element. This also allows you to change elements through the selection of another element. Note that you cannot deselect an external view element and that only one element can be selected at the same time.

Open any .CATPart document containing Geometrical Sets.
You can also open the GeometricalSets2.CATPart document.

Inserting a Geometrical Set

1. In the specification tree, select an element as the location of the new geometrical set.

   This element will be considered as a child of the new geometrical set and can be a geometrical set or a feature.

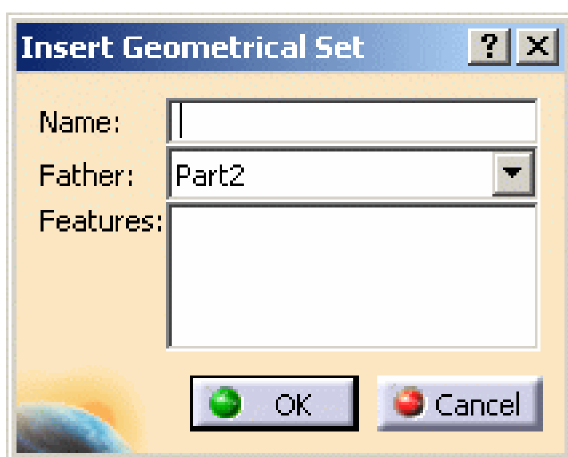
2. Select the **Insert -> Geometrical Set** menu command.

   The Insert Geometrical Set dialog box is displayed.

   The Features list displays the elements to be contained in the new geometrical set.

3. Enter the name of the new geometrical set.

4. Use the Father drop-down list to choose the body where the new geometrical set is to be inserted. All destinations present in the document are listed allowing you to select one to be the father without scanning the specification tree. They can be:
   - geometrical sets
   - parts



5. Select additional entities that are to be included in the new geometrical set.

If all selected entities belong to the same geometrical set, the father of the new geometrical set is automatically set to the father of these entities.

6. Click OK to create the geometrical set at the desired location.

The result is immediate. CATIA displays this new Geometrical Set.x, incrementing its name in relation to the pre-existing bodies, in the specification tree. It is created after the last current geometrical set and is underlined, indicating that it is the active geometrical set. The next created element is created within this geometrical set.

- You cannot create a geometrical set within an ordered geometrical set and vice versa.
- This Insert Geometrical Set dialog box is only available with the Generative Shape Design 2 product.

You can check the **Create a Geometrical Set when creating a new part** option in **Tools -> Options -> Infrastructure -> Part Infrastructure -> Part Document** tab if you wish to create a geometrical set as soon as you create a new part. For more information about this option, please refer to the Customizing section of the *Part Design User's Guide*.

Removing a Geometrical Set

Two methods are available:

- If you want to delete the geometrical set and all its contents:

1. Right-click the geometrical set then select the **Delete** contextual command.

- If you want to delete the geometrical set but keep its contents:
  This is only possible when the father location of the geometrical set is another geometrical set. This is not possible when the father location is a root geometrical set.

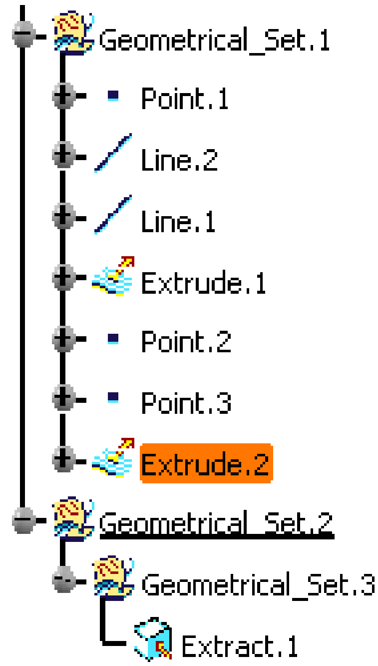
1. Right-click the desired geometrical set then select the **Geometrical Set.x object -> Remove Geometrical Set** contextual command.

The geometrical set is removed and its constituent entities are included in the father geometrical set.

You cannot delete a feature within a geometrical set created on the fly. Indeed this geometrical set is considered as private and can only be deleted globally.

Moving a Geometrical Set to a New Body

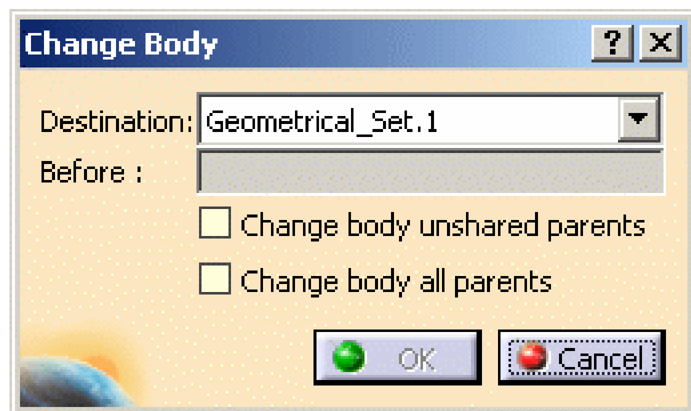
1. From the specification tree, select the element then choose the **Geometrical Set.object -> Change Geometrical Set...** item from the contextual menu.



Multi-selection of elements of different types is supported. However, note that in this case, the contextual menu is not available, and that you can access this capability using the **Edit** menu item.

The Change Body dialog box is displayed.

The list of destinations is alphabetically sorted.

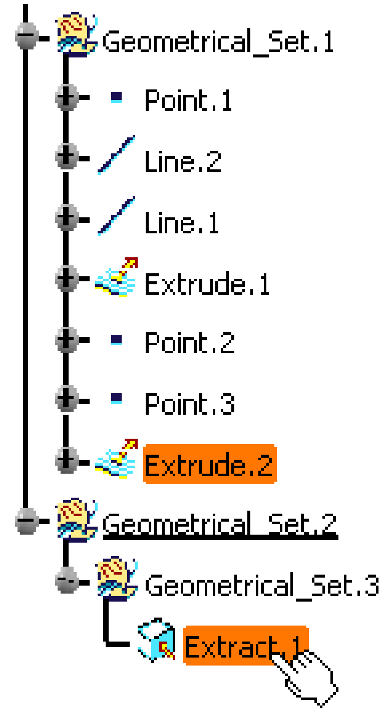


2. Select the **Destination** body where the geometrical set is to be located.

   Here we selected Geometrical_Set.3.

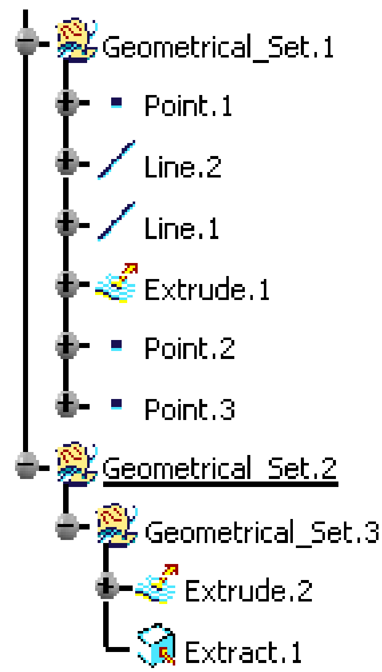
You can do so by selecting the Body in the specification tree, or using the drop-down list from the dialog box.

By default, if you select a body, the geometrical set is positioned last within the new body. However, you can select any element in the new body, before which the moved geometrical set will be located.



3. Select the element above which the one you already selected is to be inserted.

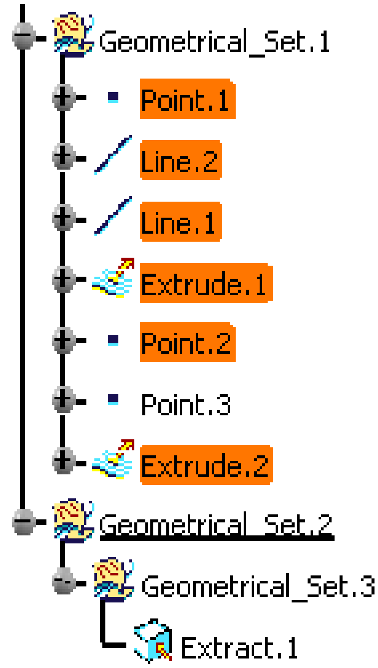
You can directly select this positioning element. In this case the **Destination** field of the Change Body dialog box is automatically updated with the Body to which this second element belongs.



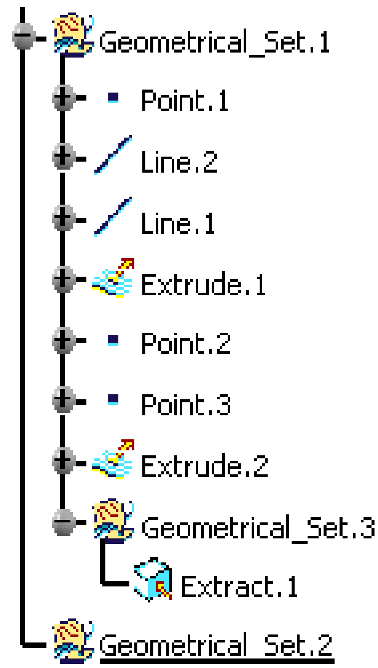
4. Click OK to move the geometrical set to the new body.

The element selected first is moved to its new location in the specification tree, but geometry remains unchanged.

- Check the **Change body unshared parents** option to move all parents of the first selected element to its new location, provided these parents are not shared by any other element of the initial body.
  In this case, all the unshared parents are highlighted prior to the move.



- Check the **Change body all parents** option to move all parents of the first selected element to its new location, regardless of whether these parents are used (shared) by any other element of the initial body.
  In this case, all the parent elements are highlighted prior to the move.

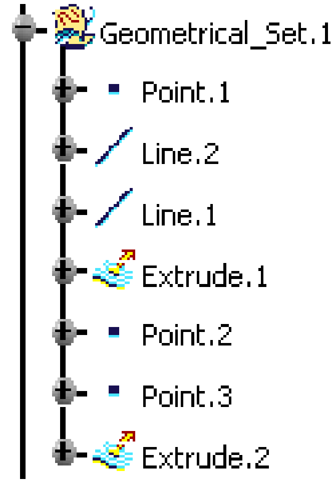


- You can move a whole branch, i.e. a whole body and its contents, at a time.
  Here we moved Geometrical_Set.3 last in Geometrical_Set.1.

Sorting the Contents of a Geometrical Set

You may need to sort the contents of a Geometrical Set, when the geometric elements no longer appear in the logical creation order. In that case, use the Auto-sort capability to reorder the Geometrical Set contents in the specification tree (geometry itself is not affected).

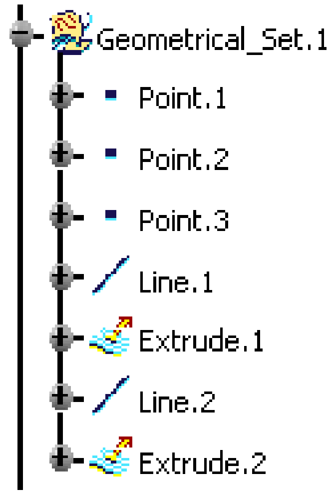
The Geometrical_Set.1 contains two extruded surfaces based on point-point lines. The specification tree looks like this:



1. Right-click the Geometrical_Set.1 from the specification and choose the **Geometrical_Set.1 object -> AutoSort** command.

Instantly, the contents of the Geometrical Set are reorganized to show the logical creation process.

The geometry remains unchanged.



Reordering Components within a Geometrical Set

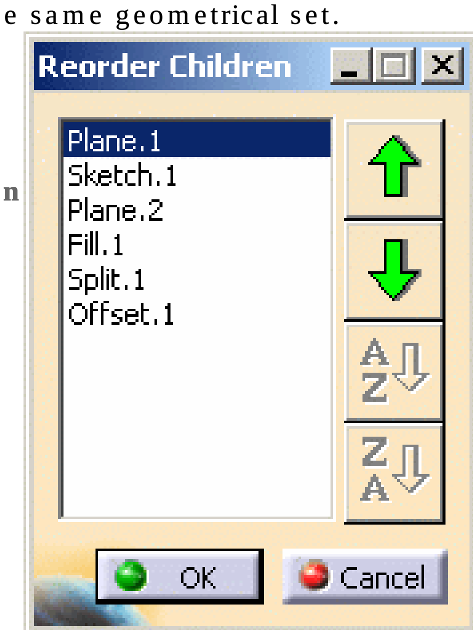
This capability enables you to reorder elements inside the same geometrical set.

1. Right-click the Geometrical_Set.1 from the specification tree and choose the **Ordered Geometrical Set.1 object -> Reorder Children** command.

   The Reorder Children dialog box is displayed.

2. Select an element.

3. Use the arrows to move an element up or down.



Reordering Features

The Reorder command allows you to move a feature in a Geometrical Set. These features can be:

- solids
- shape features
- sketches

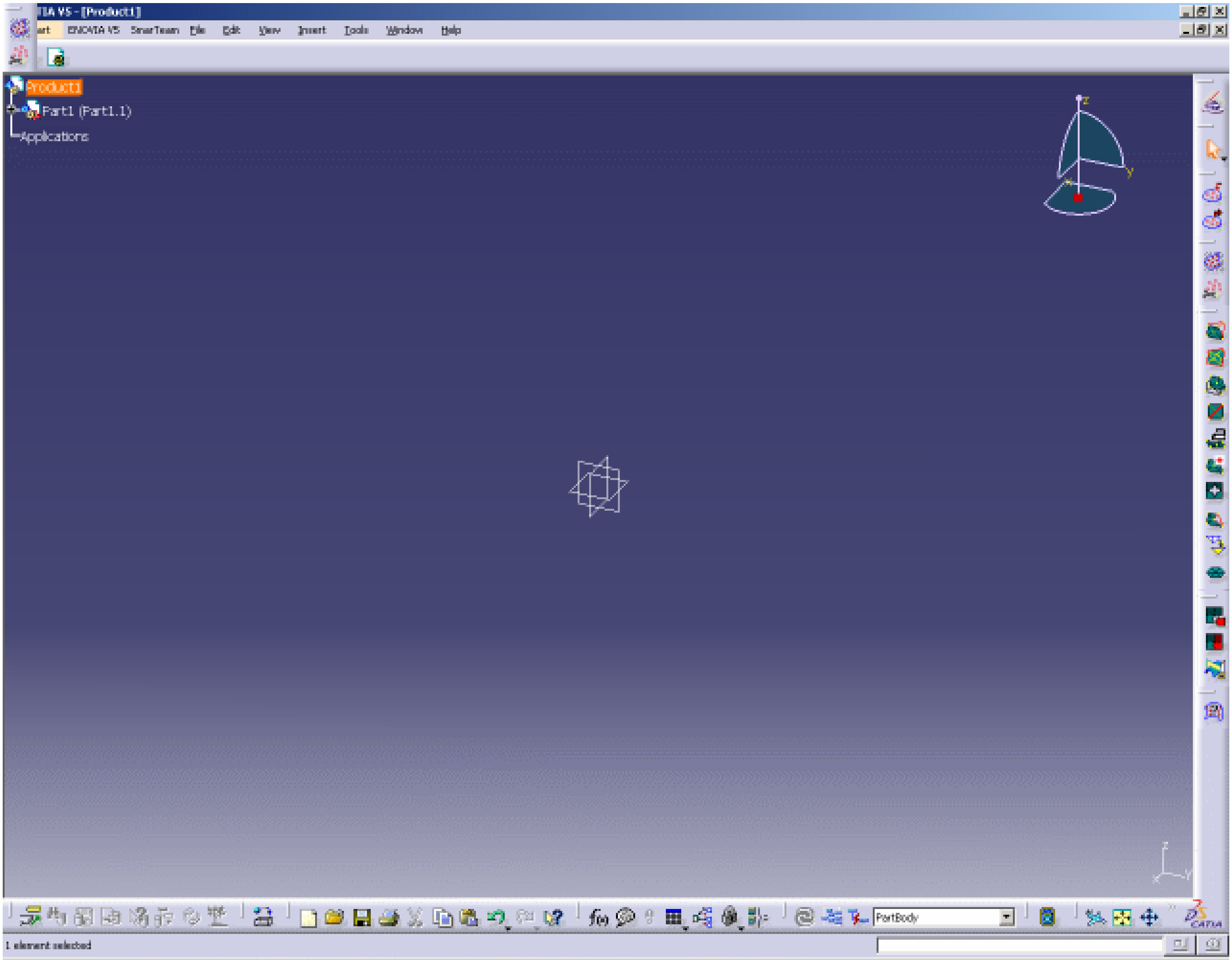
Replacing Features

This capability is only available on shape features.

Please refer to the Replacing or Moving Elements chapter in the *Part Design User's Guide*.
To manage this capability, the **Do replace only for elements situated after the In Work Object** option is available in **Tools -> Options -> Part Infrastructure -> General** tab. It allows you to make the Replace option possible only for features located below the feature in Work Object and in the same branch.

Workbench Description

This chapter describes the menus, sub-menus, items and toolbars of the workbench.

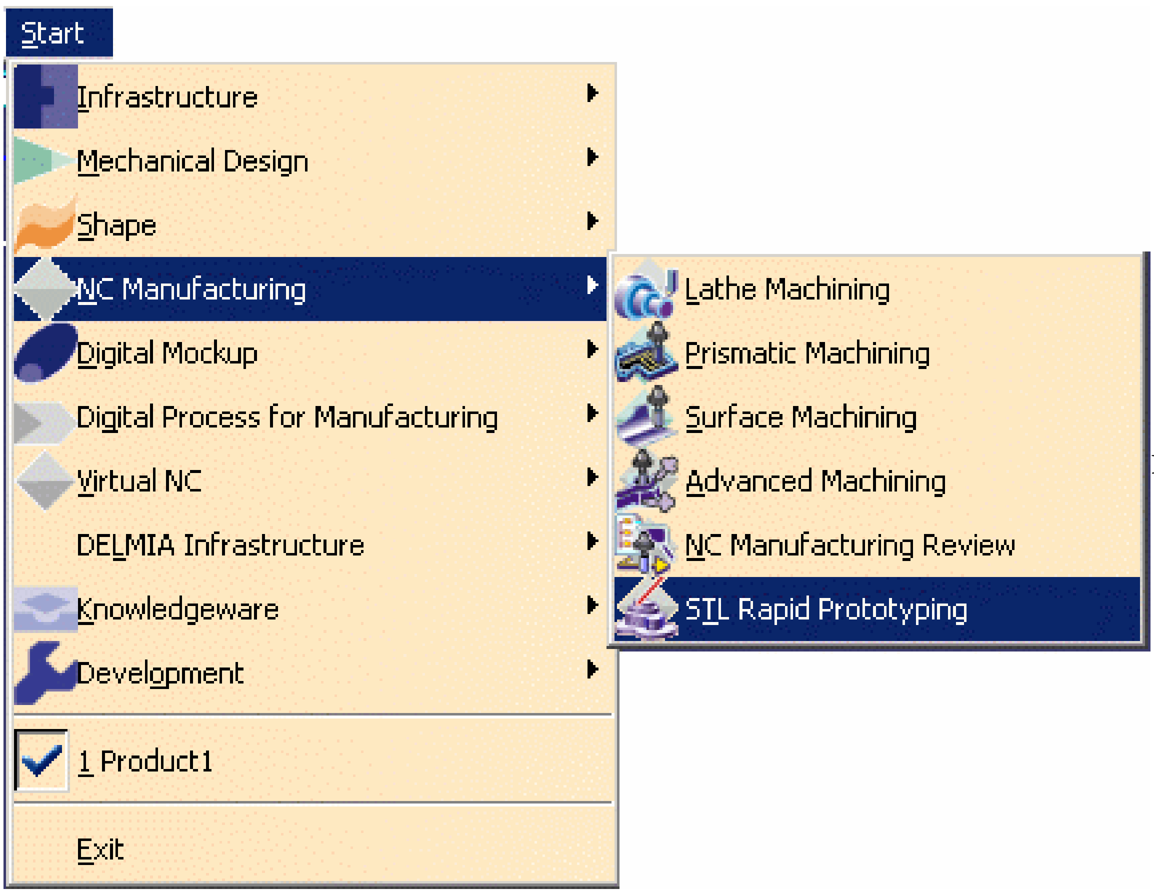


Menu Bar

This chapter describes the menus available in the workbench. Other menus are documented in the Infrastructure User's Guide.

Start	SmarTeam	File	Edit	View	Insert	Tools	Windows	Help

Start



Starts the STL Rapid Prototyping Workbench

Insert

For	See
STL Import	Importing Files
STL Export	Exporting Meshes to STL
Edition:	Activating a Portion of a Cloud of Points Removing Elements
Mesh:	Mesh Regeneration

Tessellating an Object

Offsetting a Mesh

Flip Edges

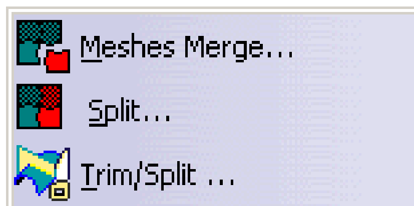
Smoothing Meshes

Mesh Cleaner

Filling Holes on Meshes

Decimating Meshes

Operation:



Merging Meshes

Splitting Meshes

Trim/Split

Analysis

Information

For the other menu items, please refer to the Infrastructure User's Guide.

Creation Toolbars

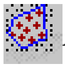
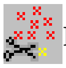
They are the following:

STL Files
Edition
Mesh
Operations
Analysis

STL Files

For	See
Import	Importing Files
Export	Exporting Meshes to STL

STL Edition

	For	See
	Activation	Activating a Portion of a Cloud of Points
	Remove	Removing Elements

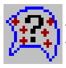
Tessellation

For	See
Mesh Regeneration	Mesh Creation
Surface Tessellation	Tessellating an Object
Offset	Offsetting a Mesh
Flip Edges	Flip Edges
Mesh Smoothing	Smoothing Meshes
Mesh Cleaner	Mesh Cleaner
Interactive Triangle Creation	Interactive Triangle Creation
Filling Holes	Filling Holes on Meshes
Decimation	Decimating Meshes
Optimize	Optimizing Meshes

Operations

For	See
Meshes Merge	Merging Meshes
Split	Splitting Meshes
Trim / Split	Trim/Split

Analysis

	For	See
	**Information**	Information

STL Display Options Toolbar

It contains the following tool:

Display Options and Graphic Properties

Specification Tree

The specification tree portion specific to STL Rapid Prototyping looks like this:

Geometrical Sets

Groups

Output of Import

Output of Mesh Regeneration

Output of Surface Tessellation

Output of Offset

Output of Polygon Smoothing

Output of Meshes Merges

Output of Split

Output of Mesh Cleaner

Output of Trim/Split

Glossary

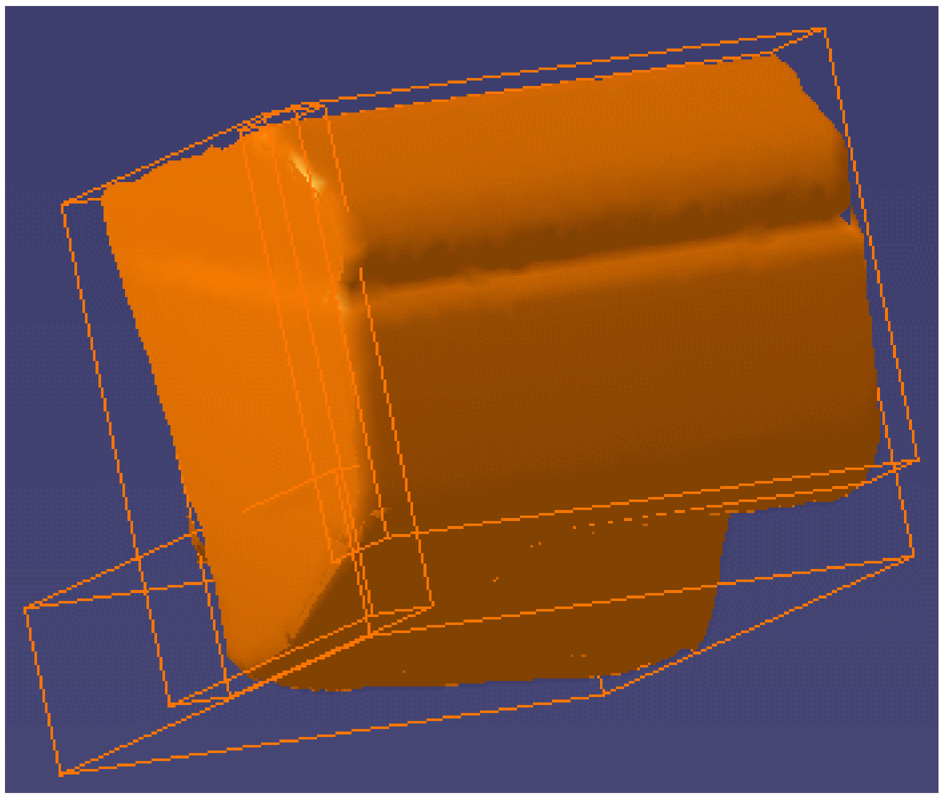
A C F M R S

A

activate This function is used to define a particular portion of a cloud for further operations.

C

cell A polygon may consist of several cells (i.e. sub-clouds): for example, the polygon below consists of two cells.



characteristic line They are particular lines corresponding for instance to curvature variations (fillets start/end) or sharp edges, ...

F

flipping edge An edge common to two triangles of a polygon may be flipped, that is rotated, to respect the shape a sharp edge of the tessellated part.

M

mesh

A mesh consists of a set of polygonal faces (triangles) which represent the surface of a 3D model. A triangulation is computed to describe the neighborhood relation of all points.

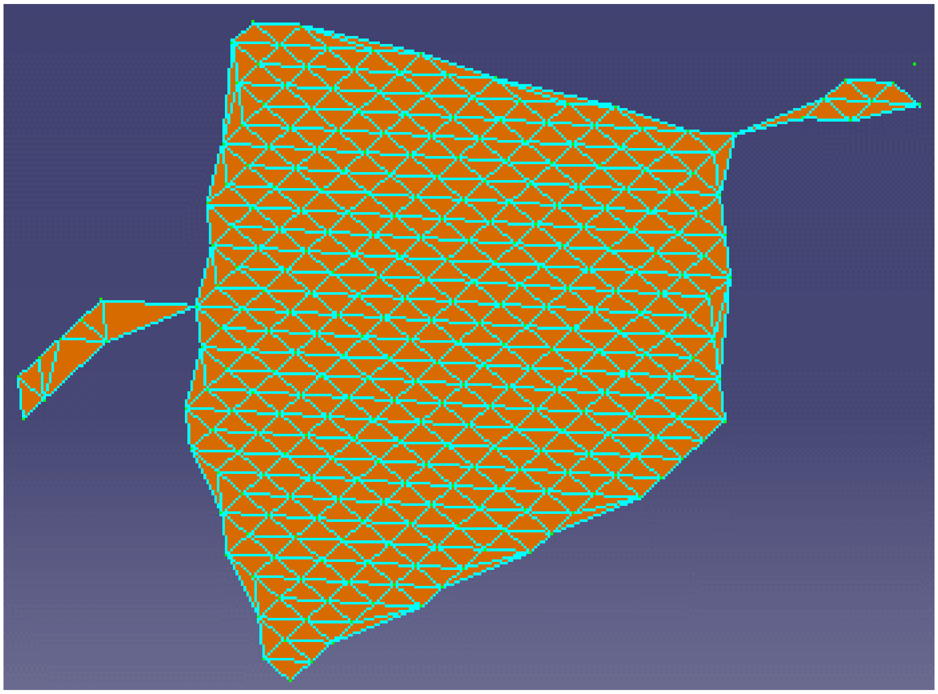
A mesh can be used to check the quality of the points, or can be processed in other applications.

A mesh may present some irregularities such as:

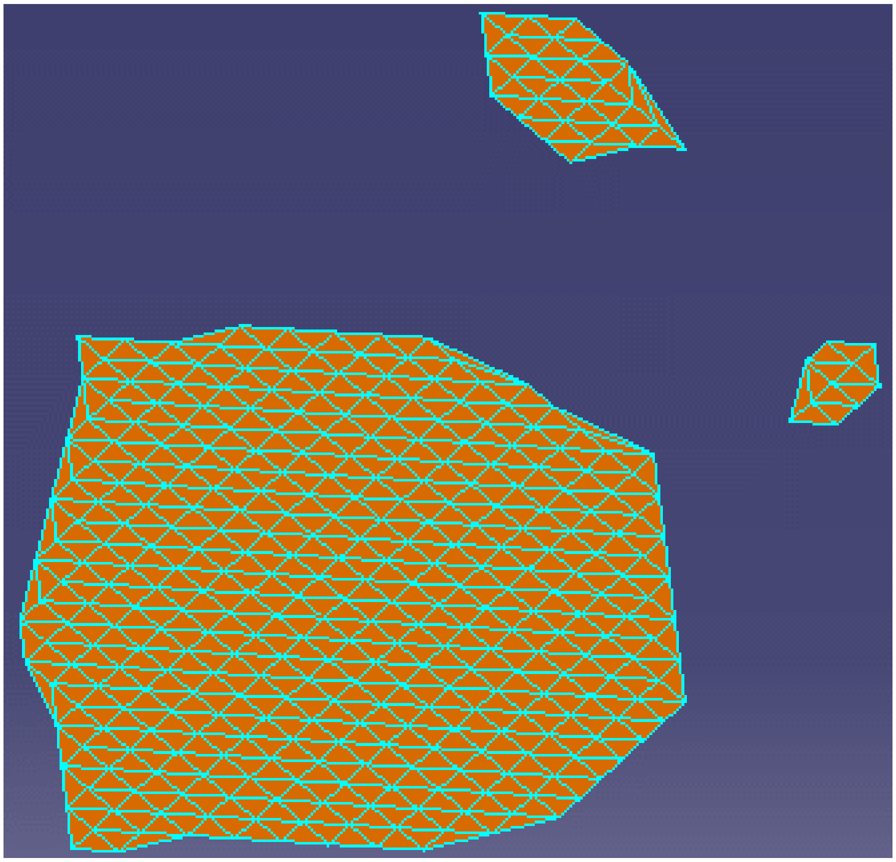
- Corrupted triangles, i.e. triangles that have the same vertex twice,
- Duplicated triangles, i.e. triangles that share the same three vertices,
- Non-manifold edges, i.e. edges shared by more than two triangles,
- Non-manifold vertices, i.e. vertices shared by two or more connected shells.

A mesh may also present some structural problems such as:

- Orientation problems, i.e. all the triangles are not oriented in the same direction,
- Isolated triangles, i.e. triangles belonging to small connected areas of the mesh,



- Disconnected zones, i.e. the mesh is made of several disconnected zones,



- Triangles with long edges.

R

remove This function deletes physically points from the cloud of points. The points can not be recovered.

S

shading A tessellation can be visualized in shaded mode. This mode is a method for visualizing the point data and getting an impression of its quality.

Index

A B C D E F G H I L M N O P R S T U

A

B

C

D

E

F

G

H

I

L

M

N

O

P

R

S

T