3DReshaper Beginner's Guide

Beginner's guide

1 Legal notice

The goal of this beginner's guide is to learn how to start using 3DReshaper. This manual is provided for informational use only, and is subject to change without notice. Technodigit assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

2 Your beginner's guide

This Beginner’s guide will walk you through some typical process using 3DReshaper 2018, called the **software** in the following. All practices used in this guide are present in the public document directory, by default:*C:\Users\Public\Documents\3DReshaper 2018 (x64).*The folder containing the samples (/Samples) used in this document can be accessed either by the shortcut created in *My Documents/Public documents* or by directly using the entry in the windows start menu (*3DReshaper 2018/Samples*).

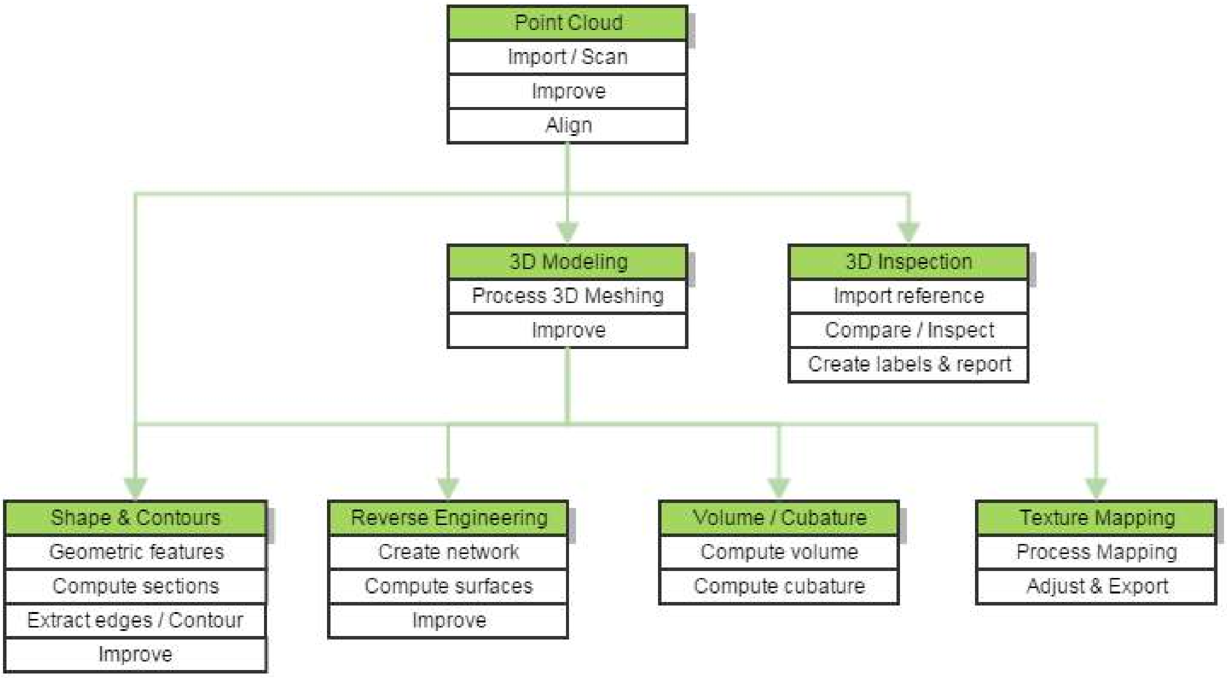
3 Content

- Basics of the software
- Point Cloud Processing
- Alignment - Registration
- Meshing and mesh improvement
- Sections and Polylines
- Measurement, Inspection and reporting
- Surveying
- Tank
- Image
- CAD

4 Basics of the software

4.1 Typical workflows

Here are the typical workflows in the software:



4.2 First steps in the software

In this section, we will learn the general features of the software: how to customize the software, how to handle objects, how to click a point, etc.

- Exercise: Browsing a Reshaper project
- Exercise: Learn all the different options to click a point
- Exercise: Understand meshes orientation

4.3 Exercise: Browsing a Reshaper project

- Loading a RSH file

- Changing the view
    - Rotating, panning and zooming
    - Predefined views
    - Keyboard shortcuts
    - Viewsets
    - Orthographic/Perspective
    - Clipping plane
- Selecting objects
    - In the tree
    - In the 3D Scene
- Editing an object
    - Showing or hiding an object
    - Renaming
    - Moving an object from one folder to another
    - Undo-Redo
    - Changing the representation and the color
- The different objects

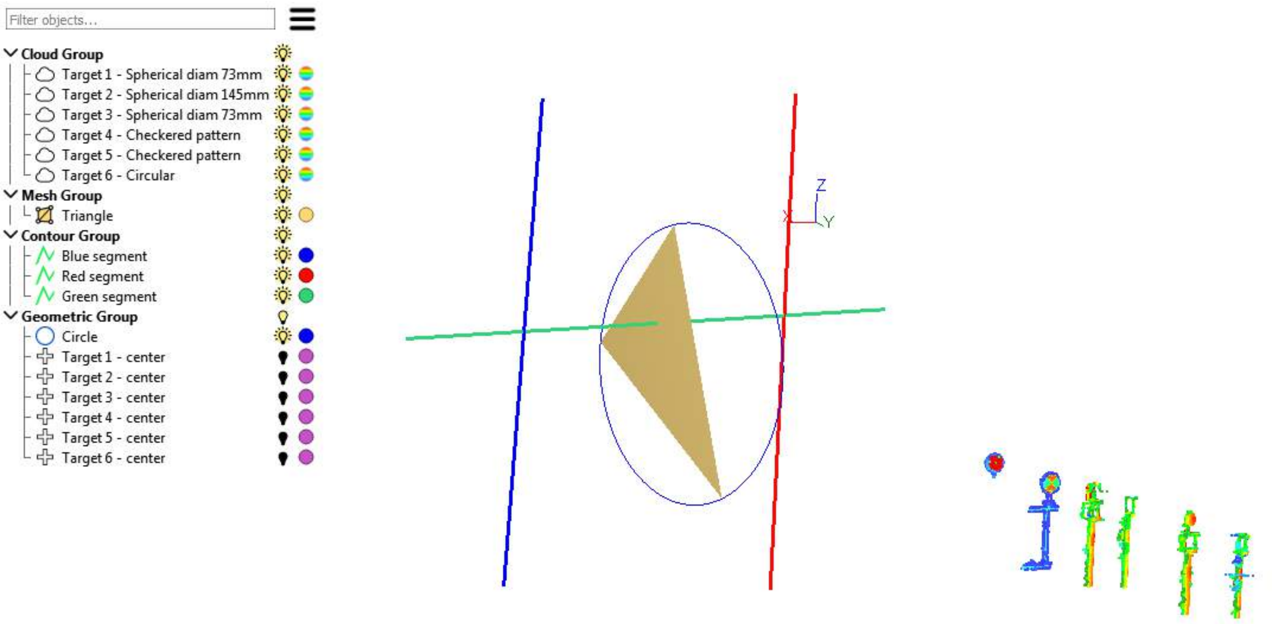
4.3.1 Loading a RSH file

Several options are available in order to open an rsh file:

- Double-click on the file in your Windows explorer.
- Launch the software and run the command File \ Open.
- Launch the software and then drag the rsh file from your Windows explorer to the software.

For this exercise, open the file “EnterPoints.rsh”.

Drag and drop as well as double-click is also possible for file formats which are known by the software. Examples of these known file formats are .pts, .stl...etc.



The EnterPoints.rsh file after opening

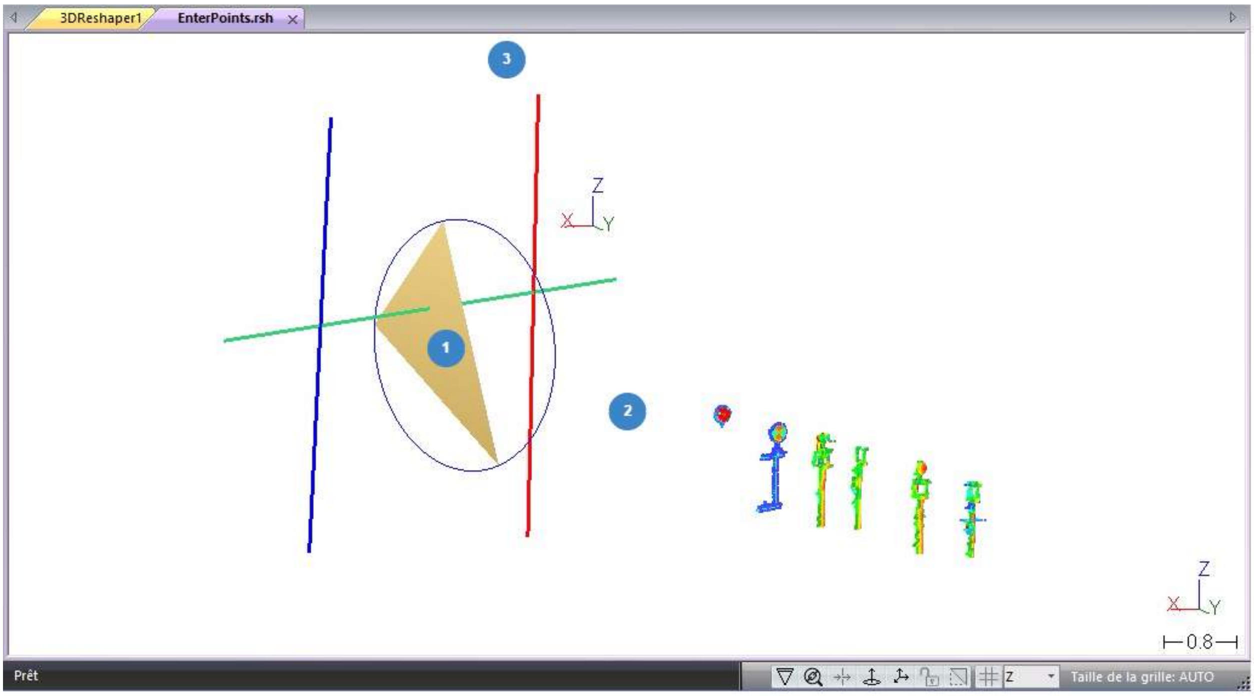
4.3.2 Changing the view

The following exercise will guide you through the most common ways to modify the view. For more details, refer to the sections located in the general instructions.

Rotating, panning and zooming

The mouse will allow you to manipulate the view:

- To **rotate the view**, **right click** on the middle of the scene and move the mouse while keeping the right button pressed.
  - If an object is behind your cursor, the corresponding point on the object will be the rotation point (also called picking point).
    Try to rotate the view with the mouse over the triangle (position 1 in the image below).
  - If no object is behind the mouse, a point in the middle of all visible objects will be used as rotation point.
    Try to rotate the view with the mouse over an empty area (position 2 in the image below).
  - In both cases, a dotted rectangle is displayed in the 3D scene meaning that you are rotating the view.
- In the same way, when clicking near the border of the scene, you can rotate the view around an axis perpendicular to the screen (the view direction) and crossing the scene center. In this case a dotted circle is displayed meaning that you are rotating that way.
  Try it by placing the mouse in position 3.



The mouse position will change the rotation conditions

Panning is also done using the mouse, by pressing both **left and right buttons**, or by pressing wheel button.

Press both buttons and move your mouse while keeping them pressed.

Zooming in and out is possible by **scrolling the mouse** or by using the **SHIFT + right button** combination:

Scroll with your mouse in the 3D scene.

Note that the point behind your mouse is not moving when zooming in or out.

It is also possible to manipulate objects in the 3D scene using a 3D mouse from 3DConnexion. See 3D mouse for more details.

Predefined views

The menu View allows you to quickly change your view to display all objects in the front view:

Run the View/Front command.

Run the View \ Zoom All command.

The View menu also contains tools allowing to split the views in up to 8 different views:

Click on View \ Split View Vertically

Click on the right view so that it becomes the active view.

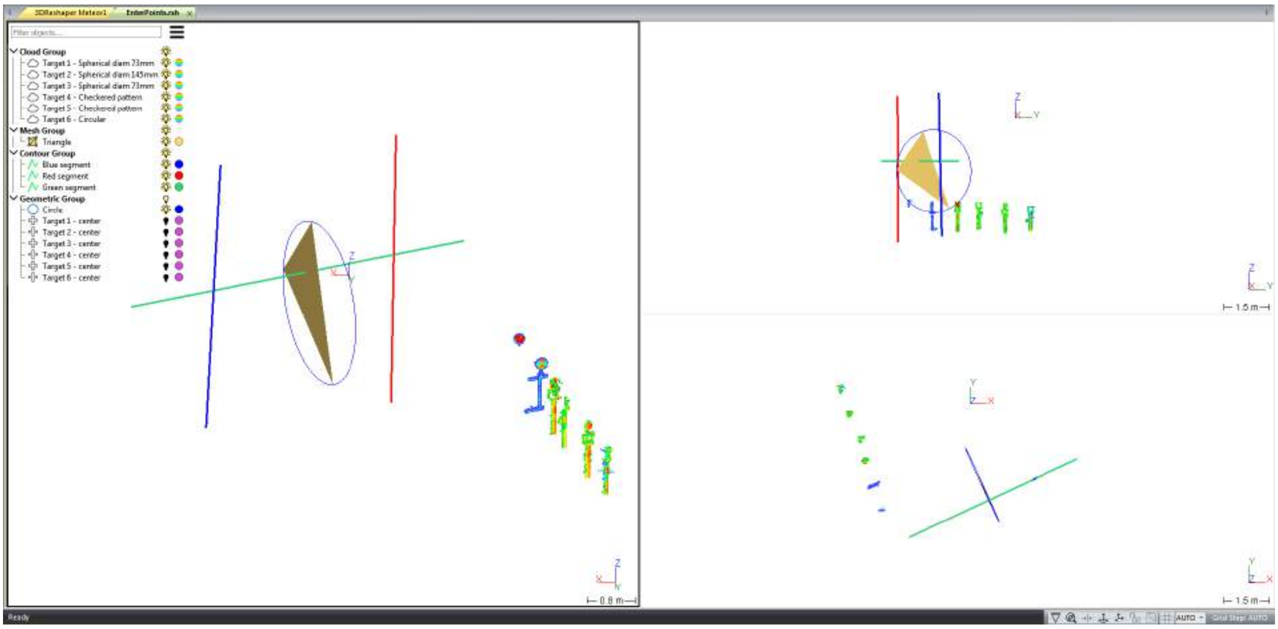
Click on View \ Split View Horizontally.

Click on the top right view and then press the **X** key in order to obtain a front view.

Click on the bottom right view and then press the **Z** key in order to obtain a top view.

Rotate the left view as you wish.

In order to go back to a unique view, run the command View \ Keep only one view.



Views can be splitted and individually oriented as needed

Keyboard shortcuts

Several keyboard shortcuts dedicated to the view are available in the software:

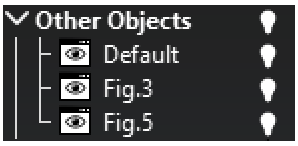
- **X**: changes the view to YZ view (also called front view)
- **Y**: changes the view to XZ view (also called right view)
- **Z**: changes the view to XY view (also called top view)
- **A**: changes the current zoom so that all displayed objects are in the view

SHIFT+X reverses the view from front to back, and of course, the same mechanism is setup to change the view to left view with **SHIFT+Y** and to bottom view with **SHIFT+Z**. Refer to the dedicated shortcut page for more details about all available shortcuts.

Viewsets

The View Set command (**View** menu) saves the current view of the 3D scene. A new object is created; you can see it in the tree in **Other Objects** folder. Thus, you can restore a view by clicking on the corresponding bulb.

Some viewsets have been created in samples and practical exercises. They may be suitable to help you localize specific zones, effects, etc. You could also click on **Default** Viewset to return to the starting view.

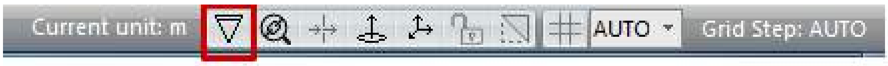


Viewsets

Orthographic/Perspective

When loading the EnterPoints.rsh file, the view is in orthographic mode by default. The view can be switched between orthographic and perspective by using:

- the **P** keyboard shortcut
- the button located in the status bar



The status bar contains buttons allowing to switch from orthographic to perspective
Press the **P** key and see the difference between an orthographic view and a perspective view.

> Perspective view can be very useful for looking inside objects. An alternative to perspective view is activating a clipping plane.
>
> More information are available in the dedicated page: Perspective and orthographic view.

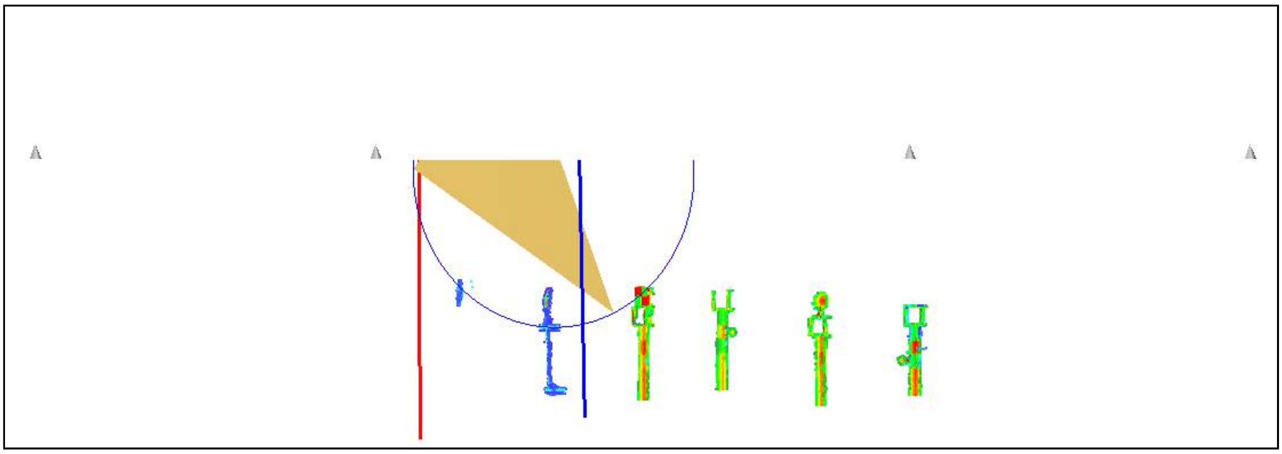
Clipping plane

A clipping plane is a visualization tool which helps to see inside an object without modifying it. Consider a clipping plane as an infinite plane; everything on one side of the plane is hidden (clipped) and the other side remains visible.

- Press **X** to display a front view and launch the command View \ Clipping plane without selecting any objects.
- Set the **Orientation** (plane normal) parallel to **Z-Axis**.
- Check the options **Step Auto** and **Activate.**
- Set **Depth** parameter to **0** and **Associate mode** to **All visible.**

- Validate with **OK.**

A horizontal clipping plane has been created.



Horizontal clipping plane

In the tree, you can hide (or display)

and switch off (on)

the clipping plane.

Press **CTRL+SPACE** to edit the clipping plane with the mouse (drag and drop, CTRL+SCROLL). Press a second time **CTRL+SPACE** to exit the clipping plane edition mode.

Status bar-Clipping plane edition

4.3.3 Selecting objects

The way of using the software is to select the data that you want to work with, for example select the cloud (s) to mesh...etc. By default, an object is displayed in purple in the 3D scene when it is selected and highlighted in the tree. You can either select an object in the tree or directly in the 3D scene.

In the tree

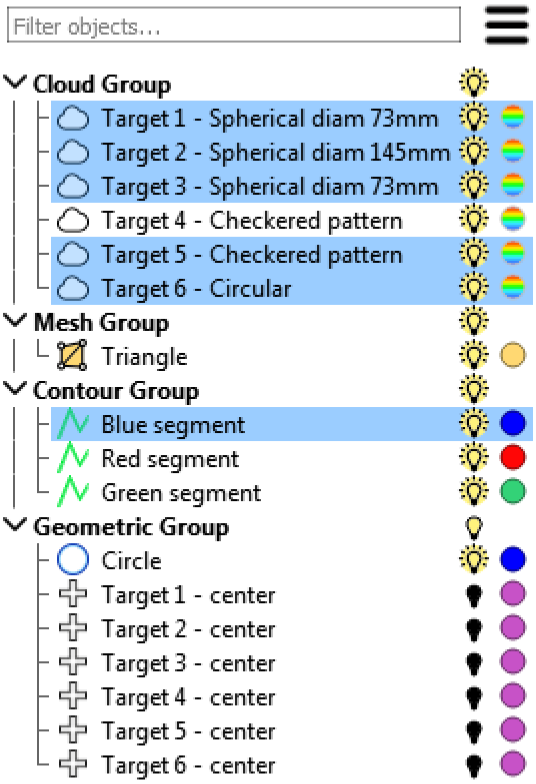
The selection of elements in the tree is very similar to the way you can make selections of files and folders in your Windows explorer:

- Select only one element one after the other by clicking on it
- Select several consecutive elements using the **SHIFT** key
- Select several non-consecutive elements using the **CTRL** key

Try selecting the clouds numbered 1 to 6 in the tree using the **SHIFT** key.

Add to the selection the "Blue segment" polyline using the **CTRL** key.

Remove to the selection the "Target 4 - Checkered pattern" cloud using the **CTRL** key.



Selection of several entities in the tree

In the 3D Scene

Objects can also be selected directly from the 3D scene:

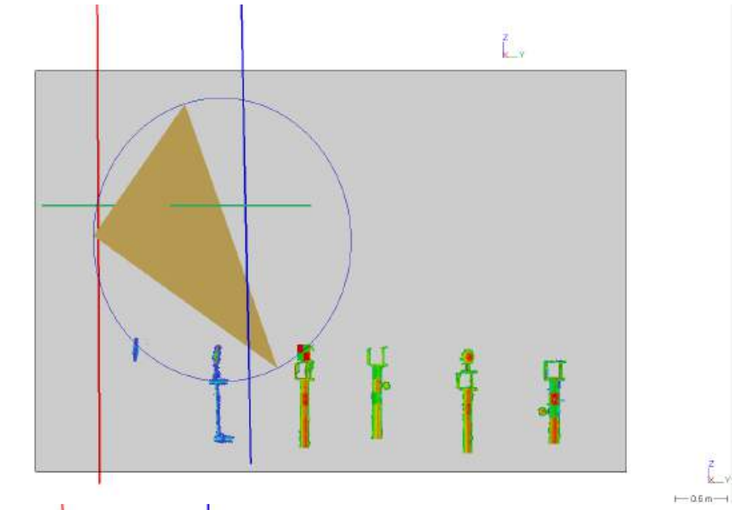
- by using the mouse **LEFT click** on an object
- by using a rectangle selection (moving the mouse from one point to another with the **LEFT click** of the mouse pressed)

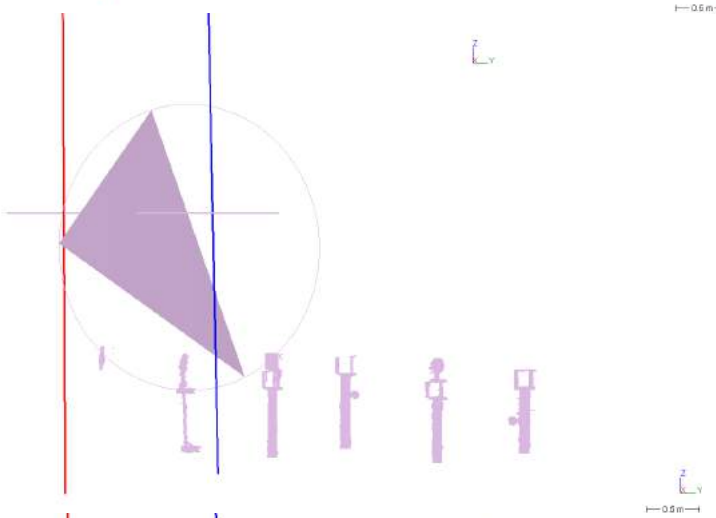
> Refer to the dedicated page in the general instructions for the difference between rectangles created from left to right or right to left.

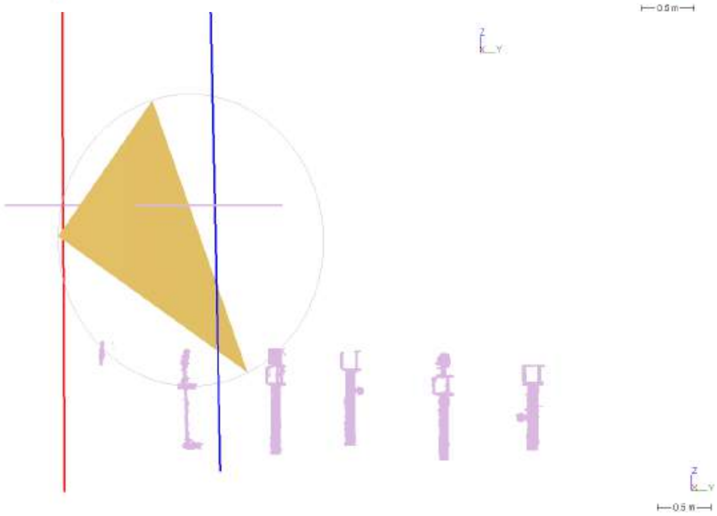
Press the **X** key and then on the **A** key to display all visible objects in front view.

Draw a rectangle like in the 1st image below from left to right. The result of the selection should be similar to the 2nd image below

Press the **CTRL** key while **LEFT clicking** on the triangle in order to remove the triangle from the selection. The result of the selection should be similar to the 3rd image below







Select all visible elements using the **CTRL+A** keyboard shortcut and deselect all elements by using **ESC** key.

4.3.4 Editing an object

Showing or hiding an object

An object can be shown (or hidden) in two ways:

- With the contextual menu: select the object to show (using the above procedure), right click with your mouse to show the contextual menu and press on **Show** (or **Hide**)
- With the tree: click on the lamp icon (which is either on or off) in order to switch between shown and hidden status

Hide the triangle by clicking on the lamp icon (which is "on" when the document opens).

Select several objects (either in the tree or in the 3D scene) and try hiding or showing them using the lamp icon on the tree or the contextual menu.

Renaming

At any time, you have the possibility to change the name of an object; to do so, you have 3 possibilities:

- With the contextual menu: select the object you want to change the name, right click with your mouse to show the contextual menu and press on **Rename**
- In the tree, left click twice (slowly, to avoid double-click) on the element you want to rename
- Use the **F2** keyboard shortcut in order to rename selected elements

Rename the Circle to "Circle 1"

Moving an object from one folder to another

Regular commands (such as cut, copy, paste, delete...) are available in the contextual menu or in the menu Home. These commands allow to cut an object from one folder and paste it in another folder (or another document). Another workflow for moving the objects is to use the drag and drop functionality of the tree which is much quicker than the regular Cut/Paste.

Drag and drop the "Circle 1" from the "Geometric Group" to the "Contour Group"

Undo-Redo

Every action you will do in the software can be undone - and then redone if needed. Undo and Redo can be applied using the dedicated Home \ Undo and Home \ Redo menus or using the keyboard shortcuts: **CTRL+Z** and **CTRL+Y**.

Select the circle and hit **DEL** to delete it. See that it moves into the Recycle bin folder and is now hidden.

Press **CTRL+Z** and see that the circle is back to the "Contour Group"

Press **CTRL+Z** again and see that the circle that was just moved from the Geometric Group goes back to this group

Press **CTRL+Z** again and see that the name of the Circle is back to original

Recycle bin

- A deleted element goes into the Recycle bin
- At any time, an object in the recycle bin can be restored in its original folder

Changing the representation and the color

Most objects can be displayed using different representations (see representation section for more details). Meshes, for example, can be displayed in: Smooth, Flat, Wire, Smooth+Wire, Flat+Wire, Textured, Real Color, Inspection. Some representations are only available if the information is available: if the mesh contains no texture information, this representation is not proposed. Changing these representations can be done in two ways:

- With the contextual menu: after right clicking on a selected objet, go in sub-menu **Representation**
- With the tree: left click on the colored icon

Select the triangle and change the representation from "Smooth" to "Smooth+Wire"

In the software, a single color is applied to objects in the representations that do not override colors. Inspection representation, for example, overrides the color. You have 2 ways to change the color of an object in 3DReshaper:

- With the contextual menu: select the object you want to change the color, right click with your mouse to show the contextual menu and press on **Color**
- With the tree: left click on the colored icon and change the color

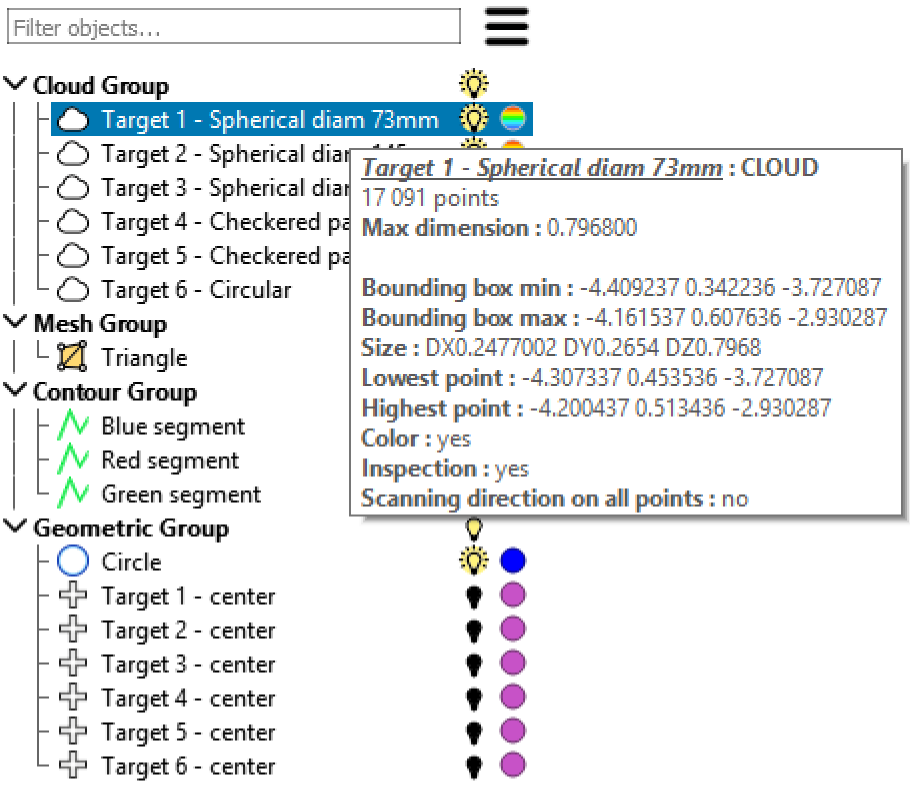
In the tree, change the representation of the cloud named "Target 2 - Spherical diam 145mm" to Smooth

Then, change the color of the same cloud to blue

Additional display options are available with the Home \ Color and Aspect command. For example, this will allow you to change the point size of a point cloud or define your default settings.

4.3.5 The different objects

When working in the software, you will certainly create different types of objects: clouds and meshes of course but also polylines, geometrical features (such as lines or circles), coordinates systems, view sets... The tree view of the document allows you to easily identify the type of objects as a specific icon is dedicated to each object. Depending on the type of objects, different information is available within its properties. You can view object properties using the command Properties in the contextual menu, or simply by keeping the mouse up to an object in the tree view.



Object properties

When using the contextual menu, you can display properties of several objects at once if several objects are selected.

4.4 Exercise: Learn all the different options to click a point

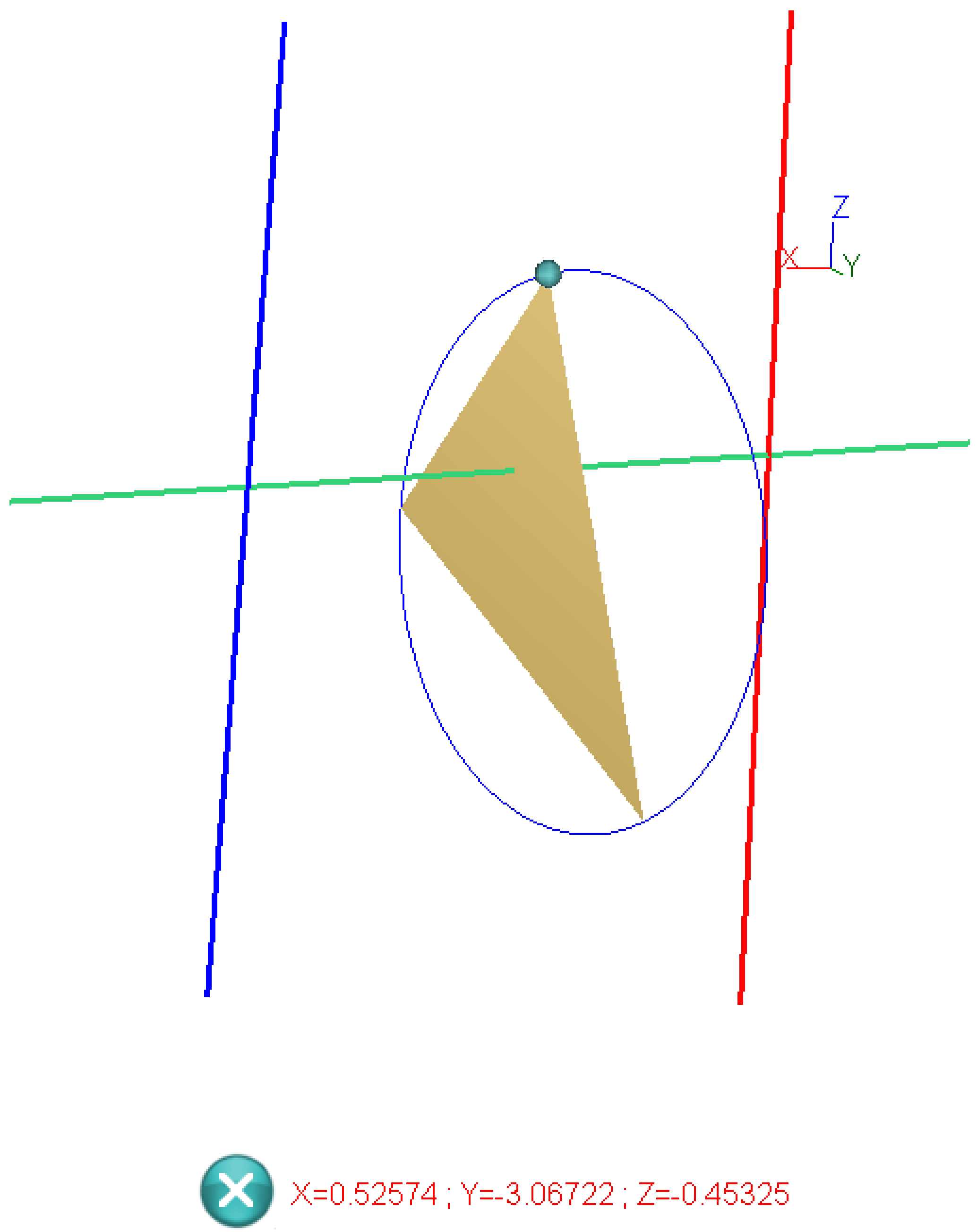
In the software, each time you have to click a point, a ribbon, like the one in the following picture, will appear.

The ribbon with options to enter a point

This ribbon will allow you to choose between several options to click a point. Some options can be selected at the same time depending on their compatibility (for example, if you select **Point on Selection**, **Vertex / End** will be automatically unselected).

Open the file “EnterPoints.rsh” in order to practice. Make sure all the objects from the tree explorer are visible and open the command Construct\Point.

Therefore, when the software is waiting for a point, you will see, at the bottom of the scene, the coordinates of the selected point and an icon showing which snapping it is. If the point is not just behind the cursor, you will also see a small ball in order to locate the point. For example, in the following picture, the current point is (0.52, -3.06, -0.45) and is located on one vertex of the triangle.



When the software is waiting for a point input, a preview is made in real time in order to see where the point will be clicked

4.4.1 Point on selection

With this option, you can click on a point on the object behind the cursor exactly where the cursor is. Click on a point on the triangle: you will see that the created point is exactly just behind the cursor and on the triangle. Then try to click on a polyline, the green one for example, the point will be also located exactly on the line.

4.4.2 Vertex / End

With this option, you can click on an existing point. It means that if you click on a triangle, the point will be created on a vertex. If you click on a line, it will be located on an extremity, etc. Try to click on a point on the triangle; the created point will be on the nearest vertex. Then click on a line; the point will be located on the nearest extremity.

4.4.3 Nearest 3D projection

This option means that the point is projected in 3D on the nearest entity. "3D projection" means the shortest distance between a point and a 3D object.

4.4.4 Middle / Center

With this option, you can click on the middle of a segment or the center of a geometric feature. Try to click on a point on a segment: the created point will be the middle of the segment. Then click on a point on the circle; the created point will be the center of this circle.

4.4.5 Intersection 2D/3D

This option means that the clicked point is the intersection between two lines (in 2D or 3D). The intersection will be computed according to the zoom factor. This means that if there is not a real intersection between two lines and if you make a zoom to focus on the intersection area to see the small gap between lines, the system will not accept to make the intersection. However, if you zoom out, the intersection will be clicked (and the point will be equidistant from the two lines). The minimum distance between the two lines will be given in a warning message and the system will sound as a bell, to tell you that there is no real 3D intersection between the lines.

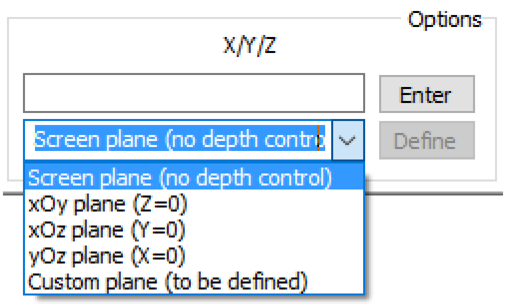
Select only the option **Intersection 2D/3D**, then zoom considerably on the intersection between the green and the blue segment, and then try to click on this intersection. The point will not be created as the distance between the two lines is quite large compared to the screen size. Now, zoom out and try again. This time the intersection will be created as the distance is now very small compared to the screen size (and the clicked point is equidistant from the two lines). However, a warning message appears in order to explain that there is not a real intersection between the two segments (and the given distance is the smallest distance between the two segments).

Now, try again clicking on the intersection between the green and the red segment. The intersection will be clicked whatever the zoom is, as there is a real intersection between this two lines, and you will not have a warning message.

4.4.6 XYZ

This option allows entering a point without projection on geometry. You can enter manually (or with copy /paste) an XYZ coordinate. You must enter in the field one, two or three values separated by a character that is neither a number nor a character nor a dot:

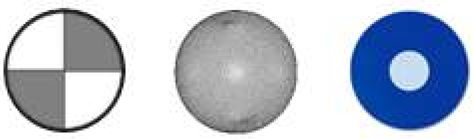
- Enter "1, 2, 3". The software will understand X=1; Y=2 and Z=3.
- Enter now "1.1 2.2". The software will understand X=1.1; Y=2.2 and Z=3. As the third coordinate is missing, the Z coordinate will keep the previous value.
- Enter "1.1,X2.2;3.3". The software will understand X=2.2 because the letter "X" was present, Y=1.1 because Y is the first unnamed value and Z=3.3.
- Enter "DX3". The software will understand X=5.2 because the previous value was 2.2 and DX3 means "add 3 to X"; Y= 1.1; Z= 3.3 because it keeps its value.



Options to define the plane where the points are created with the XYZ option
However, you can also click on points directly in the 3D scene, even if there is no object behind the mouse. By default, the point will be created in the screen plane, but you can define the plane where points are created (Picture above):

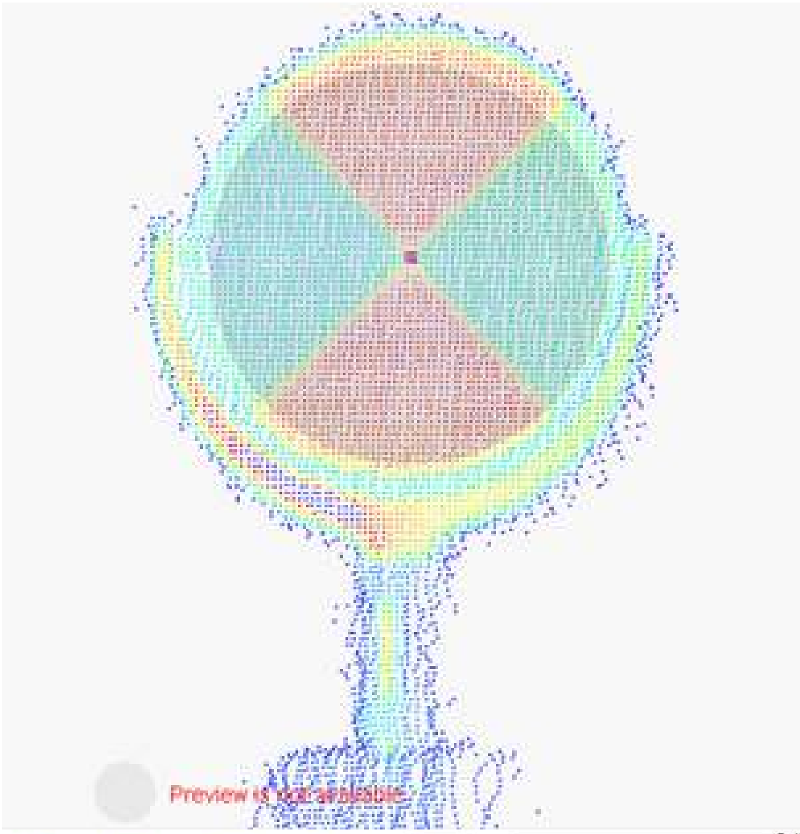
- Screen plane: click on some points; they will be created in a plane parallel to the screen plane (impossible to control the depth).
- xOy plane: click on some points; they will be created in the xOy plane (the Z of the points will be 0).
- xOz plane click on some points; they will be created in the xOz plane (the Y of the points will be 0).
- yOz plane: click on some points; they will be created in the yOz plane (the X of the points will be 0).
- Custom plane: click the "define" button, then click on the circle and click on some points, they will be created in the same plane than the circle.

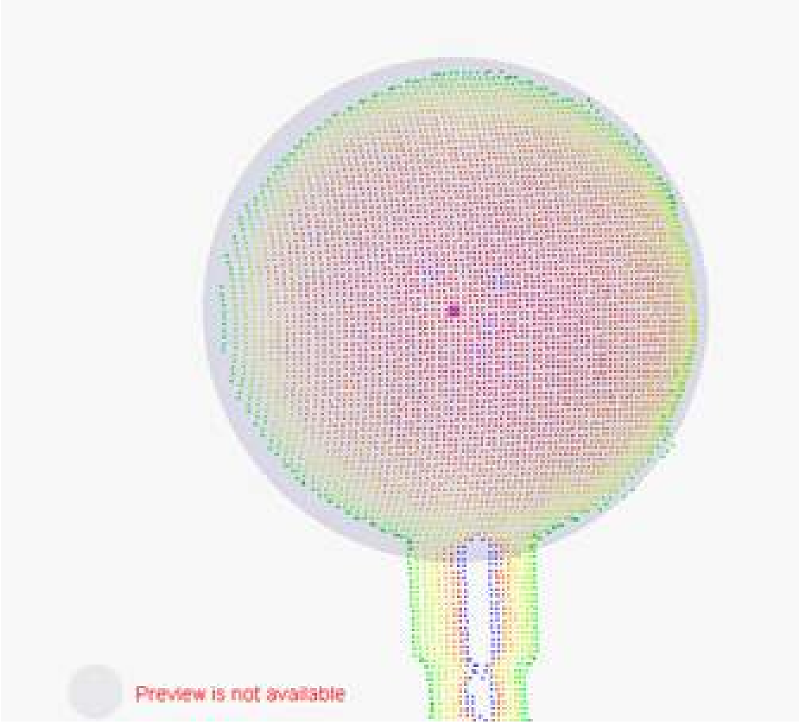
4.4.7 Surveying target



Target compatible with 3DReshaper

If your point cloud contains some surveying targets (black and white, spherical or blue and white) like one on the picture above, you can click on the center directly on 3DReshaper. For example, select the option **Surveying target** and **With checkered pattern**, enter 0.15 as the diameter target and then click on a point on the target located on the point cloud **Target 5 – Checkered pattern**. The created point will be the perfect center of the target. Use the **Inspection** representation in order to better detect targets in the cloud (for targets with checkered pattern and for circular targets).





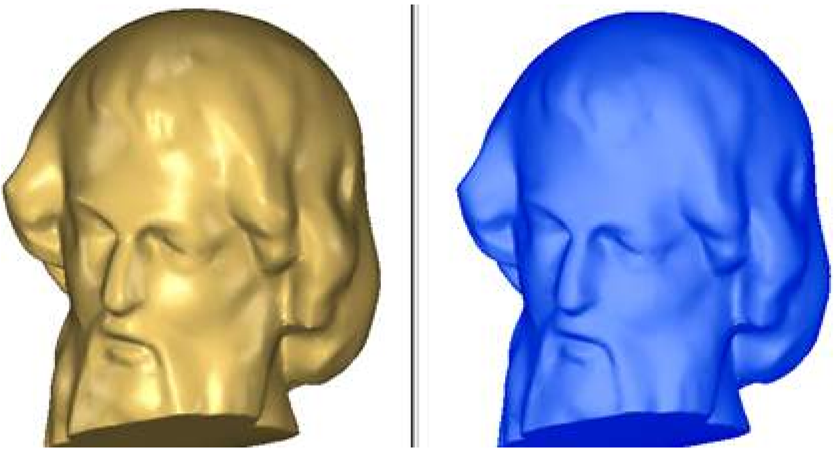
With the "Surveying target" option, you can click automatically and precisely the target center (with checkered pattern on the left, spherical on the right)

4.5 Exercise: Understand meshes orientation

Open the file “FillHoles.rsh”.

In the software, the default color for meshes is gold. Select and show only the mesh “4-Closed Mesh”. Select it again; right click and then select **Properties**. Have a look at the volume, and if it is positive: everything is OK.

Now select the mesh and right click and select **Reverse**. The color will change. Have a look again at the properties; the volume is now negative: normals are not well oriented. Try to change the color of the mesh (select the mesh and right click and go to the **Color** menu). The mesh will not take the selected color (it will be an opposite color).



On the left the normals are OK; on the right the normals are inverted.

5 Point Cloud Processing

In this section, you will see how to import or create point clouds in the software, and you will learn how to work on a cloud to improve it before using it for further processing.

To have an overview of the supported format, see Import Cloud(s).

- Import a point cloud
  - Exercise: Import several point clouds at the same time, and merge them
  - Exercise: Convert a cloud from a unit to another
- Improve a point cloud
  - Exercise: Remove or separate a part of the cloud
  - Exercise: Clean a cloud using automatic filters
  - Exercise: Reduce a point cloud

5.1 Import a point cloud

- Exercise: Import several point clouds at the same time, and merge them
- Exercise: Convert a cloud from a unit to another

5.1.1 Exercise: Import several point clouds at the same time, and merge them

Import 6 clouds

1. Open the menu File \ Import \ Import cloud(s).
   Note: the File menu is located under the 3DR icon at the top left corner of the application
2. In the "Import Clouds" dialog box, click **Add**.
3. Select all the files from “ImportCloud-1.asc” to “ImportCloud-6.asc” by pressing the **Shift** key and click on **Open**.
   It is also possible to drag and drop the files into the **Import window**. In this case, all points will be imported.

Six independent clouds are created in the Cloud Group.

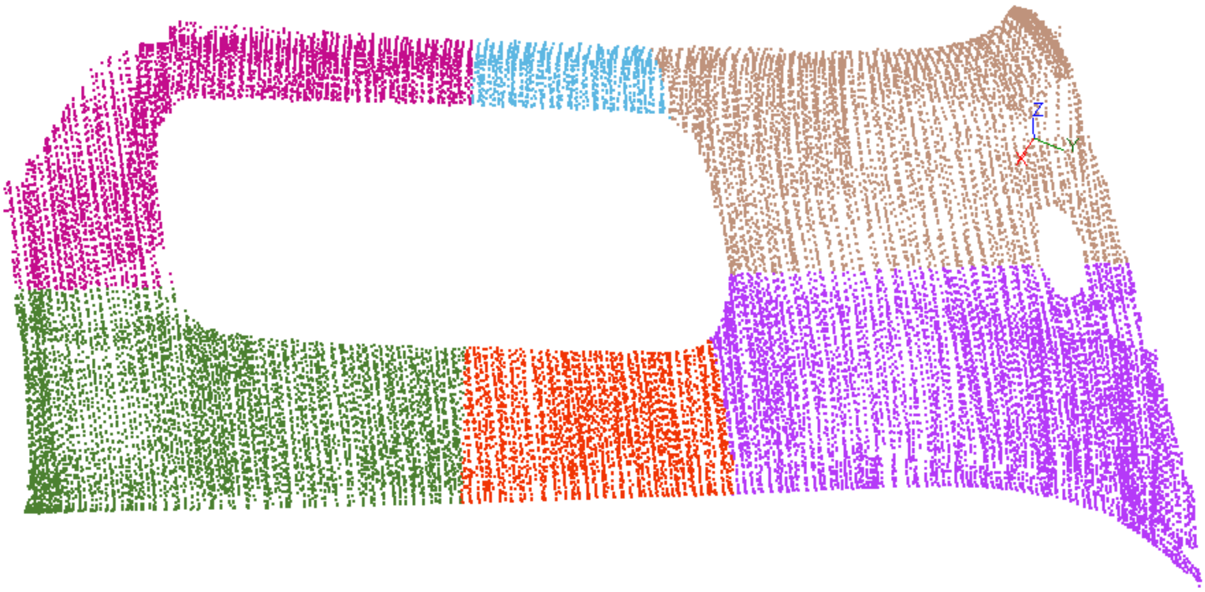
Merge the 6 clouds

They can be merged into one unique cloud.

1. Select the 6 clouds and go to Cloud \ Merge Clouds.
2. Choose whether to keep the colors of the clouds or not.

3. Validate the result with **OK**.

You obtain one cloud called "Merged Cloud".



Merged cloud keeping the initial color of each cloud

5.1.2 Exercise: Convert a cloud from a unit to another

Open the file BestFitOnRef.rsh.

This file contains several clouds, in meters and one in feet as well as a reference mesh in meters. Show only the clouds named "Aligned Dam" and "Aligned Dam in ft" and press **A** to make a Zoom All. Select both clouds, right click and select **Properties**. Have a look at the size of each cloud: one is roughly 3 times bigger than the other.

To do the conversion, select the cloud in feet and then go to Transform \ Scale and choose the **Scale** tab (if not already selected). Define the center (0, 0, 0) using the Entering Point Procedure. Then, select the option **Same scale for X, Y and Z** and enter 0.3048 (converting feet to meters requires a multiplication by 0.3048). Press the **OK** button. Have a second look at the properties; the size is now exactly the same.

```
Clouds properties before unit conversions

2 objects selected
        2     Cloud(s)

Aligned Dam: CLOUD
    500 000 points
    Max dimension: 30.716181m
    Bounding box min: 2.918194 7.77359 -0.37
93353
    Bounding box max: 13.018897 38.489771 13
.965062
    Size: DX10.100703m DY30.716181m DZ14.
344398m
    Lowest point: 12.2715 27.608797 -0.37933
53
    Highest point: 3.783792 8.847399 13.9650
62
    Color: no
    Inspection: no
    Scanning direction on all points: no

Aligned Dam in ft: CLOUD
    500 000 points
    Max dimension: 100.774883m
    Bounding box min: 9.574137 25.503899 -1.
244539
    Bounding box max: 42.712945 126.278783 4
5.817134
    Size: DX33.138808m DY100.774883m DZ47.
061674m
    Lowest point: 40.260825 90.580036 -1.244
539
    Highest point: 12.414017 29.026906 45.81
7134
    Color: no
    Inspection: no
    Scanning direction on all points: no
```

```
Clouds properties after unit conversions

2 objects selected
        2     Cloud(s)

Aligned Dam: CLOUD
    500 000 points
    Max dimension: 30.716181m
    Bounding box min: 2.918194 7.77359 -0.3
793353
    Bounding box max: 13.018897 38.489771 1
3.965062
    Size: DX10.100703m DY30.716181m DZ14.
344398m
    Lowest point: 12.2715 27.608797 -0.3793
353
    Highest point: 3.783792 8.847399 13.965
062
    Color: no
    Inspection: no
    Scanning direction on all points: no

Aligned Dam in ft: CLOUD
    500 000 points
    Max dimension: 30.716181m
    Bounding box min: 2.918194 7.77359 -0.3
793353
    Bounding box max: 13.018912 38.489771 1
3.965062
    Size: DX10.100718m DY30.716181m DZ14.
344398m
    Lowest point: 12.2715 27.608793 -0.3793
353
    Highest point: 3.783792 8.847399 13.965
062
    Color: no
    Inspection: no
    Scanning direction on all points: no
```

Note

In case you have chosen to activate the units in the software, an additional tab is available and will allow you to directly scale from one unit to another (mm to m, inch to m…).

5.2 Improve a point cloud

A point cloud always needs some processing to remove undesired and noisy points before any further use. Besides, if you work with big point clouds, you may need to split them into independent parts in order to work separately on each one. Different ways of improving a point cloud are available in the software, by using manual or automatic functions.

- Exercise: Remove or separate a part of the cloud
- Exercise: Clean a cloud using automatic filters
- Exercise: Reduce a point cloud

5.2.1 Exercise: Remove or separate a part of the cloud

Open the file

Open the file CleanWithObject.rsh.

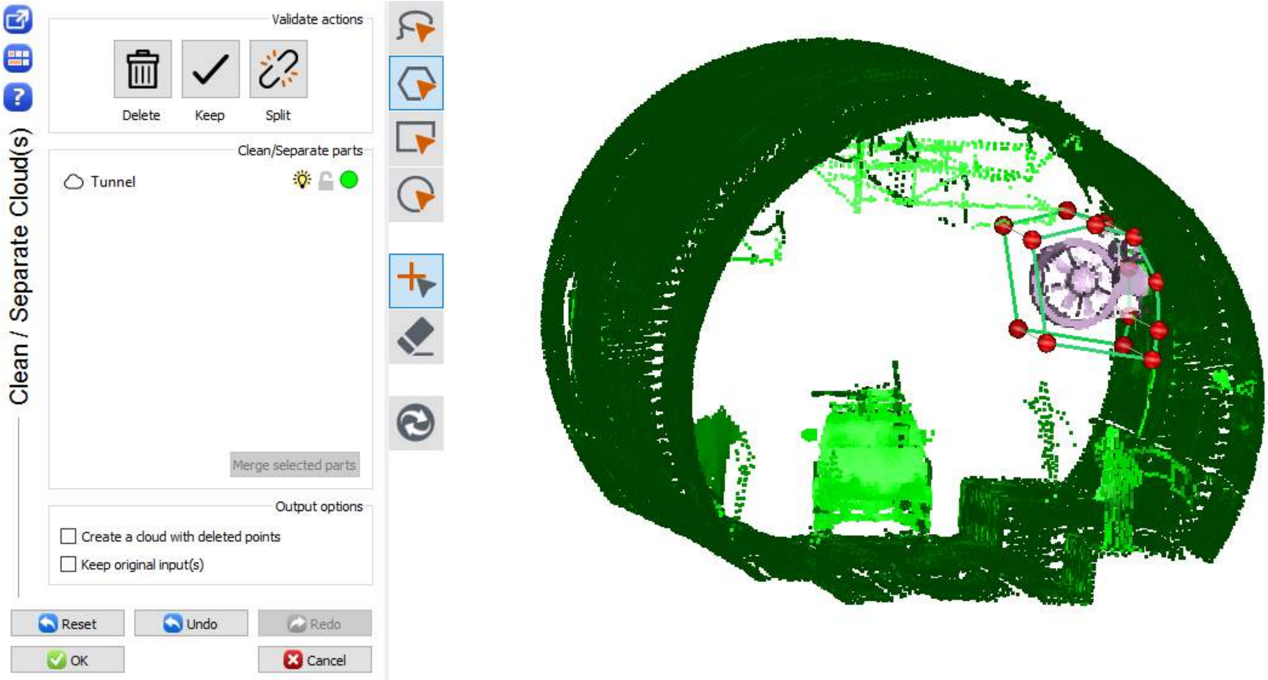
The file contains the point cloud of a tunnel. To make an inspection of the tunnel, we need to keep only the points from the vault. We can use three different tools to split a cloud into several parts and separate the vault from the rest of the points.

Draw a polygon

Select the point cloud “Tunnel” and go to Cloud \ Clean / Separate.

Orient the view (you may need the **ALT** key to slow down the rotation), draw a polygon around the points you want to select and press **Enter** to validate the selection. While drawing the polygon, you can cancel the last point clicked with the keys **Del** or **Backspace** on the keyboard.

The points in the selection are highlighted. Now you can rotate the view and move a ball in order to stretch the contour. When you move a point, it remains in the same plane. Drag and drop the contour or press **Shift** while moving the ball in order to create a selection box around the points. Click on the bin icon to delete selected points. Repeat the action on all the other big areas and validate with **OK.**



Remove a part of the cloud by drawing a polygon

Use an object

Right click on the cloud “Copy Tunnel” and click **Show only** to begin the second part of the exercise. The file also contains a cylinder, which is the best cylinder extracted from the point cloud of the tunnel. Display it in the scene.

Select the cloud “Copy Tunnel” and the cylinder and go to Cloud \ Separate with Object.

Use the option **Separate using tolerance** and set the **Tolerance value** to 0.2. Press **Preview** to preview the result. All the points located at a distance smaller than 0.2 meter from the cylinder are highlighted in a different color.

You can check the option **Keep points near the object** and uncheck **Keep points far from the object** to directly delete the points farther than the given tolerance, or check both options to divide the cloud in two sub clouds.

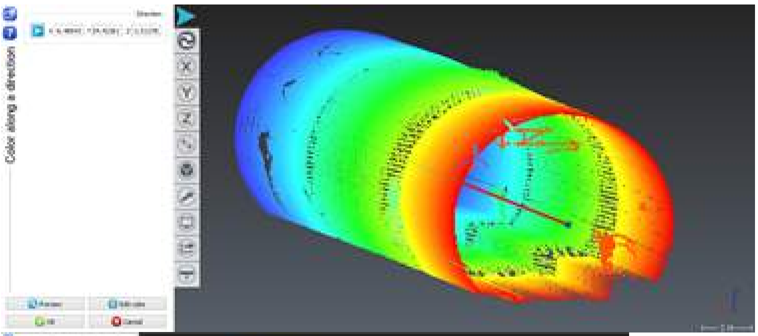
Separate a cloud with an object

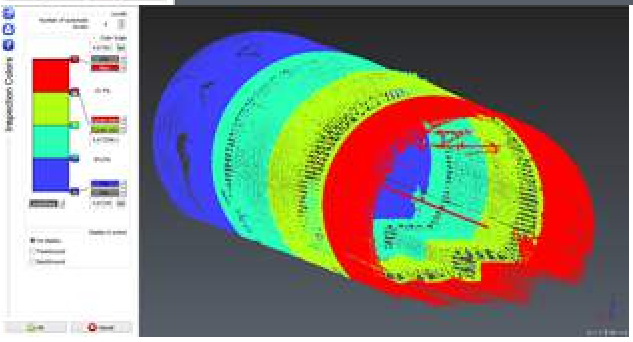
Use colors

It is possible to split a point cloud by coloring it along a direction and then explode it depending on the colors. We are going to divide the tunnel into four independent parts.

First, we are coloring the point cloud along the central axis of the cylinder used previously. Display the polyline called “Central axis” in the scene. Select the point cloud of the tunnel and go to Measure \ Color along a Direction. Choose **direction of a component** (

) and click on the central axis. An arrow is displayed in the scene to represent the direction. When you click on **Preview**, the point cloud is colored along this direction with a color gradient. Click on **Edit color** to modify the colors. In the upper part of the dialog box, set the number of color levels to 4 and click **OK** to validate the color map. Click **OK** again to validate the colored cloud.





Now we can explode the point cloud according to the colors we have just set. Select the point cloud and got to Cloud \ Inspection Colors. The four sub clouds are added into the Measure Group and you can work on each one separately.

5.2.2 Exercise: Clean a cloud using automatic filters

The software provides several automatic filters in order to clean up a point cloud and remove noisy points.

Automatic Filter: Noise

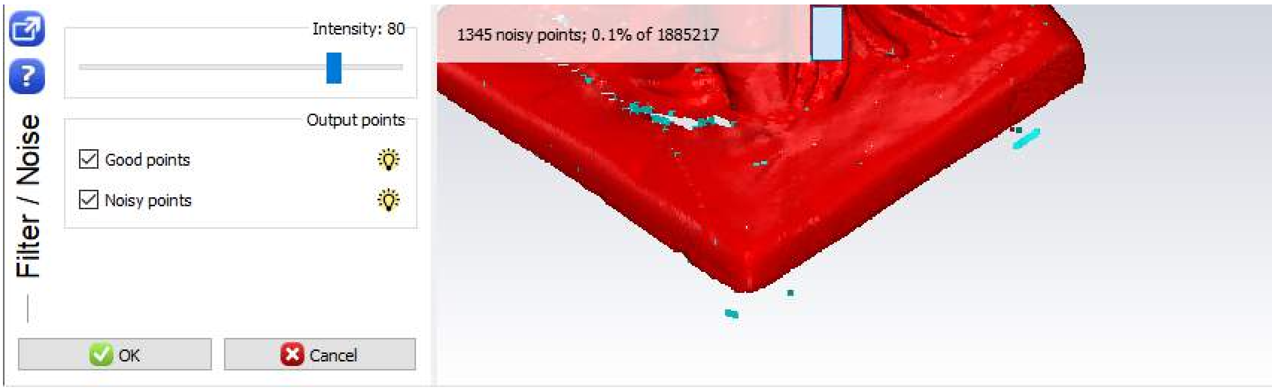
Open the file Victory.rsh.

This file contains 2 point clouds, one with noise, and one already filtered. Show only and select the cloud named “Victory + noise” and go to Cloud \ Noise.

If the goal is to delete sparse points inside the cloud, select the first option **Keep only good points**. The points involved are highlighted and displayed in a different color. You can adjust the slider to remove more or less points. In this example we can put the **threshold** to 80 in order to remove points which are in fact measurement errors (blue points in the picture). A text appears in the dialog box to notify you how many points will be deleted.

Note

In order to work correctly, this filter requires a point cloud with a regular density.



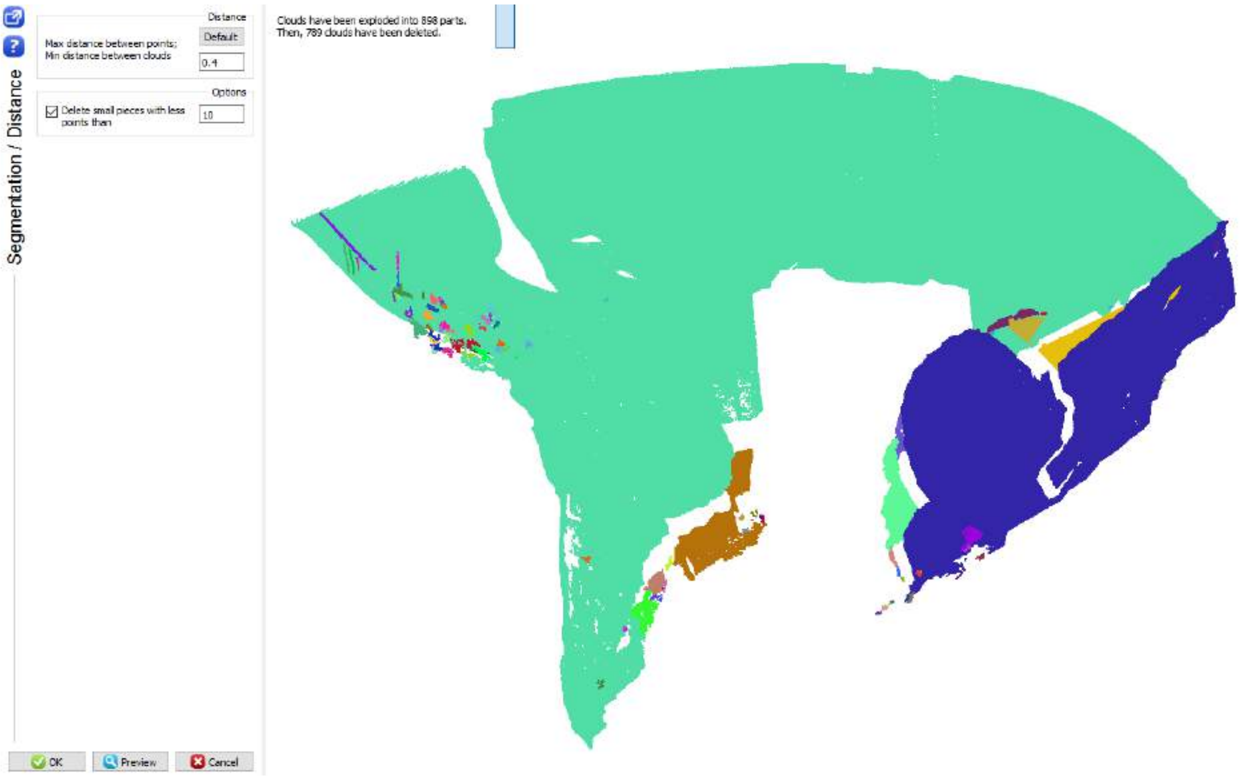
Remove noisy points in a cloud

Automatic Segmentation: distance

Open the file CleanPointCloud.rsh in BGG Surveying folder.

This file contains the point cloud of a dam having noisy and undesired parts. Select the cloud named “DamRock” and go to Cloud \ Distance.

Use this filter in order to split the cloud in smaller clouds and isolate the part of the dam. The cloud is split according to the maximum distance between points. This distance also corresponds to the minimum distance between sub clouds. You can compute a first value by clicking on **Default value** and preview the result. Then, you can change this value to fit your needs. Set the parameter to 0.4. You can delete automatically the small clouds with the option **Delete small pieces with less than** 10 points. Click on **Preview** to preview the results and **OK** to validate them.



Explode a cloud with a distance criterion
All the sub clouds are added into the Cloud Group and ordered from biggest to smallest regarding their number of points. If the cloud is exploded in more than 1000 parts, the smallest will automatically be deleted in order to keep only the 1000 biggest sub clouds.

Refer to the Cloud \ Distance help for more information about the distance filter.

5.2.3 Exercise: Reduce a point cloud

Two different filters are available to reduce a big point cloud in order to work with less data:

- By keeping a certain number of points: Reduction
- By keeping best points evenly spaced: Regular Sampling

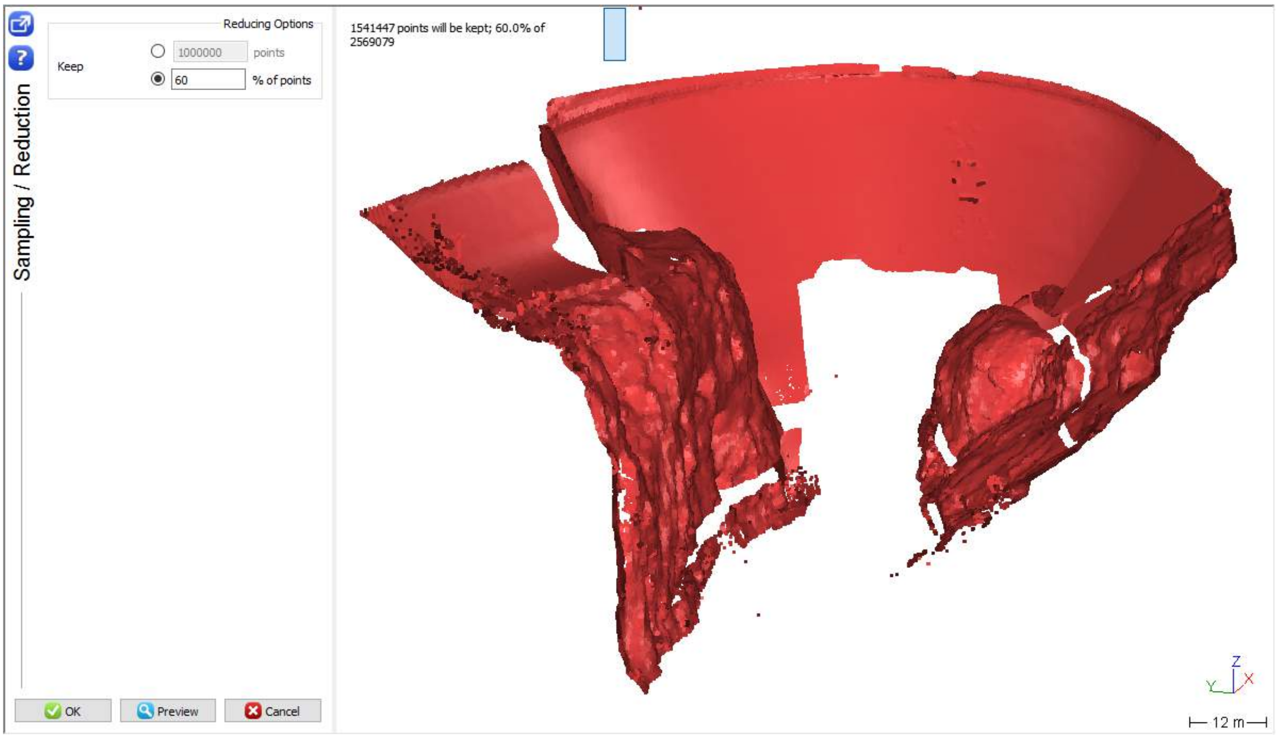
Load the file CleanPointCloud.rsh.

Reduction (Keep a certain number of points)

Select the cloud "DamRock" and go to Cloud \ Reduction.

With this filter, the number of points to keep has to be fixed and points will be deleted in high density areas. It is the same method as the reduction of a cloud during the import.

In our example, the main cloud still contains about 2.5 million points. If we want nearly 1.5 million points, the cloud can be reduced to keep only 60% of the points.



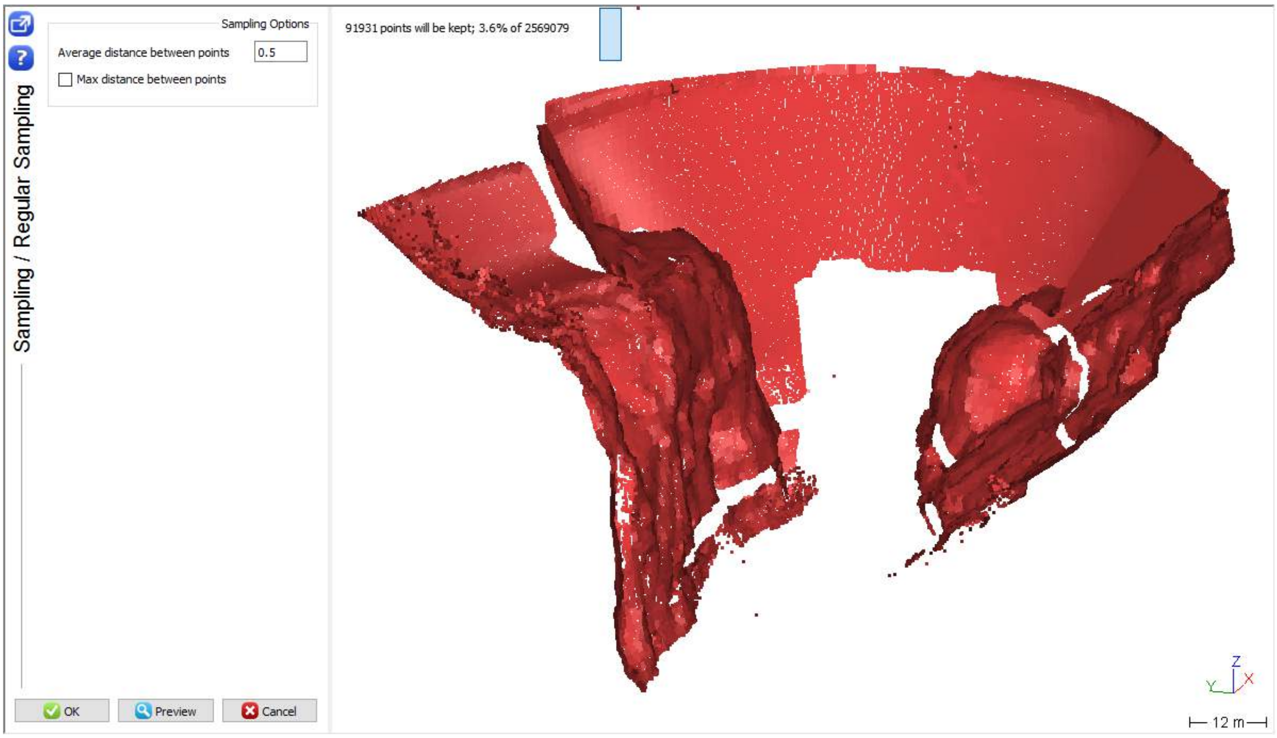
Reduce a point cloud keeping a certain number of points

Regular sampling (Keep best points evenly spaced)

Another way to reduce a big point cloud is to keep only the best points. With this filter, only the best points evenly spaced will be kept. The average distance between points gives the size of a grid that will be projected on the point cloud. One best point is computed for each cell, taking into account all the points inside the cell. If the density of the points is too low, the size of the cell is automatically increased. This size can be limited with the option **Maximum distance between points**.

You can use the point cloud called “Copy DamRock” to test this filter. Display it in the scene, select it, and go to Cloud \ Regular Sampling. Enter 0.5 as the **Average distance between points** and click on **OK**.

The initial cloud had nearly 2.5 million points and the reduced cloud contains only a bit less than 92 000 points. You can verify the number of points in the cloud with the **Properties** option in the contextual menu.



Reduce a point cloud keeping best points

6 Alignment - Registration

- Align clouds together
  - Exercise: Best fit between clouds with overlapped area
  - Exercise: Align clouds according to specific points (surveying target centers)
- Move an object to a coherent coordinate system
  - Exercise: Move to the Coordinate System (CS) of a 3D model
- Move to a reference model
  - Exercise: Align a point cloud on a reference model according to the shape (Best Fit)

6.1 Align clouds together

In most cases, you have to align all your scans in the same coordinate system based on the shape of the scanned object or based on particular points. It's the case, for example, with surveying targets.

For a proper alignment based on the shape (best fit), your scans must meet two criteria:

- each scan must share some overlapping areas with adjacent scans
- each scan should contains at least one change of shape (e.g., corner, hole and angle).

For a proper alignment based on particular points, you must have at least three common points in the two scans

- Exercise: Best fit between clouds with overlapped area
- Exercise: Align clouds according to specific points (surveying target centers)

6.1.1 Exercise: Best fit between clouds with overlapped area

Open the file BestFitClouds.rsh.

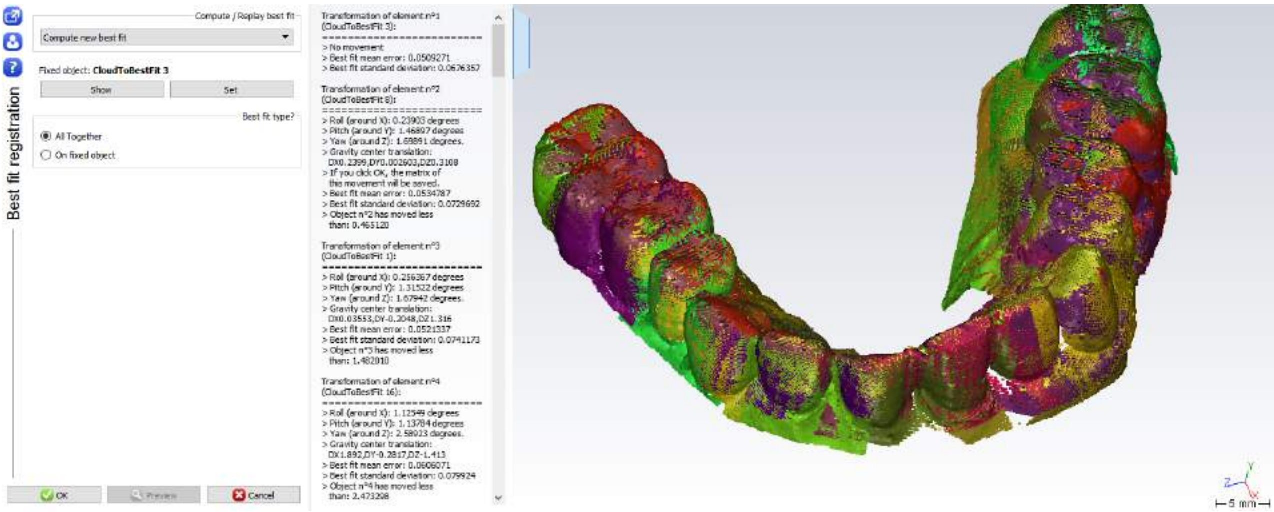
This file contains 32 clouds with overlapped areas. If you zoom in, you can see that all the clouds are not correctly aligned.

Select all of them (press **CTRL-A**) and then go to Transform \ Best Fit Registration.

Set the options:

- Select **Compute new best fit**: we don’t want to replay a previous best fit.
- Select **All together**: as there is no reference cloud (a cloud that has common parts with all the other clouds), we say that all the clouds should be aligned with all the other clouds.
- Disable all options in **Advanced mode**.

Then click **Preview**; you will see a dialog on the right (report) in order to see transformations applied on each object, such as the registration standard deviation error and the registration mean error of each component. You can copy-paste this text if needed. Note that the first cloud will not move because we need to keep one object immobile in order to converge to a solution.



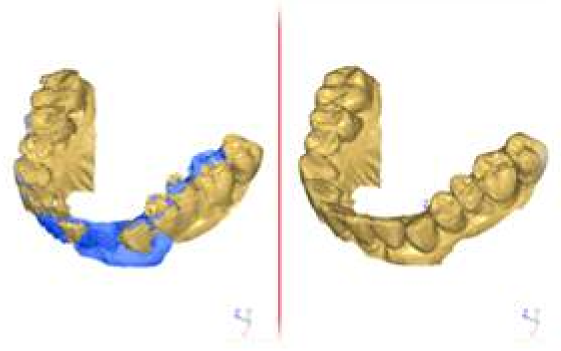
Result of best fit (4 first clouds)

> **Note**
>
> If you want to preserve the orientation on one or several axes, you can use the option **Define constraints**. This may be useful in the case of scan files having the Z axis already aligned with the vertical.

Click **OK**, all the clouds are now aligned.

On the next pictures, you can see why the best fit is important. These two meshes have been created with same parameters:

- On the left, no best fit has been done. There are a lot of holes, reverted normal (blue color) and in some parts there are several layers.
- On the right, we did a best fit alignment. The normal are OK everywhere; no holes, and the mesh is perfectly smooth.



Comparison between two meshes with/without best fit alignment

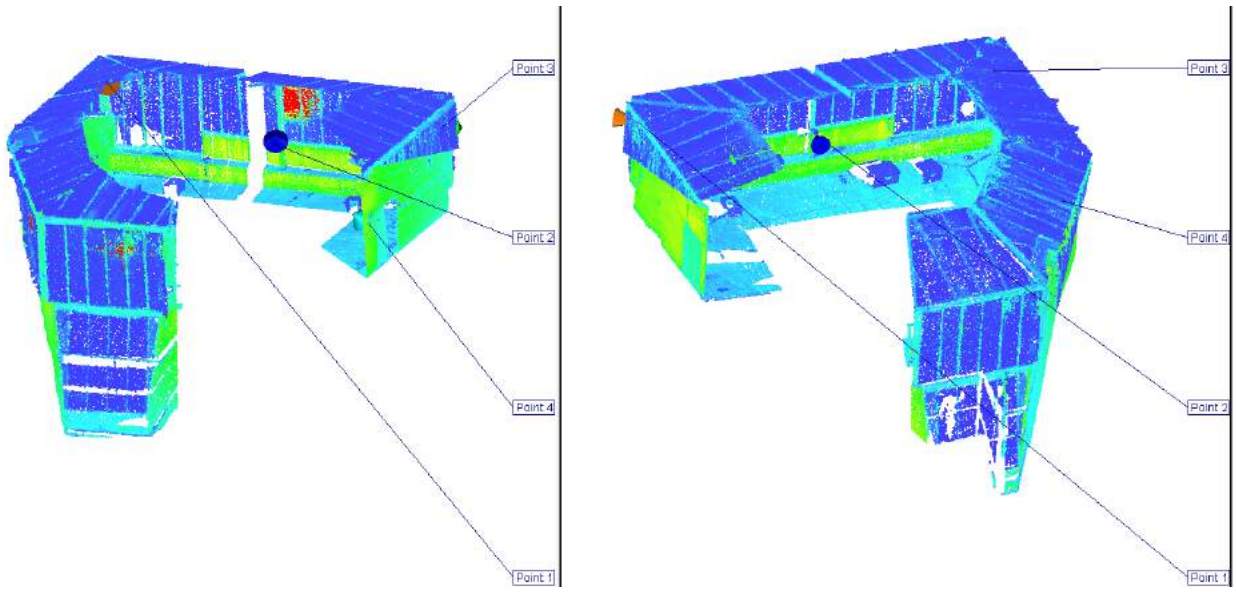
6.1.2 Exercise: Align clouds according to specific points (surveying target centers)

Open the file AlignTargets.rsh.

This file contains two clouds to align according to surveying targets "Cloud 1" and "Cloud 2". Show both clouds, then select "Cloud 2" and go to Transform \ Best align N points.

The screen is automatically divided into two parts. On the left, there is the selected cloud (the one that will move); on the right, the other one (the one that will not move). In the top ribbon, select only the option **Surveying target** and set a **checkered pattern target** with a **Diameter** of 0.15.

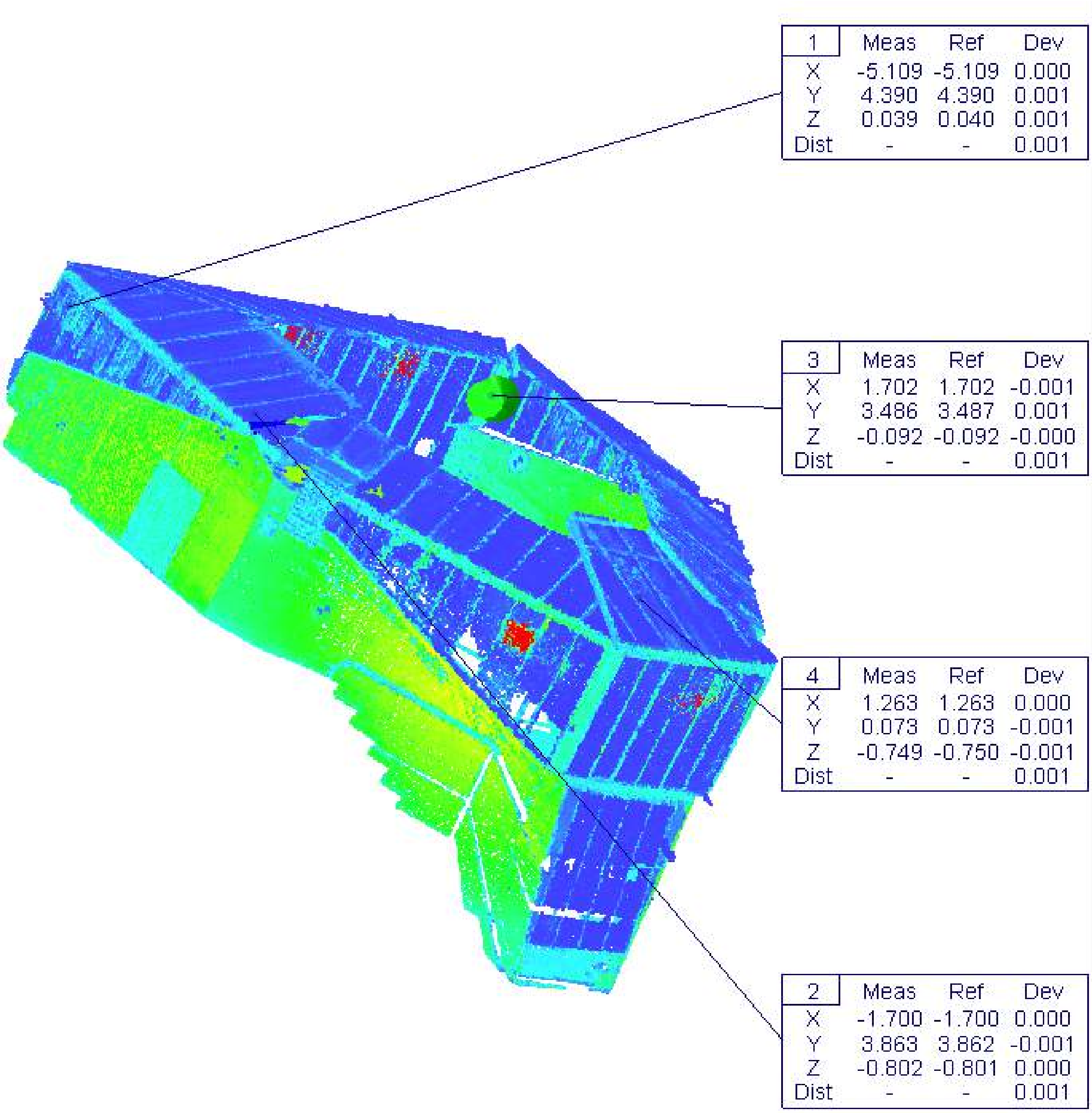
Select the option **Define constraints** and then select the option **Preserve orientation of the Z axis** in the list box below as the Z axis is aligned with the vertical in both clouds. Check the options to allow translations along the three axes.



Use target centers to align point clouds
Find the target corresponding to the point 1 on the previous figure and then click a point on the target in the left view. The created point will be automatically the target center. Repeat for the three other targets (points 2, 3 and 4). Then click the same targets in the same order in the right view.

Unselect the option **Apply best fit** as we want to do an alignment according to targets only.

Once the four couples of points have been clicked, you can click **Preview**.



Labels to show deviations during the alignment according targets (less than 1mm in this case)
One label per couple of points will be created in the right view in order to see deviations. Identify the couple with the biggest deviation. Cancel the alignment by clicking **Reset**. Select the mode **Vertex / End** and click on the label corresponding to the biggest deviation (left or right view) in order to inactivate this couple of points. The arrow on the target becomes gray. Click on **Preview** again. The deviations should be lower and the alignment is done with only 3 couples of points.

Click **OK** to validate. Both clouds are now aligned: you can use the command Cloud \ Merge Clouds clouds in order to merge them. Note that the color of your cloud might change during the merge because the cloud contains both texture and inspection information. You can select the cloud to restore the I**nspection** representation then go to Measure \ Edit Colors to take only one color level.

Note: if you make a mistake during the selection of target, you can reverse step by step by pressing the DEL key.

6.2 Move an object to a coherent coordinate system

Even if the measure does not need to be placed in a reference coordinate system, you may have to change the current coordinate system in order to make it coherent with the object.

For example, on a facade, you may want to have the X axis orthogonal to the wall.

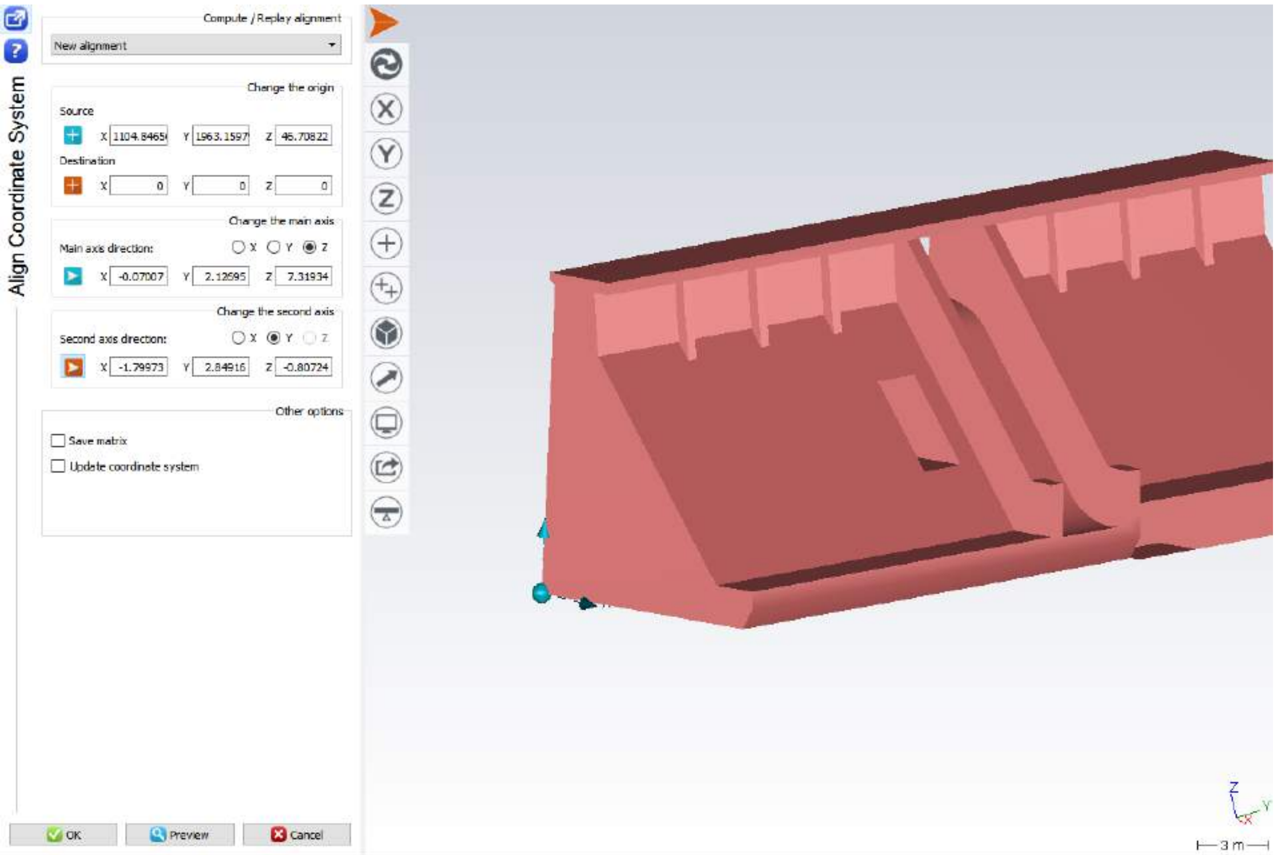
- Exercise: Move to the Coordinate System (CS) of a 3D model

6.2.1 Exercise: Move to the Coordinate System (CS) of a 3D model

Open the file BestFitOnRef.rsh.

This file contains two meshes "Theoretical Dam (bad CS)" and "Theoretical Dam (good CS)". Select and show only the mesh "Theoretical Dam (bad CS)" (you will probably need to type **A** to make a Zoom All). Then press **X**, **Y** or **Z**. You can see that the axes of the coordinate system are not coherent with the mesh. Now select and show only the other mesh "Theoretical Dam (good CS)". Press again **X**, **Y** or **Z**. Now the axes are coherent.

Show only the mesh "Theoretical Dam (bad CS)", select it and go to Transform \ Align Coordinate System.



Move to the coordinate system of the dam

You have to respect an order when using the command. So first, click

and use the option **Vertex / End** to place the **Source point** on the bottom left corner of the dam. A new dashed coordinate system will appear where you have clicked the point (see above). Keep (0, 0, 0) for **Destination point** as we want to have the bottom left corner of the dam as the origin of the new frame.

Then, change the **Main axis direction**. Choose **Z**, and click

to define the new Z axis. For example, use

and click 2 points on a vertical edge.

Finally, define the **Second axis direction** in a similar way by choosing **Y** and clicking on

. For example, use

and click 2 points on a horizontal edge.

You can now click **OK**. Press **A** to do a zoom all, and then **X, Y** or **Z** on the keyboard. You will see that all the axes are now coherent with the model.

Important

When using Transform \ Align Coordinate System, the selected objects will moved to the new position. The 3D coordinates of the object are updated. This command differs from Construct \ User Coordinate System command.

6.3 Move to a reference model

Each time you have to do a comparison or an inspection, you have to put your measures in the reference coordinate system. To do this, there are two methods:

- An alignment according to the shape to minimize distances between the measure and the reference.
- An alignment according to the geometry, generally used in mechanical field, when constraints are defined by circles, lines or planes.

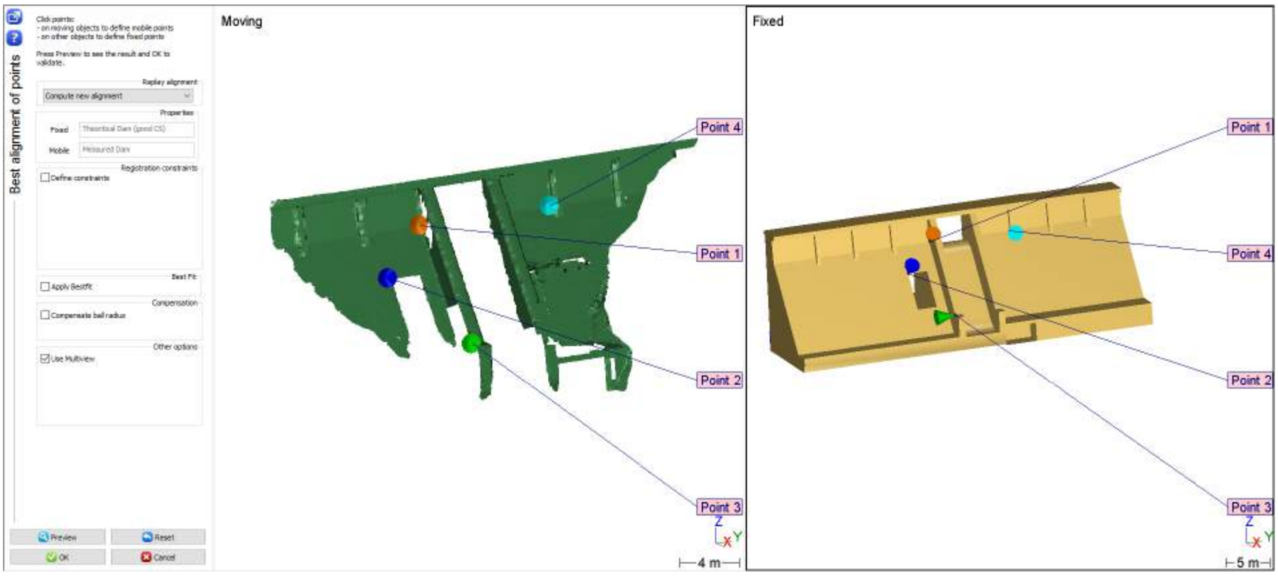
- Exercise: Align a point cloud on a reference model according to the shape (Best Fit)

6.3.1 Exercise: Align a point cloud on a reference model according to the shape (Best Fit)

Open the file BestFitOnRef.rsh.

This file contains a cloud "Measured Dam" and a mesh "Theorical Dam (good CS)". We will see how to align the cloud on the mesh.

Show only these two objects. Select the point cloud only and launch the command Transform \ Best align N points.



Enter points during the Best Align N Points command

The screen will be divided in two parts:

- On the left, the selected objects, in this case the cloud "Measured Dam". All objects on the left will move.
- On the right, the non-selected objects, in this case the mesh "Theorical Dam (good CS)". All objects on the right will **not** move.

Select the option **Compute new alignment** in the list. Then, enter couples of points in order to do a rough alignment by combining a point on the cloud and a point on the mesh. To realize this step, select the option **Point on selection**. First, click on a point on the cloud and then the corresponding point on the mesh. Click at least 3 couples of points (you can click on edges to make it easier). To have a good result, your points should not be aligned and all over the object (see Figure above).

Once you have entered your couples of points, click **Preview**. The cloud will move in the right view in order to see if the rough alignment is correct. One label per couple of points is also created in order to see deviations.

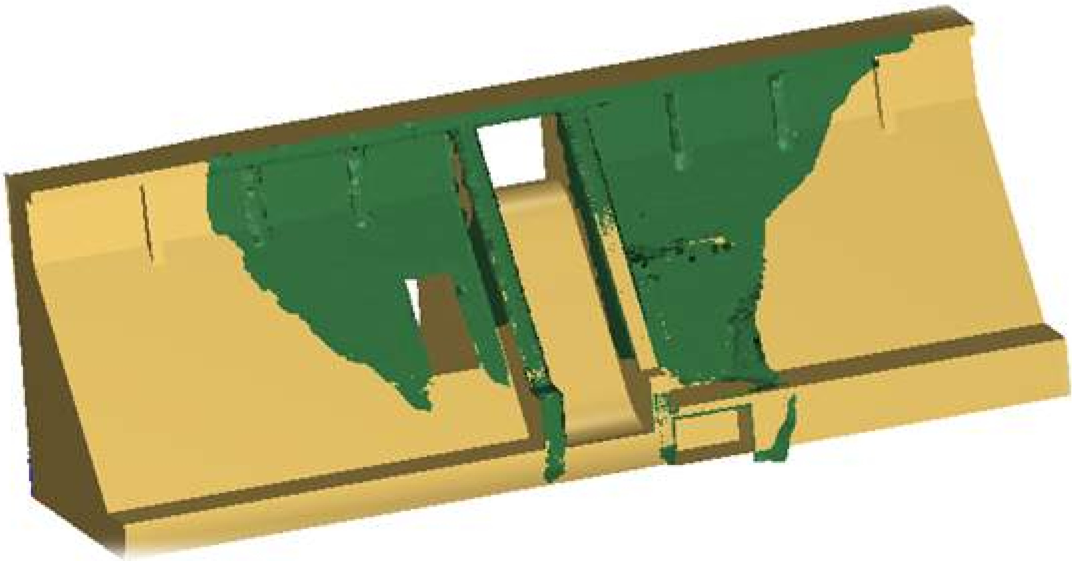
If the result is not correct, press **Delete** twice to remove the last couples of points and enter a new couple of points. You can also erase all current points with **Reset**.

If the result is correct, select now the option **Apply Bestfit** and click **Preview** again. The software has now computed a best fit in order to minimize deviations between the mesh and the cloud. Next to the dialog box a small report is displayed in order to summarize all the transformations.

Note

You can define some constraints during this alignment. For example, if the Z axis is correct on the cloud, select the option **Define constraints** and then choose the option **Preserve the orientation on the Z axis**. It means that rotation around X and Y will be disabled.

Click **OK** to validate. The cloud is now in the reference coordinate system, and you can for example proceed to a comparison.



Cloud and mesh aligned with the best fit method

7 Meshing and mesh improvement

The software allows you to create a model by meshing point clouds. This operation has several goals:

- Get an accurate surface model of your measured object.
- Control the quality of your digitalization (precision, lack of points, etc.).
- Keep only the most relevant points of your digitalization and thus reduce the model.
- Improve the accuracy of the result by eliminating incoherent points, filtering and/or smoothing the mesh.
- Be able to export the result of your digitalization in other software even if this software is unable to process files of several millions of points.
- Process reverse engineering.
- Reproduce the digitalized part: machining or making with rapid prototyping is possible with a mesh, but not with a 3D point cloud.
- Make 3D presentations, animations, photo-realistic rendering.
- Make finite element computations.
- …

The software has various tools to mesh your point clouds. Few parameters are enough, so that it makes the processing nearly automatic. This operation is extremely fast even if you have a large number of points. Thus, you will be able to make several attempts with various parameters until obtaining the desired result.

- Mesh creation
  - Exercise: Create a 3D mesh of the Samothrace Victory
  - Exercise: Create a 2D mesh from a cloud digitalized in top view
  - Exercise: Extrude a profile
- Mesh improvement
  - Exercise: Improve the 3D Mesh of the Samothrace Victory
  - Exercise: Merge meshes with common borders
  - Exercise: Merge meshes with different borders
  - Exercise: Improve global aspect and edges
  - Exercise: Fill holes with curvature filling
  - Exercise: Reconstruct perfect holes on a mechanical part
  - Exercise: Apply the color of a point cloud on a mesh

7.1 Mesh creation

This section shows the difference between:

- The 3D mesh technic
- The 2D mesh technic
- The meshing technic by extrusion of a contour along a path

- Exercise: Create a 3D mesh of the Samothrace Victory
- Exercise: Create a 2D mesh from a cloud digitalized in top view

- Exercise: Extrude a profile

7.1.1 Exercise: Create a 3D mesh of the Samothrace Victory

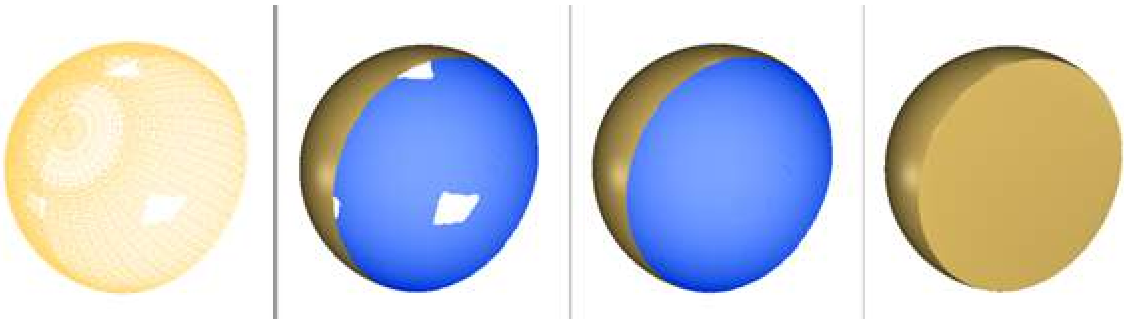
Open the file Victory.rsh.

This file contains two point clouds: one **Victory + noise** with some measurement noise, and one **Victory** already filtered with the command Cloud \ Noise and ready to mesh (see Exercise: Clean a cloud using automatic filters).

Explanation about hole management

Whatever the selected meshing method, options concerning the hole management are the same. You have the choice between three modes:

- Hole detection: enter the size of the smallest hole you want to keep.
- Try to keep the external border: all the holes will be filled, except the external border.
- Try to create a watertight mesh: the result will be a closed mesh (without any holes).



Holes management during the 3D mesh
From left to right:

- the cloud to mesh
- hole detection
- keep only the external border
- try to create a watertight mesh

You can find more information about holes in the Exercise: Fill holes with curvature filling.

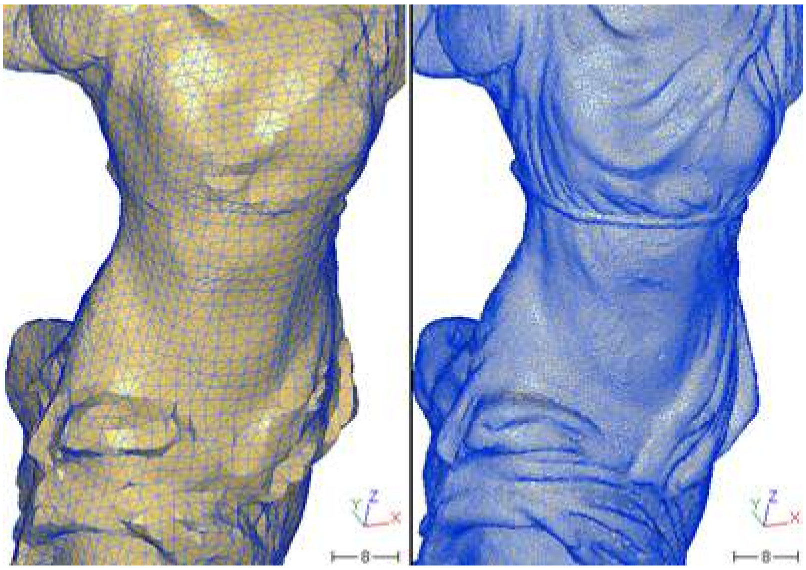
Regular Sampling

Select the cloud **Victory** and launch the command Mesh \ 3D Mesh. Then, select the option **Regular Sampling** and **Try to create a watertight mesh**. The field **Average distance between points** is filled automatically according to the point cloud properties (size, number of points, etc.). Then click **Preview** to visualize the 3D mesh.

The regular sampling method will project a grid on the cloud and select the most representative point inside each square of the grid. Then it computes a 3D mesh from all the selected points.

Change the representation mode to **Flat + Wire** in order to see triangle shapes. You will notice that they are quite regular and equilateral. The value set for the **Average distance between points** roughly determines to the size of the grid projected on the cloud. In this case, the value is approximately 0.6 and corresponds to the average distance between vertices.

Change the **Average distance between points** to **2**, so the grid will be 3 times bigger than before. Have a look at the triangle shapes: they are quite big but still regular.



Regular Sampling: this method creates regular meshes where the size of the triangles depends on the average distance between points
In some cases, a regular sampling is not the best choice:

- if the level of details is not the same all over the mesh,
- if you enter a small average distance on a cloud with some measurement noise, or
- if the final result appears too facetted.

In these cases, you can use the **Meshing in two steps**.

Meshing in two steps

The goal of this method is to create a mesh in two steps:

- first, we create a rough mesh in order to get the global shape without any holes or errors,
- then, we stretch this rough mesh according to the point cloud in order to add all the details.

Select the cloud **Victory** and go to Mesh \ 3D Mesh. Then, select the option **Meshing in two steps** and **Try to create a watertight mesh**. The field **Average distance between points** is filled automatically according to the point cloud properties (size, number of points, etc.). Then click on **OK** to compute the first rough 3D mesh.

The dialog box corresponding to the second step will be opened automatically once the rough mesh is computed. It corresponds to the command Mesh \ Refine Mesh \ From a point cloud.

There are two meshing methods:

- **Take points of the clouds**; will give you better results if the point cloud contains only precise points and if you want to preserve sharp edges.
- **Interpolate new points**; if your point cloud contains a lot of points and/or noisy points (measurement errors), it is strongly advised to interpolate new points.

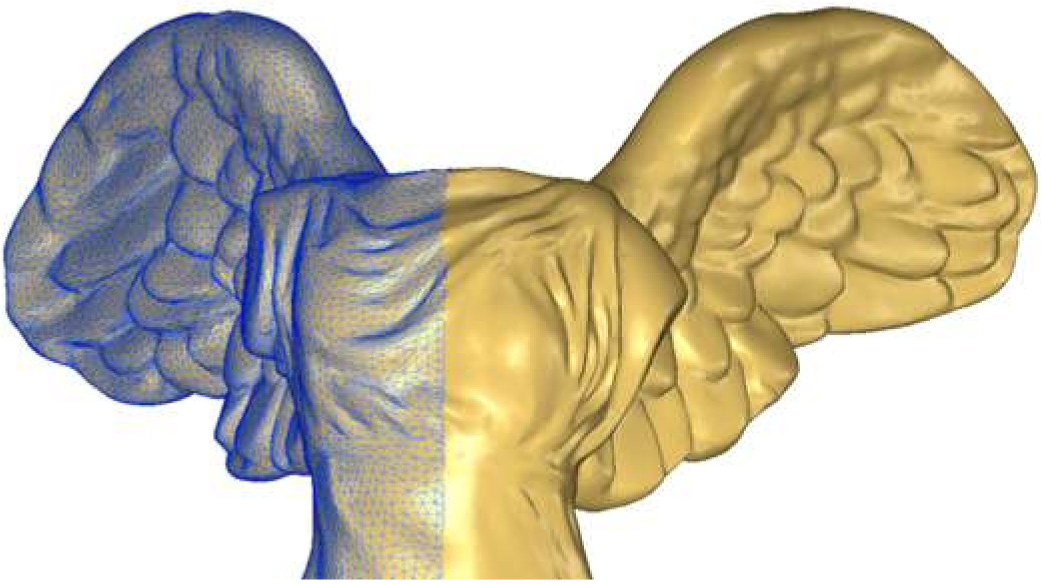
Some parameters are common to both methods:

- **Outlier point distance**; to reject the points located too far away from the polyhedron you can enter **1** .
- **Local Reorganization**; to give a better mesh of sharp angles and small fillets you can select it.
- **Holes management**; as we have a closed mesh, select **No free border modification**.

Select the option **Interpolate new points** and **Refine with deviation error** as we do not care to have points evenly spaced. This option will create new points according to an estimation of the best shape to create. Then set the **Deviation error** to **0.05**, it means that the maximum distance between the mesh and a “perfectly smooth” surface will be less than 0.05. There are 2 others parameters in order to control the refinement:

- **Maxi number of triangles**; in order to avoid having too many triangles in the mesh. Enter 1 million.
- **Minimum triangle size**; in order to avoid very small triangles in the mesh. Enter 0.5, this value should be bigger than the deviation error.

Click on **Preview** to compute the refined mesh. Once it is finished, you can have a look at the triangle shapes by changing the representation to **Flat + Wire**: triangles are not regular now, size depends on the details.



Refine by interpolating new points

Now change the refine parameters and use the **Take points of the cloud** option and **Deviation error with best points only** as there is still some noise in the cloud. Enter 0.05 for the **Deviation Error** in order to compare with the previous method and click on **Preview** to refine the mesh.

As you can see in the next picture, the mesh is spikier, so we added some noise to the mesh. In order to avoid this, we should try again with a bigger deviation error. This noise does not appear with the method **Interpolate new points** because it has been reduced during the computation of the new points.

Comparison between the two refinement methods; Left: Interpolate new points: Right: Take points of the cloud

In order to avoid spiky results, the deviation error must always be bigger than the scanner accuracy when you use the option **Take points of the cloud**.

7.1.2 Exercise: Create a 2D mesh from a cloud digitalized in top view

Open the file 2DMesh.rsh

This file contains a measurement of a digitalized half glasses scanned from above. The measurement process carried out a deviation error simplification every time the trajectory of the measurement probe was rectilinear. It results in a highly non-homogeneous density point. If you make a rotation of the model, you will be able to note that the vertical walls contain very few points or even no point at all. The horizontal plane contains disseminated points and the “shape” part contains a high density of points.

For this kind of cloud, the quickest way to create the mesh is the command Mesh \ 2D Mesh.

Select the point cloud and enter the command. Then use **Z** as meshing direction (the cloud has been measured in top view) and enter 0.2 in the field **Deviation error**.

> **Note**
>
> If you enter 0 for the **Deviation error**, all the points will be meshed.

Click **OK** to compute your mesh. You can cancel the next dialog box Mesh \ Find contour restrictions. The result obtained is easier to analyze if you select the **Flat** or **Flat + Wire** display mode: there is no hole on the mesh, but triangles are highly stretched along the meshing direction.

To obtain a nearly perfect result, we can smooth the model with a **Deviation** authorizing very small displacement of the points to avoid accuracy deterioration, for example 0.0001. You can also make a smoothing with an intensity of zero to make only a reorganization of the triangles.

> **Note**
>
> If you want to create 2D meshes from points and polylines, you can use the command Mesh \ Constraint meshing in order to add polylines as breaking lines or feature lines in your mesh.

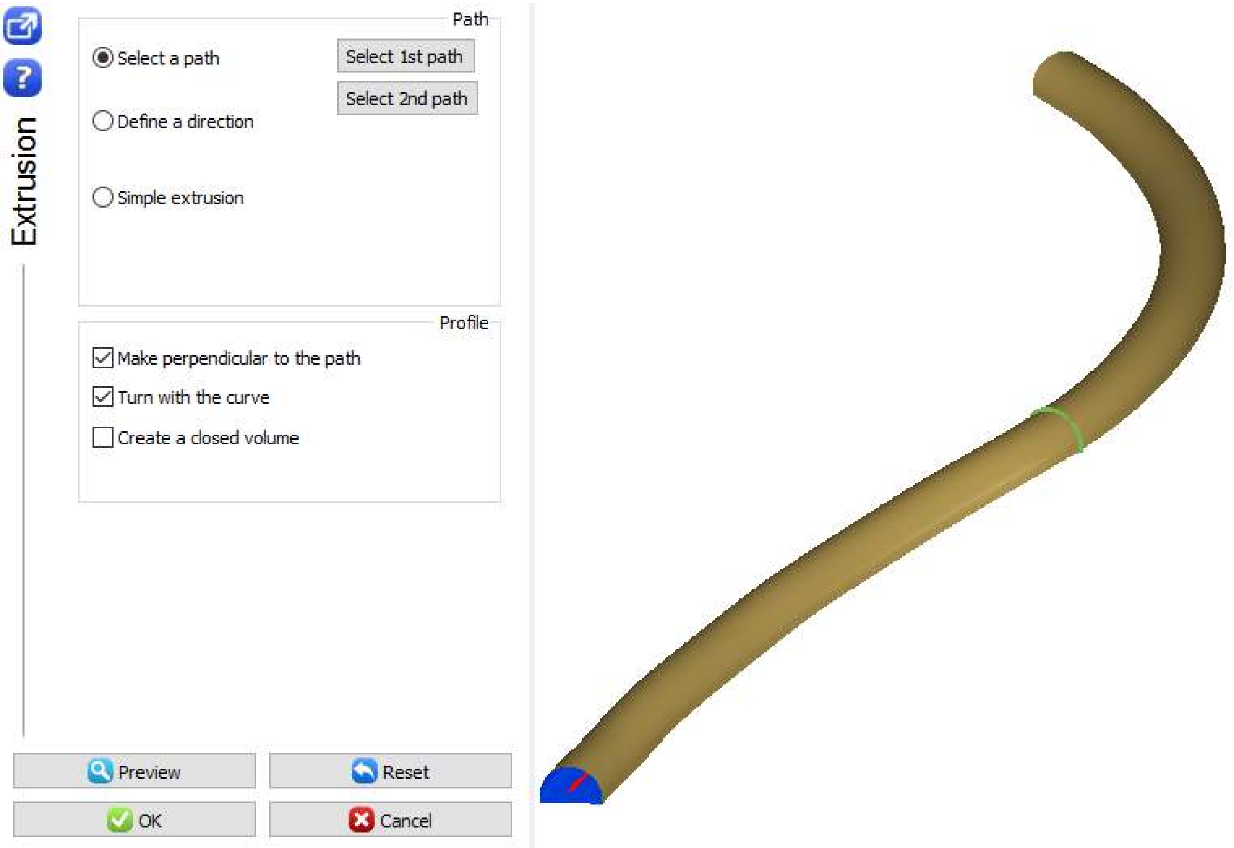
7.1.3 Exercise: Extrude a profile

> Open the file CrossSections.rsh.

This file contains a theoretical section as well as the neutral axis of a tunnel.

Select the Theoretical section and go to Mesh \ Extrusion. Click the button **Select 1st path** and click on a point on the neutral axis. Unselect all the options and click **Preview**. The result is not as expected because the profile has moved along the path but it has not turned with the path (only translations). Select now the option **Turn with the curve** and click **Preview** again. A theoretical tunnel is now created as expected.

> **Note**
>
> - The option **Create a closed volume** will close the two holes at the beginning and at the end of the extruded mesh.
> - The option **Make perpendicular to the path** will move the profile in order to be perpendicular to the first path vector (not necessary here as the section is already perpendicular to the neutral axis).



Extrusion along a path

7.2 Mesh improvement

In the software there are many tools in order to improve meshes. The main ones are described in this chapter.

- Exercise: Improve the 3D Mesh of the Samothrace Victory
- Exercise: Merge meshes with common borders
- Exercise: Merge meshes with different borders
- Exercise: Improve global aspect and edges
- Exercise: Fill holes with curvature filling
- Exercise: Reconstruct perfect holes on a mechanical part
- Exercise: Apply the color of a point cloud on a mesh

7.2.1 Exercise: Improve the 3D Mesh of the Samothrace Victory

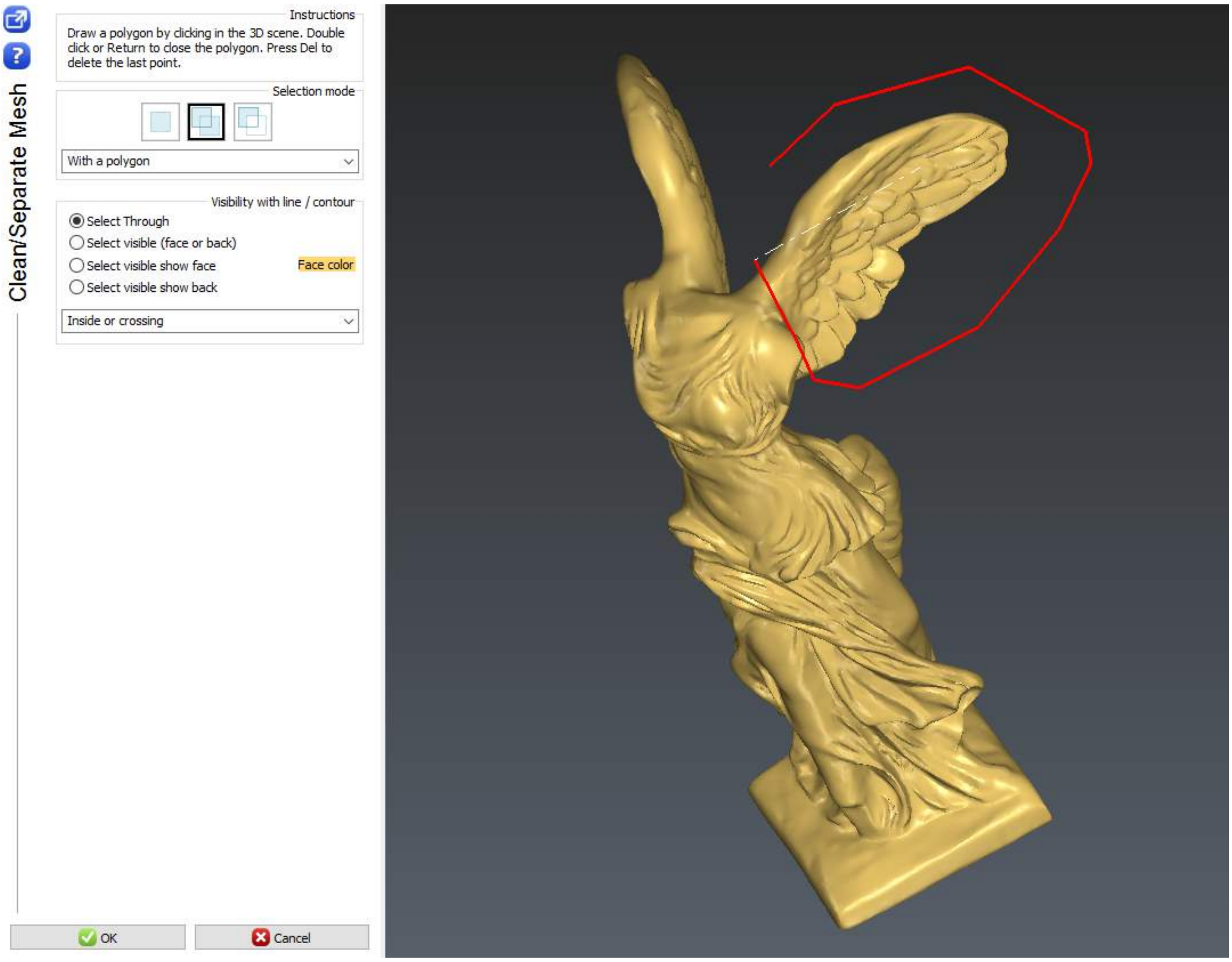
Open the file Victory.rsh.

Then show only the mesh **Victory (complete)**. Throughout this exercise, we will see how to cut a mesh and how to work independently on different parts of the mesh (by reducing a part and, on the contrary, refining another one).

Cut / remove triangles

Select the mesh **Victory (complete)** and go to the command Mesh \ Clean / Separate. This command allows you to delete triangles or to cut your mesh in several pieces. Here we will cut the two wings in order to have 3 different meshes.

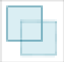
Set the selection mode as **With a polygon**, and then select the options **Select Through** and **Inside or crossing**. **Select through** means that we will select all triangles inside the polygon, even the hidden ones. Set the view in order to see the entire wing and draw a polygon around it (like in following picture).



Cut a part of a mesh by selecting triangles inside a polygon
Press **Enter** to validate the selection. Then do the same for the second wing.

You can choose **Inside a circle (pencil)** as the selection mode in order to adjust the selection. For example, if you have selected too many triangles, set the global mode to **Remove from the selection** and unselect triangles with the pencil.

Selection mode

- : New selection
-  : Add to current selection
-  : Remove from current selection

Once the two wings are selected, click **OK** to validate. Then click the button **Separate and keep the two parts** as we want to cut the wings. If you want to delete triangles, the good choice is **Delete selected facets** .

As we are creating several pieces of meshes, a pop-up will ask you what to do with all the different parts. Select the option **Explode the result in several meshes** to separate the thee parts (the two wings and the main part).

Refine mesh

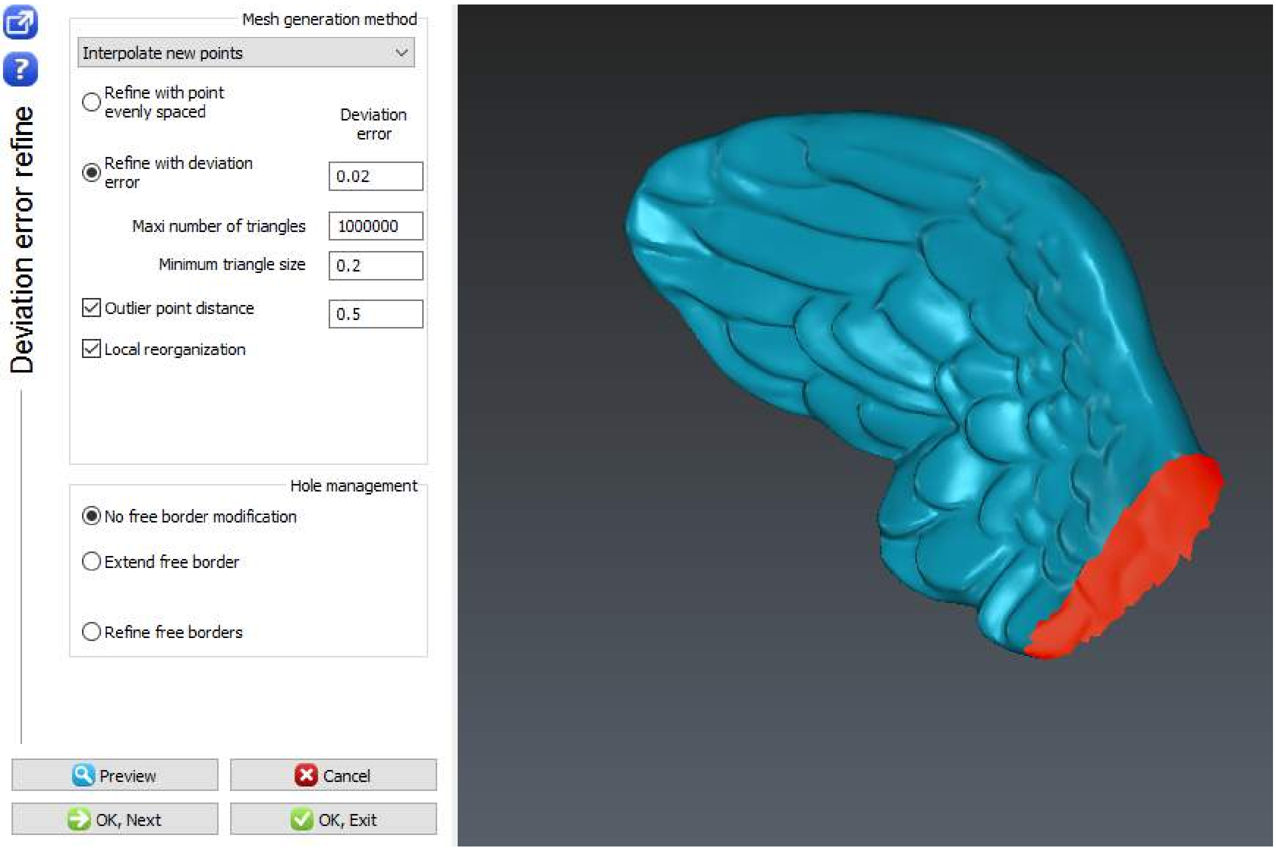
We saw in the 3D mesh creation chapter that we can do a mesh in two steps: create a rough mesh and then refine it in order to add details to the mesh. The command Mesh \ Refine Mesh \ From a point cloud corresponds to this second step.

Select both the cloud **Victory** and the mesh **Wing 1 (to refine)**, and then go to Mesh \ Refine Mesh \ From a point cloud. We have to select the cloud AND the mesh because we will refine the mesh with the points from the cloud. As you can see, it is the same command as the one described in the Exercise: Create a 3D mesh of the Samothrace Victory.

Parameters to obtain the actual mesh were **Interpolate new points** with 0.05 for the **Deviation error**. If we want to refine the mesh, we have to reduce this deviation error.

Enter for example:

- Choose **Refine with deviation error**
- 0.02 for the **Deviation error.**
- 0.2 for the **Minimum triangle size**
- 0.5 for the **Outlier point distance**
- **No free border modification**, so that we can very easily merge all the parts at the end



Refine with deviation error and interpolate new points

Click **Preview** and **OK, Exit** to validate. The wing is now refined. Change the representation to **Flat + Wire** for example, and compare the triangle size on the new wing and on the rest of the mesh. You will see that their sizes are smaller.

Reduce the number of triangles

Sometimes you need to reduce the number of triangles if:

- you want to export a very light mesh, or
- you do not need a lot of details on your mesh.

Select the mesh **Wing 2**. Then launch the command Mesh \ Reduce. This command contains two main options as there are two methods to reduce a mesh:

- **Control the deviation**: means that you enter a maximum deviation; the software will minimize the number of triangles and respect the given deviation.
- **Keep the aspect**: means that you enter a number of triangles; the software will reduce the mesh in order to respect this number.

Usually, the **Control the deviation** method gives a better result. So select this option, and then:

- enter 0.5 mm for the **Deviation,**
- select the options **Optimize vertices position** and **Try to preserve equilaterality,**
- unselect the option **Reduce on free borders** in order not to modify the external border, and

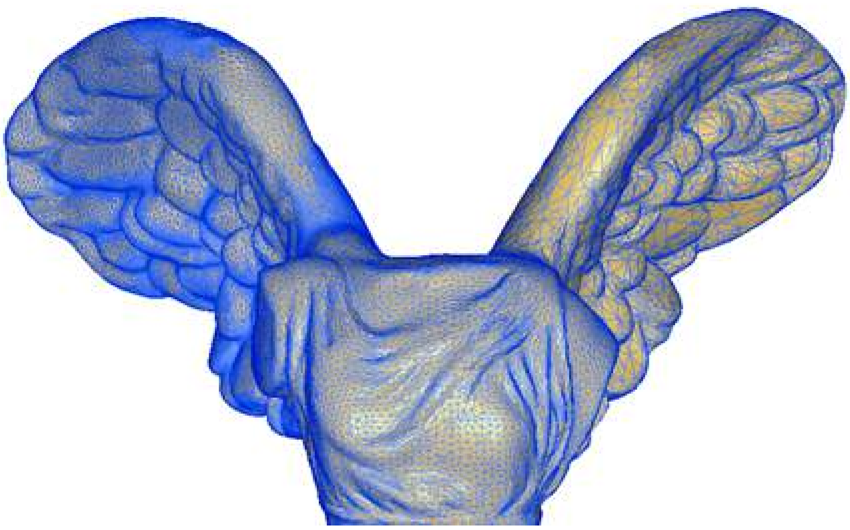
- select the option **Preserve sharp edges**, and set the **Angle between facets** to 10° and the **Specific deviation** to 0.05 mm.

Then click **Preview** to see the result. The given reduction rate is 55%, it means than we divided the mesh size approximately by 2.

7.2.2 Exercise: Merge meshes with common borders

Open the file Victory.rsh.

As the 3 meshes (**Main part**, **Wing1** and **Wing2**) have common borders, we can create a closed mesh (so a closed volume). To do so, select the three meshes then go to Mesh \ Merge common borders, select the option **Do not modify borders** and click **OK**. You will have only one mesh without any holes.



The merged mesh with 3 levels of details

7.2.3 Exercise: Merge meshes with different borders

Open the file MergeMeshes.rsh.

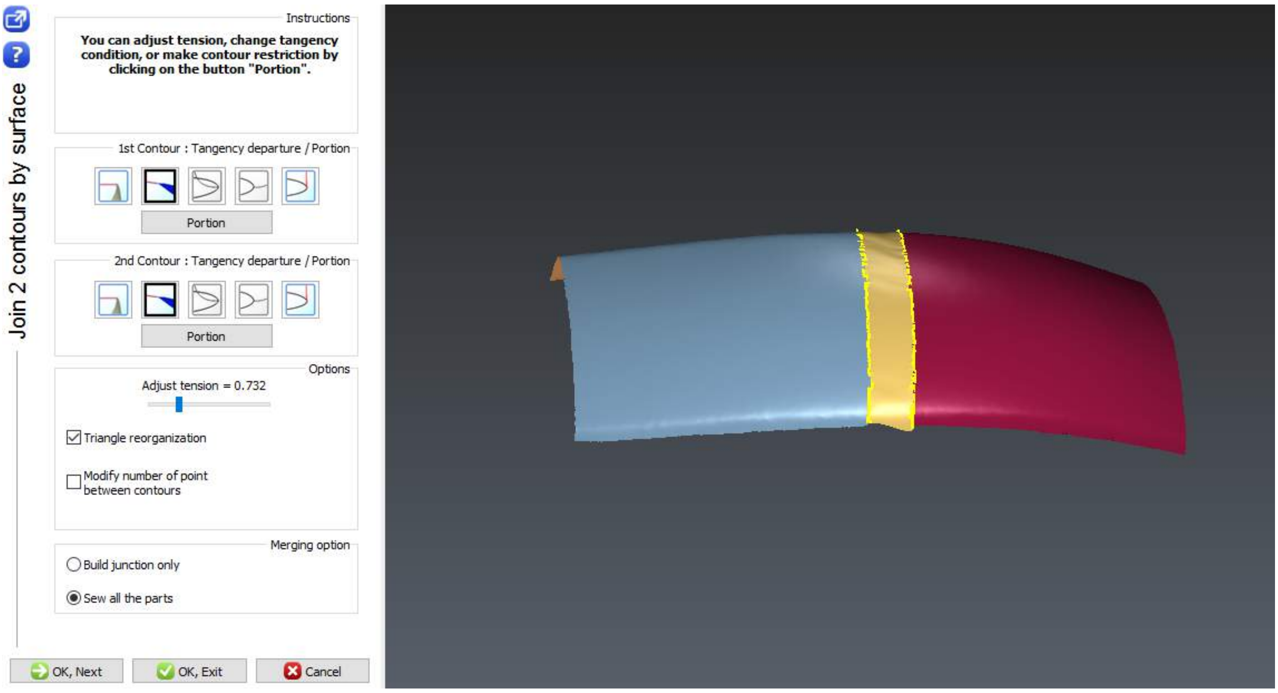
This file contains 3 meshes, and the goal of the exercise is to merge the meshes together.

With different borders and without overlapped area

Show only meshes **Part 1** and **Part 3**. As you can see, there is a gap between the two meshes. In order to merge them, we need to build a junction.

Go to Mesh \ Join 2 contours. Then click a point on each mesh. Choose to continue with the entire contour. The result is not the expected one because the complete borders have been joined, while we want the junction to be only on the middle part.

Click the button **Portion**, and then click on 2 points on the first contour in order to delimit the area. Then, click on a point on the wanted portion and press **Enter**. Do the same for the second contour.



Merge two meshes with different borders and without overlapped area
Then you can click the buttons in order to change the tangency criteria (buttons with pictures above **Portion** button). The best choice in this case is **Tangent to the surface** (second button from the left). Select the option **Triangle Reorganization** in order to improve the result. Do not forget to select the option **Sew all the parts** in order to have only one mesh at the end.

Then click **OK, Exit** to validate. You can use the smoothing in order to improve the result on the sewing zone (see section Local smoothing in the Exercise: Improve global aspect and edges).

With different borders and overlapping area

Undo as necessary to retrieve the initial state of the file. Show only meshes **Part 2** and **Part 3**. As you can see, there is an overlapping area between the two meshes. In order to merge the meshes, there are three possibilities:

- Remove triangles to create a gap between the two meshes (use the command Mesh \ Clean / Separate), then apply the method described above **with different borders and without overlapped area.**

- Remove triangles to make the two borders strictly identical. To do that, select one mesh and the polyline **Section** and go to Mesh \ Constraint meshing. Select the options **No direction, 3D computation** and **Cut the mesh along the polylines**. Do the same with the second mesh. This command will cut the meshes along the polyline. Then, show only the 2 biggest parts, select the two biggest meshes and go to Mesh \ Merge common borders in order to create only one mesh.
- Select both meshes and go to Mesh \ Stitch Meshes. Keep the default values and click **Preview**. The command will automatically stitch meshes according to the overlapping area (note that this command is not limited to 2 meshes).

Compound meshes

You can create only one object from different meshes, even if they do not have common borders. Select all the meshes you want to group and then go to Mesh \ Group in compound mesh. The associated command Mesh \ Explode compound mesh will allow you to retrieve all the independent parts.

Note

This command is useful, for example, if you want to compute an inspection and your reference contains several independent parts, because to compute the inspection you can select only two objects: the one corresponding to the reference and the one corresponding to the measure.

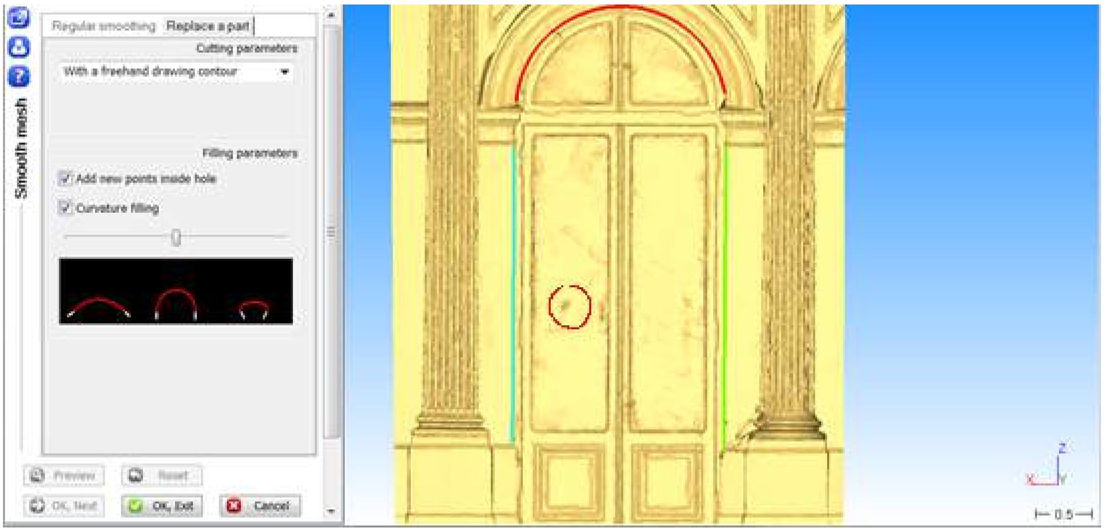
7.2.4 Exercise: Improve global aspect and edges

Usually, you want to obtain a smooth mesh in flat or curved parts, but with the sharpest edges possible. In this exercise, you will see how to remove noisy parts, how to smooth the mesh and how to add sharp edges.

Open the file Smoothing&Edges.rsh

Local smoothing a smoothed fictive line

Select the mesh **Facade** and go to Mesh \ Smooth \ Replace a part. Select the option **With a freehand drawing contour** for example, and enable both options **Add new points inside hole** and **Curvature filling** . Then, draw a contour around an aberrant zone like in the next picture. When you release the button, the surrounded area will be automatically replaced by a new smoothed one. You can repeat this on several areas in order to remove all small defects.



Replace a part of a mesh

Then, click **OK, Next** to validate this step and click on the tab **Regular Smoothing**. Click on the button with the pencil in order to enable the local smoothing. The cursor is now a circle in order to represent the pencil size (there is an option to adjust it). Move the pencil on the mesh while pressing the mouse left button in order to apply the smoothing. Only triangles inside the pencil will change.

Click **OK, Exit** to validate all the corrections.

Global smoothing

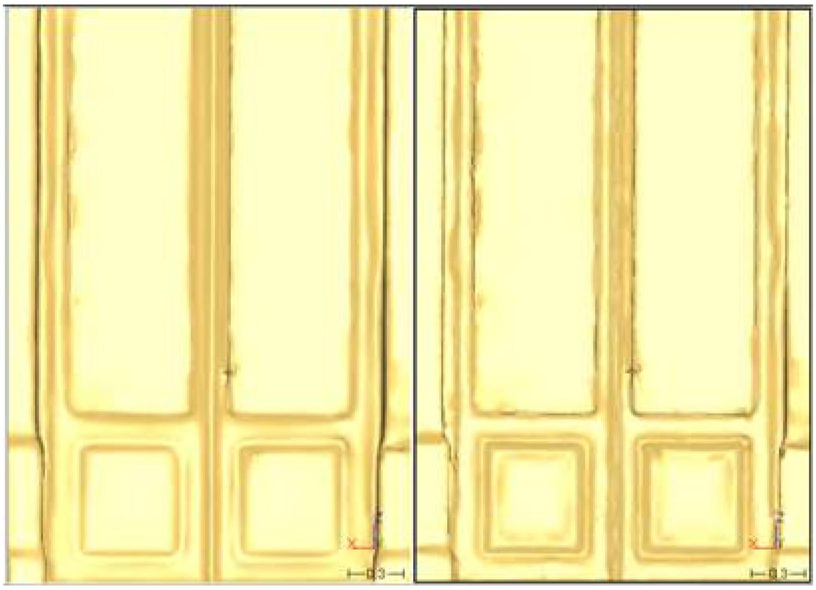
Once all aberrant zones are removed, we can apply a global smoothing in order to make the mesh less faceted, spiky and noisy. Select the mesh and go to Mesh \ Smooth \ Regular Smoothing. There are two methods for smoothing:

- **Smooth noise**: choose this option if you need a very smooth mesh.
- **Keep Details**: keep this option to preserve details.

In both cases you can control the smoothing deformation, and preserve the accuracy using **Control Deviation** option.

Choose **Smooth noise**, do not select **Control Deviation** and set the **Smoothing intensity** to 10. Then click **Preview**. You can see that the global shape looks smoother but all sharp edges have been rounded.

Now select the option **Control Deviation (Max.)** and set the **Deviation** to 5 millimeters (0.005 as meter is the unit of the file) and the **Smoothing intensity** to **20**. Then click **Preview**. With this option, we control the deformation; it means that the maximum distance between the two meshes (before / after) will be 5 millimeters. The global shape has been improved and compared to the previous smoothing. Sharp edges are less rounded.



The smoothing without deviation control makes a smoother surface (on the left), while the smoothing with deviation control preserves the accuracy (on the right)

Re-create sharp edges

In the software, you can recreate sharp edges with a dedicated command. You must first create polylines corresponding to the sharp edges you want to add, and then the software modifies the mesh automatically. There are several tools to create these polylines (like Fictive Line Extraction, Planes Intersections, etc.); they are all described in the section Sections and Polylines.

In the contour group, you will find three polylines:

- **Edge Vault** obtained with the command Polyline \ Feature line,
- **Edge Corner 1** obtained with the command Construct \ Plane \ Extract Plane and Intersection,
- **Edge Corner 2** obtained with the command Construct \ Plane \ Extract Plane and Intersection.

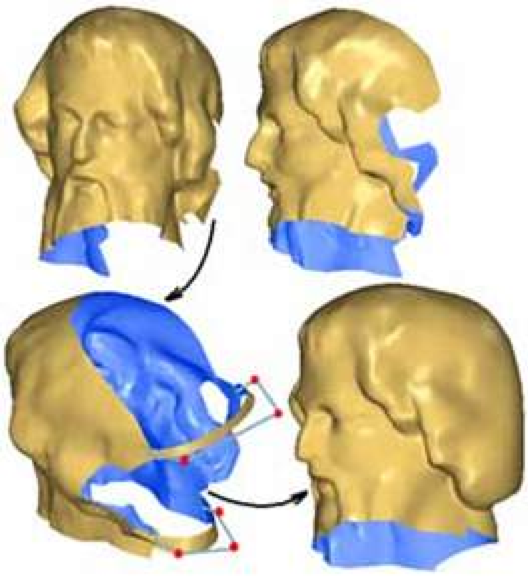
Select the mesh and the three polylines and go to Mesh \ Sharp Edges. Click on the button **Click a point on the limit of the zone to modify** and then click a point on the mesh in order to define this zone. To do this, just click a point on the mesh near one of the selected polyline, then all the triangles between this point and the polyline will be changed. If the point is too close or too far from the polyline, the result will not be correct.

Once the point is clicked, press the **Preview** button. The 3 sharp edges will appear on the mesh. Click **OK** to validate. Feel free to click different points in order to see the difference.

7.2.5 Exercise: Fill holes with curvature filling

Open the file FillHoles.rsh

It represents the face of a statue, but the back of the head could not be measured, as well as a part of the nose. This file contains one mesh for each step of this exercise and the goal is to recreate the complete head. For the back of the head we can not directly use the command Mesh \ Fill holes because the hole is too large and the neck part should not be filled at all.



In the case of the file FillHoles.rsh, it is necessary to preserve continuity with the surface of the border to recreate the back of the head

The first step is to start from the object **1-StartingMesh** and to create bridges like on the picture above. The result you should obtain is something like **2-ReadyToFill**; you can click on this object in the object explorer to have an idea of the expected result.

Create a bridge between free borders

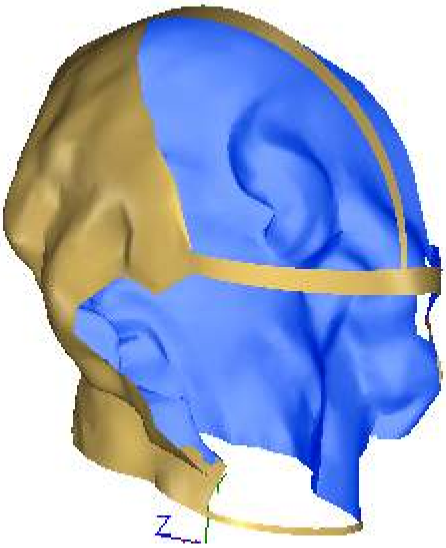
Launch the command Mesh \ Bridge. We will create two bridges: one that is used to delimit the back of the neck, and a second one that will be used to guide the subsequent hole filling process by its round smooth shape imitating the back of the head.

Do the following steps:

- click the two free edges as shown on the figure above. The default bridge you obtain does not have the appropriate shape (too sharp and the medium orientation is not vertical),
- drag the red balls to change the length of the first and last segment as well as the middle ball to obtain something more rounded,
- you can also move the **Tension** slider to automatically adjust the control points of the bridge,
- move the slider to adjust the **Middle twist orientation,**
- when you are satisfied with this first bridge, select **Sew all the parts** and click **OK, Next** to start your second bridge,
- click the two free edges as shown on the above figure. The default bridge you obtain looks too much like a circle,
- press **CTRL** to select the 3 red balls together,
- move the 3 balls together to get something like **2-ReadyToFill**, and
- click **OK, Exit**.

Note

You can also create a junction attached on another junction like you can see on the next picture.



Example of bridges

Fill Holes

Now we can fill all the holes with the command Mesh \ Fill holes. Select the object you have obtained previously or simply select the object **2-ReadyToFill** and launch the command.

- select the option **By click** and click all the holes except the neck hole,
- activate options **Filling Holes** and **Curvature filling**. Adjust curvature filling slider to increase or decrease the volume of the head,
- press the **Preview** button. You should obtain something like the object **3-AfterFill,**
- lick the button **OK, Next,**
- click the neck hole in order to select it,
- deselect the option **Curvature filling,**
- press the **Preview** button. You should obtain something like the object **4-ClosedMesh,**
- click **OK, Exit** to validate the command,

Now, if you want something perfect, you can use the command Mesh \ Smooth and you will obtain a shape like **4-ClosedMesh**.

7.2.6 Exercise: Reconstruct perfect holes on a mechanical part

In the software, there is a dedicated command to automatically fit mesh borders with polylines in order to have perfect borders/holes.

Open the file MeshImproveBorders.rsh.

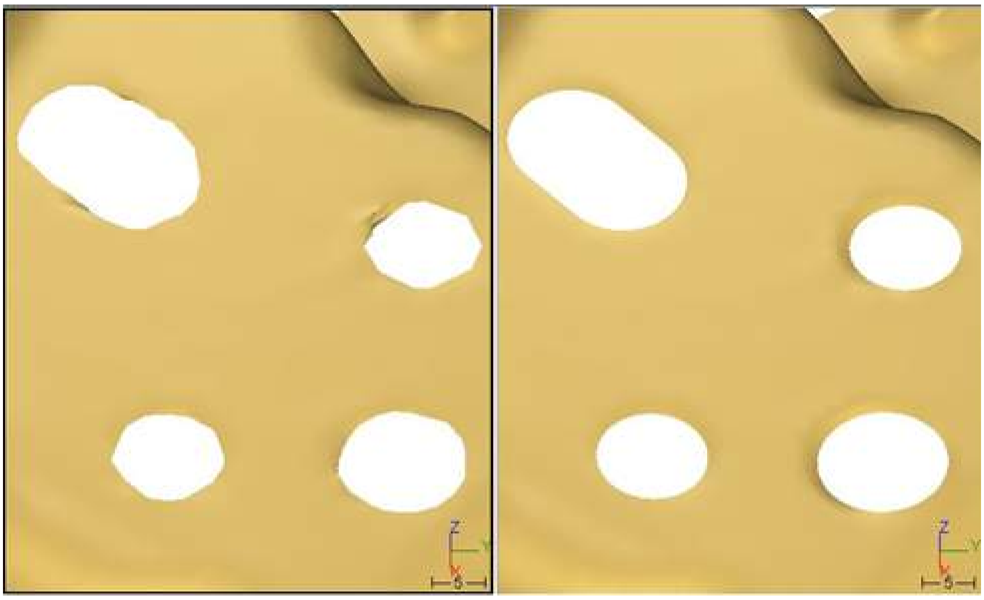
In this file, there are a mesh and 5 polylines corresponding to 5 holes. They have been obtained thanks to features extraction (see example in Exercise: Compute best shapes from clouds and polylines) and polylines improvement (see section Improve polylines).

Select the mesh and the 5 polylines and go to Mesh \ Set Borders. There is only one parameter to set: the **Reconstruction Distance**. It is the width of the mesh that will be modified in order to fit to the new border. To define this parameter you can:

- enter directly a value
- click the button **Click two points** and then enter two points in order to define the distance

Enter, for example, a value greater than 10, like 11. A warning message is displayed saying that 2 borders are too close. Click anyway on **Preview**. The result is bad because two holes have not been reconstructed (highlighted in red) due to a too high value.

Enter now 1.5 then click on **Preview** again. The result is now perfect like in the following picture. You can click **OK** to validate.



Borders have been improved in only one click

7.2.7 Exercise: Apply the color of a point cloud on a mesh

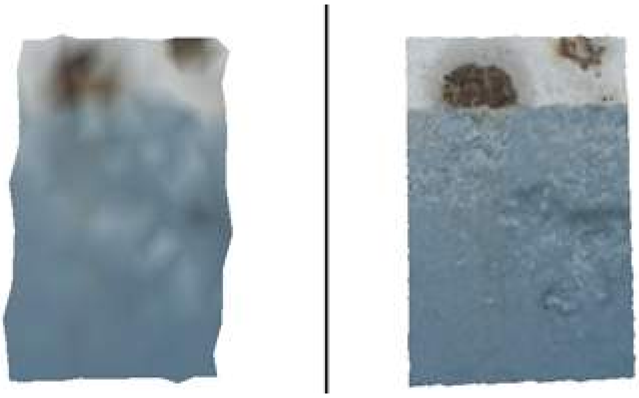
When a cloud contains color or inspection information, it is possible to apply these colors on the mesh. However, it should not be confused with textures as here we apply just a color on each vertex, not inside a triangle.

Open the file MeshColor.rsh.

This file contains a colored point cloud and two meshes with a different triangle size.

- show only the objects **Cloud with Color** and **Mesh with small triangles**, select these two objects and go to Mesh \ Take color from cloud,
- select the options **Previous color, if any, is overwritten ONLY if one point in the cloud can provide the color** and **Only if the distance to vertex is less than 1% of the mesh size,**
- click **Preview**. The color has been correctly applied on the mesh, and
- click **OK** to validate.

Now, redo this process with the mesh with big triangles. The result will not be correct; there is a big blurring effect due to the triangle size as, with this command, we color only vertices.



When we apply colors from the point cloud on a mesh, the triangle size is very important.

8 Sections and Polylines

See the section Construct \ Polyline to know how to create a polyline manually.

- Create sections
  - Exercise: Create a planar section on a point cloud
  - Exercise: Create planar sections on a mesh (extract contour lines)
  - Exercise: Radial sections on a mesh
  - Exercise: Guided sections on a mesh
- Manage polylines
  - Exercise: Cut polylines
  - Exercise: Chain polylines
- Improve polylines
  - Exercise: Improve polylines to create the 2D plan of a building
- Feature/Border/Fictive lines
  - Exercise: Extract a feature line from a mesh
  - Exercise: Rebuild a sharp edge using a fictive line
- Polyline extraction
  - Exercise: Extract planar contours from a point cloud
  - Exercise: Extract the neutral axis from a tubular shape (mesh or cloud)

In the Polyline menu, you can use some other tools that are not detailed in this Guide, but you can refer to the Help files:

- Polyline \ Join 2 polylines: to compute a curve to link the extremities of two polylines
- Polyline \ Contour / Hole: to extract the lines of the free contours and the holes in a mesh
- Polyline \ External contour: to compute the external contour of a mesh seen from a specific direction

Several other tools are available in the software to extract and manage polylines. They are situated in the menu Create:

- Construct \ Intersection: to compute various intersections, between polylines, meshes and geometric shapes
- Construct \ Projection: to compute various projections, such as a point or a polyline on a plane or on a mesh

8.1 Create sections

With the software, it is possible to create different kind of sections on a mesh or on a point cloud:

- Freehand sections on a mesh
- Sections along a curve on a mesh
- Radial sections on a mesh
- Planar sections on a mesh or on a point cloud

Additional parameters are necessary to create a section on a point cloud; they will be explained in the first exercise.

- Exercise: Create a planar section on a point cloud
- Exercise: Create planar sections on a mesh (extract contour lines)
- Exercise: Radial sections on a mesh
- Exercise: Guided sections on a mesh

8.1.1 Exercise: Create a planar section on a point cloud

Planar sections can also be computed directly on point clouds.

> Open the file “SectionsBuildingPlan.rsh”

It contains a point cloud of the ground floor of a building. The inner points have been removed in order to lighten the file, only points around the walls have been kept. We will use this example again in the following paragraphs to show how to create the 2D plan of the building.

Select the point cloud and go to Polyline\Planar Sections. Choose Z for the plane direction. Uncheck the option **All over** and enter **1** for the number of sections, as we only want to create one section.

To define the section plane, place the mouse in the scene and press **X** on your keyboard to set the view along X axis. You may need to press also **A** to make a zoom all. Click on

in the dialog box and zoom in the scene to click a point on the top of the point cloud.

To create sections on a point cloud, additional parameters have to be entered:

- The plane thickness gives the thickness of the point cloud to take into account to create the sections.
- The chaining distance: if the distance between two points is lower than the given chaining distance, a new segment is created.

These parameters have to be entered regarding the density of the point cloud, and the average distance between the points. Check **Activate the noise reduction** in order to have smoother polylines.

Parameters to create a section on a point cloud

You can try different values for the plane thickness and the chaining distance to compare the different results.

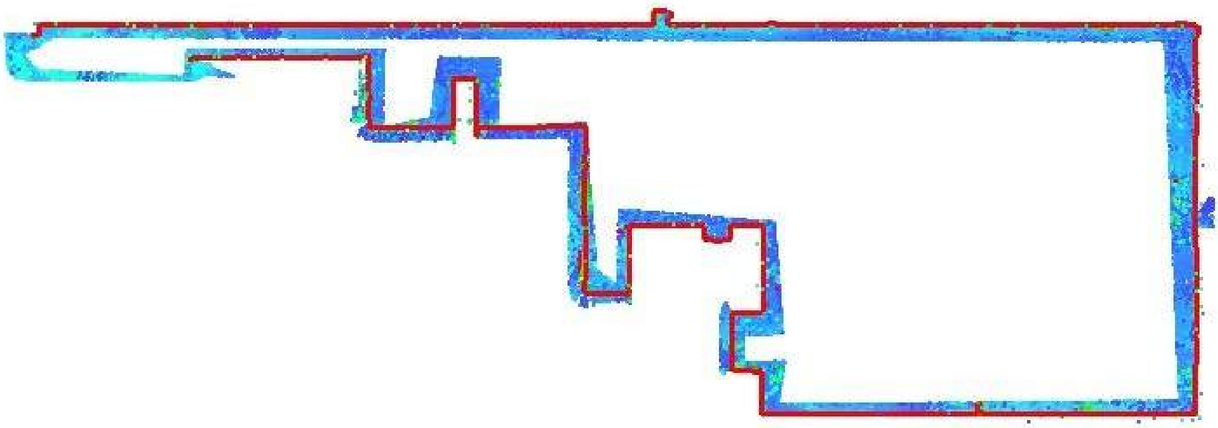
Section on a point cloud - Compare values for plane thickness and chaining distance

If you enter a plane thickness too low, there will not be enough points, so you will have very short polylines as in above picture (**part a**) and you can lose some details. If you enter a plane thickness too high, the points on the floor will be taken into account, so you will have very noisy polylines as **part b**.

If you enter a chaining distance too low, you will have many short independent polylines as in **part c**. If you enter a chaining distance too high, all the points will be chained together as in **part d**. You will obtain long polylines and you may need to cut them for further processing.

For this exercise, set the plane thickness to **0.35**, the chaining distance to **0.5** and **activate the noise reduction**. Click **Preview** and **OK, Exit** to validate the results. A new folder containing all the polylines is added in the Contour Group.

Now, we are going to see how to manage and improve the polylines in order to create the 2D drawing of the building. Look at the picture below to see an example of result. The reduced point cloud is blue and the 2D polylines are red.



2D plan of a building

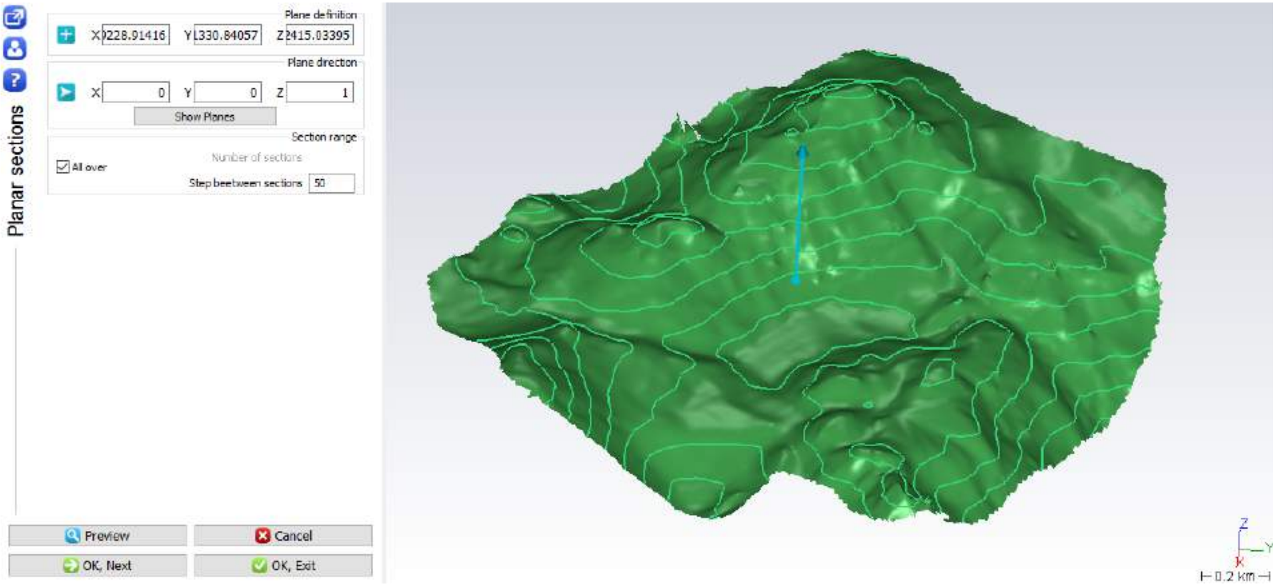
8.1.2 Exercise: Create planar sections on a mesh (extract contour lines)

Open the file “SectionsContourLines.rsh”.

It contains the mesh of a mountain, and we want to create contour lines on it. Select the mesh and go to Polyline \ Planar Sections. This command allows computing one or more sections all in parallel planes defined by a given direction.

Here we are going to compute sections on the mesh with a regular step along the direction of the Z axis, so enter Z for the plane direction. The point used to define the plane is not needed as we will compute lines all over the mesh. Check the option **All over** and enter **50** for **Step between sections** in order to have a section each 50 meters on the whole mesh.

When you press **Preview** a pop-up appears to inform you that 36 polylines have been extracted. Click **OK, Exit**. All the polylines are inserted in a new folder called **Planar sections Z MountainLake** in the Contour Group. Each polyline is named after its value in Z.



Create contour lines

8.1.3 Exercise: Radial sections on a mesh

Open the file “SectionsDynamic.rsh”.

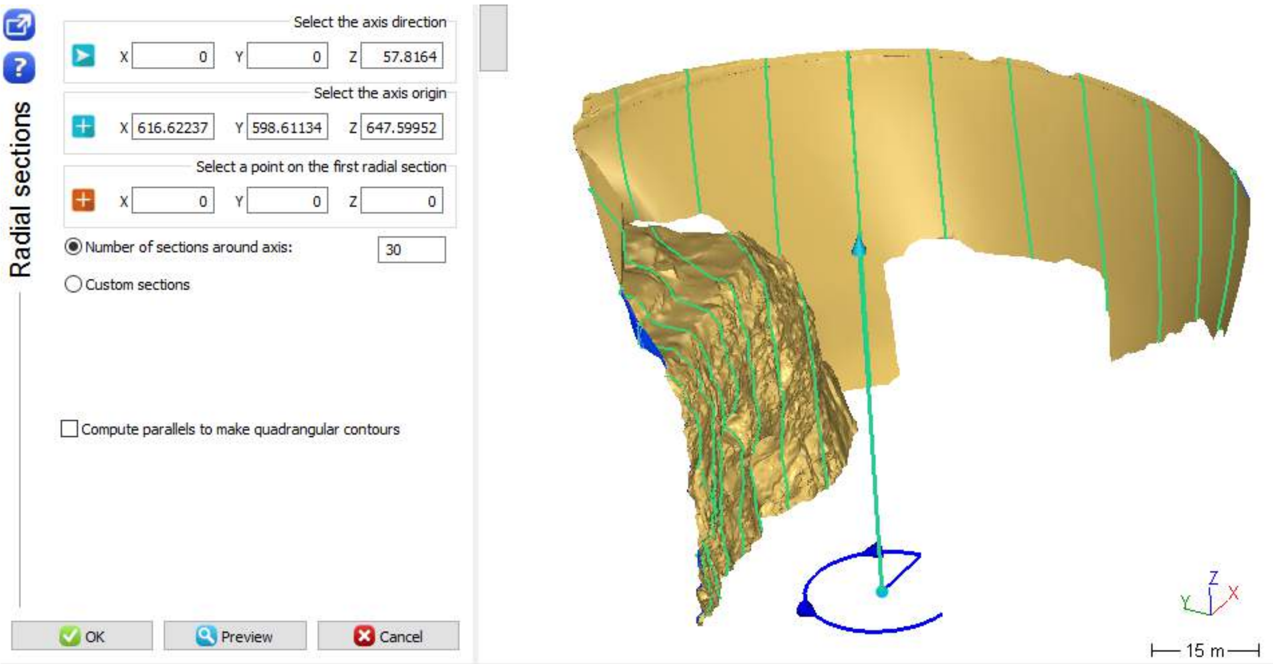
In the scene, show only the mesh **DamRock** and the polyline **Center axis** located in the Contour Group. We are going to create regular sections on the mesh with a different tool which allows computing planar sections around an axis.

Select the mesh and go to Polyline \ Radial Sections. Use **Center axis** to define the axis (

to define its direction,

to define its position). The axis is displayed in the 3D scene (in cyan in the image below), with arrows showing the direction of the rotation. If you set the view to X (press **X** on the keyboard), you can see that the axis is vertical.

Enter the number of sections you want to create around the axis and preview the results. If you enter **30**, each section is created with a rotation angle of 12 degrees from the previous one around the defined axis (360/30 = 12).



Create radial sections on a mesh

8.1.4 Exercise: Guided sections on a mesh

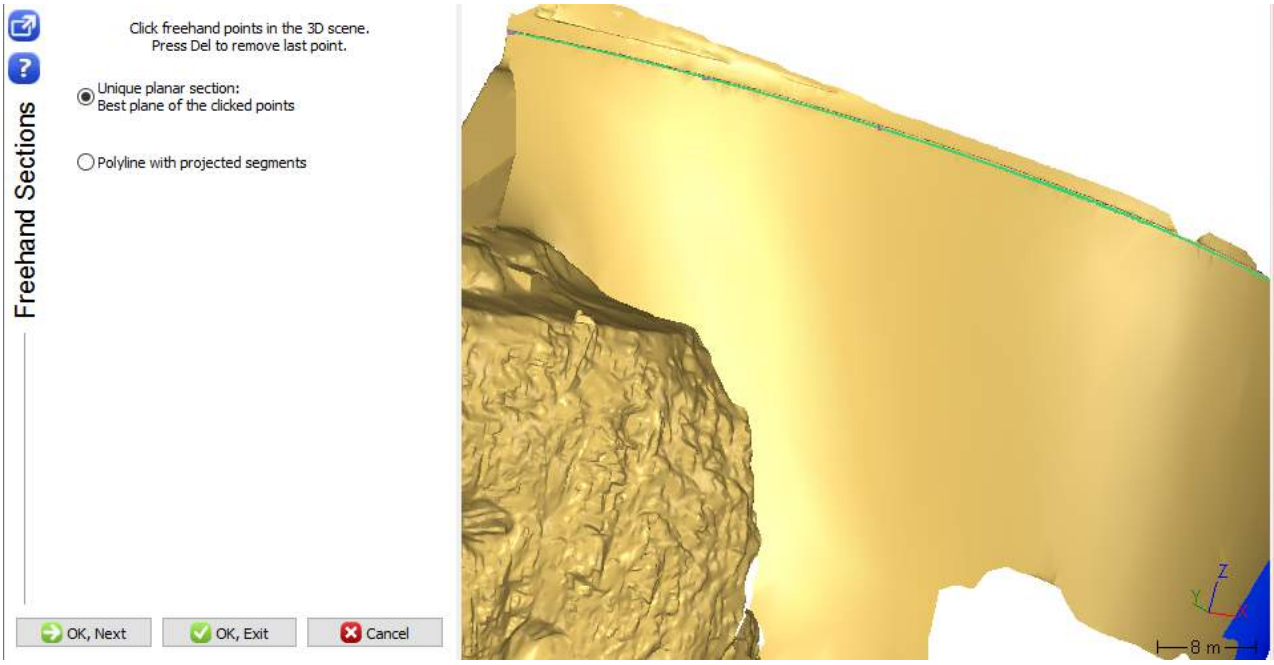
Open the file “SectionsDynamic.rsh”

It contains the mesh of the dam used in the exercise concerning point cloud processing.

Freehand sections

Select the mesh and go to Polyline \ Freehand Sections. Choose the option **Unique planar section: Best plane of the clicked points** so that the section will be drawn in the best plane of all the points clicked. Now click a few points on the dam below the upper edge as shown in the following picture. Make sure to choose **Point on selection** in the upper ribbon in order to click points on the mesh.

With the option **Polyline with projected segments**, each time you click a point on the surface, the segment between this point and the previous one is projected on the mesh according to the view direction. In this way you can force the section to go through specific points that are not all necessarily on the same plane.

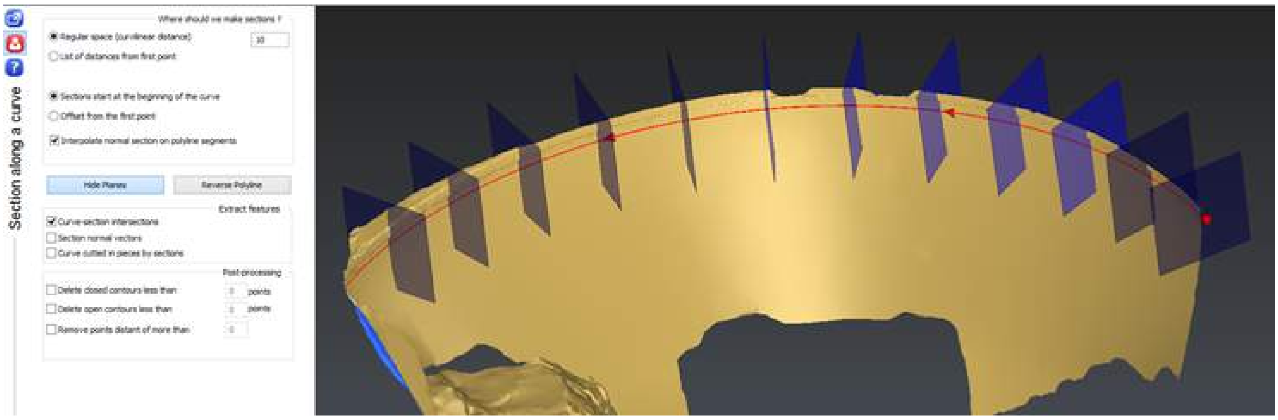


Create a planar freehand section on a mesh

Now validate the result. A new polyline is created (or a set of polylines. In this case, explode the set and continue the exercise with the longest polyline).

Sections along a curve

Select the polyline created just before (or the polyline **FreeHand section**) and the mesh and go to Polyline \ Sections along Curve. You can see an arrow on the polyline indicating its direction.



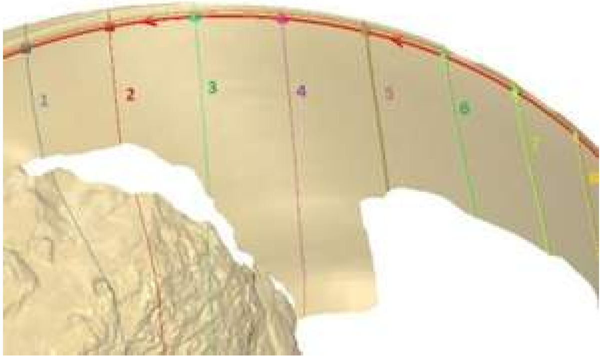
Create a planar freehand section on a mesh

You can draw sections with a regular step or by giving a list of distances from the first point of the polyline. It is possible to set another point as the origin of the sections by entering an **Offset from the first point**. All the distances to enter are curvilinear distances along the polyline.

Check the option **Regular space** and enter a distance of **10**. You can click on **Show planes** to display temporarily the planes where sections will be created. The planes are locally perpendicular to the polyline.

With the advanced parameters, it is possible to create points on the polyline where sections are created. See the section Polyline \ Sections along Curve for more detailed explanations.

Click **Preview** then **OK** to validate the results. The sections created are named according to the mesh on which they were computed and their number. One section can be either a polyline or a set of polylines depending on the holes in the mesh. In the picture below, sections 1 to 4 are sets of polylines and sections 5 to 8 are polylines.



Polylines and sets of polylines

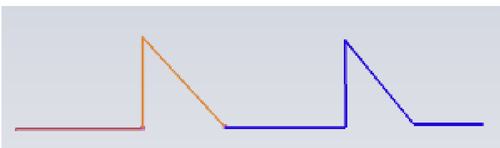
> **Note**
>
> You can explode the sets of polylines by selecting them and go to Polyline\Explode Polylines.

8.2 Manage polylines

- Exercise: Cut polylines
- Exercise: Chain polylines

8.2.1 Exercise: Cut polylines

Any polyline can be separated into two at any point. The polylines do not have to be selected before opening Polyline \ Cut Polylines. Select the appropriate option in the upper ribbon to cut the polylines at specific points. This can be a way to manually clean a polyline.



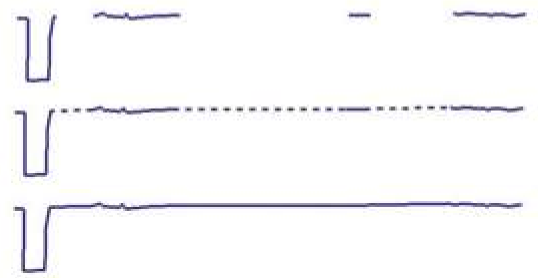
Polyline cut in three parts

8.2.2 Exercise: Chain polylines

Polylines can be automatically chained together by computing new segments between unconnected points.

Open the file “SectionsBuildingPlan.rsh”.

You can chain the polylines of the group **Lines to chain**. Select the polylines and go to Polyline\Chain /Group Polylines. Select the options **Chain and create new segments** and **As many Polylines as necessary**. The new segments appear as dotted lines.



Automatic chaining of polylines

Note

- Polylines can also be grouped in a set of polylines. After creating the planar sections on this file (Exercise: Create planar sections on a mesh (extract contour lines)), we could group the polylines and create one set of polylines per elevation.
- A set of polylines can always be exploded into individual polylines by using Polyline\Explode Polylines.

8.3 Improve polylines

- Exercise: Improve polylines to create the 2D plan of a building

8.3.1 Exercise: Improve polylines to create the 2D plan of a building

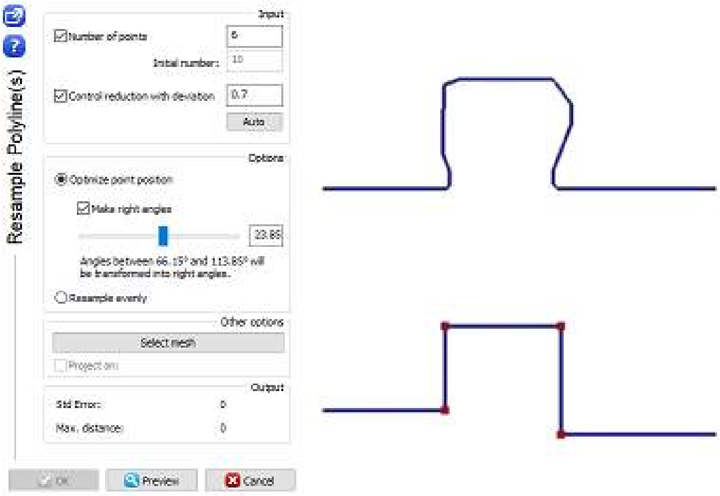
Open the file “SectionsBuildingPlan.rsh”.

Use the planar section previously created on the point cloud. First, you can try the tools seen previously (Cutting and Chaining) on some polylines. Then' we are going to see further processing to improve the polylines. You can look at the polyline **Final Building Contour** to see an example of a result that you could obtain.

Simplify / resample a polyline

A polyline can be simplified in order to reduce its noise or to recreate right angles.

Select a small noisy polyline (for example LineToChain1) and go to Polyline \ Resample Polyline(s) . Set the number of points to have on the resampled polyline. If you want a straight line, enter **2**. Choose the option for the polyline positioning. The option **Optimize point position** will compute a polyline going through the noise in order to reduce the standard deviation error. The option **Resample evenly** will keep the positions of the two end points of the polyline and create new points evenly spaced. This tool can also be used to recreate the right angles on a polyline. You can try it on the polyline called **RightAngles**. See the result in the picture below.



Resample a polyline and make right angles

The standard deviation and the maximum distance between a point and the new polyline are displayed at the bottom of the dialog box. The box **Control reduction with deviation** allows you to constrain the polyline reduction to be lower than a particular deviation. This threshold can be automatically set up by clicking on the button **Auto**.

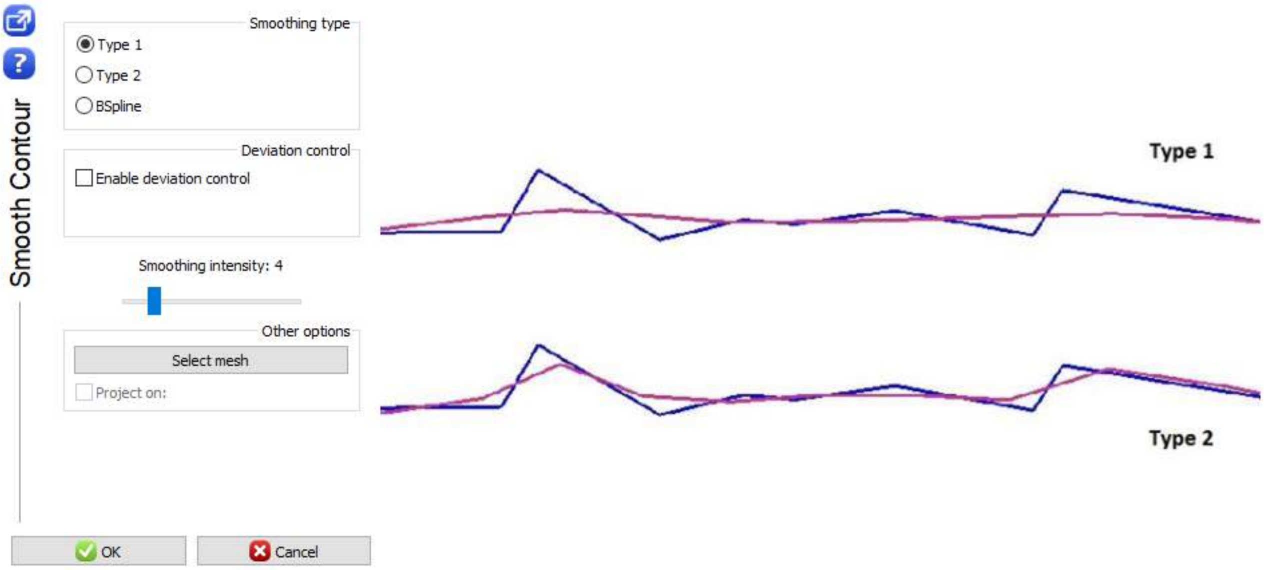
Smoothing

To reduce the noise in a polyline, you can also select the polyline and go to Polyline \ Smooth Polyline(s). Three types of smoothing are available. The smoothing intensity represents the number of iterations of the process.

See the picture below to compare the types 1 and 2 used on the same polyline with the same intensity. The original polyline is in blue and the smoothed polyline in pink. Type 1 creates a smoother polyline whereas with Type 2 tries to keep the general shape of the polyline. With Type Bspline, points are resampled, so that they are regularly spaced on the polyline.

As another example, if a polyline representing a rough circle is smoothed with Type 1; it will tend to a smaller circle. With Type 2, the radius of the circle will be approximately preserved.

With Type 1 and Type 2, it is also possible to control the deviation error by entering the maximum deviation authorized between the smoothed line and the original.

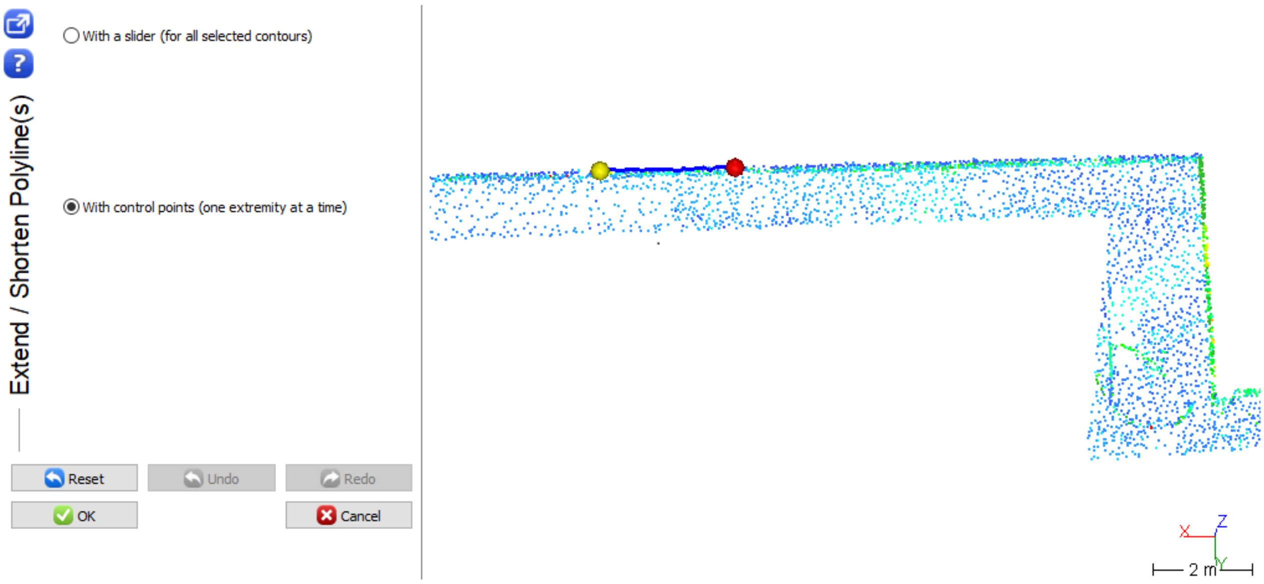


Smooth a polyline

Extend / Shorten Polylines

You can first display the point cloud to see how to extend the polylines.

The length of a polyline can be modified very easily by extending or shortening its last segments. Select a polyline and go to Polyline \ Extend / Shorten Polyline(s). Move the control points at each end of the polyline, or use the slider.



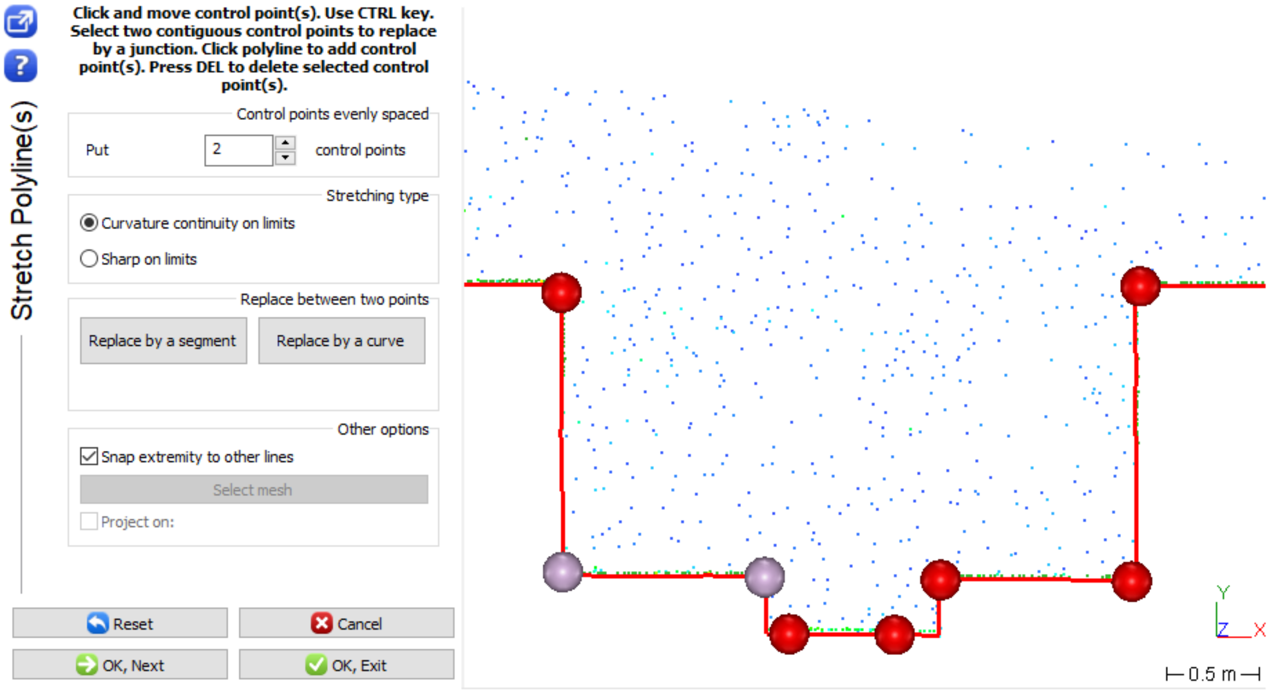
Extend or shorten a polyline

Strecthing

Select a polyline and go to Polyline \ Stretch Polyline(s)

Two control points are automatically displayed at the extremities of the polyline.

You can add intermediate control points by increasing the number of **Control points evenly spaced** or by clicking specific points on the polyline in the 3D scene.



Stretch a polyline with several control points

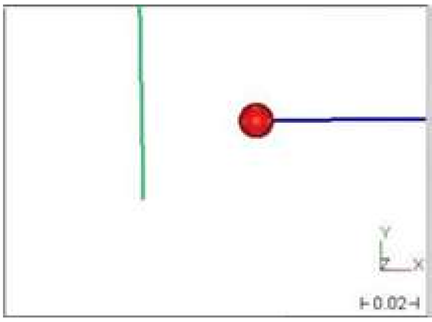
The polyline automatically changes when you move a control point. You can change the **Stretching type** to have either curvature continuity or sharp angles around the control point.

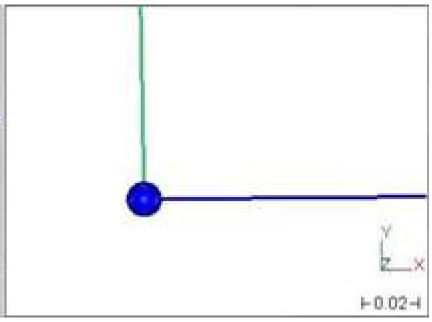
If you select two consecutive control points using the **CTRL** key, you have the possibility to replace the part between them by a segment or by a curve. When you edit the curve, you just need to select one of the big balls to validate the curve.

At any time, you can:

- Add a control point by clicking on the polyline where you want the new point.
- Press **DELETE** to delete the selected control points that are not on the extremity.

Activate the option **Snap extremity to other lines** to be able to link two polylines by moving a control point from one polyline on the other. When the control point is blue, it means that the polylines are connected.



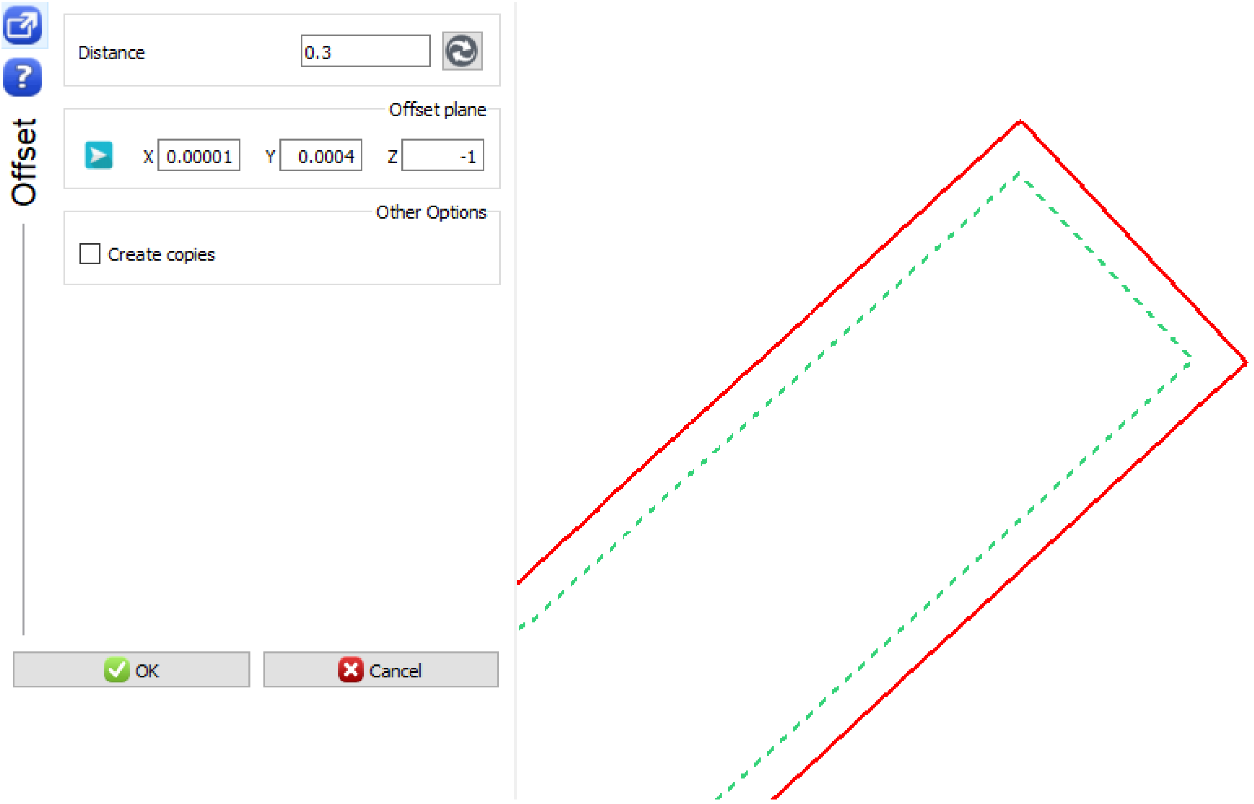


Connect two polylines

It is also possible to stretch a polyline projected on a mesh by selecting both the mesh and the polyline before launching the command. If you check the option **Project on: the selected polyhedron**, the polyline will automatically be projected on the mesh. This can be very useful for reverse engineering.

Offset

To go further with this exercise, you can select your final contour or show the object **Final Building Contour** and go to Transform \ Offset to create the contour of the external walls. Choose the direction for the offset and set the distance. Here we can compute an offset of **0.3** meters in the best plane of the contour. The side of the offset can be reversed if necessary. You can look at the polyline **Final Offset contour** to see the result.



Offset a polyline

8.4 Feature/Border/Fictive lines

The software provides tools to compute polylines following sharp edges, fillets or small radii on a mesh.

It is good to know that the curvature of a mesh can be negative or positive:

- A negative curvature has its center inside the mesh; the shape is convex.
- A positive curvature has its center outside the mesh; the shape is concave.

Three types of lines can be extracted from a mesh:

- The **feature line** is the line on the sharp edge.
- The two **border lines** are the lines on both sides of the feature line.
- The **fictive line** is the line which can be used to recreate the sharp edge.

> The extracted lines could also be useful for constraint meshing. See chapter Meshing and mesh improvement.

- Exercise: Extract a feature line from a mesh
- Exercise: Rebuild a sharp edge using a fictive line

8.4.1 Exercise: Extract a feature line from a mesh

Open the file “FeatureLines.rsh”. It contains the mesh of a pillar and four points which will help you for the first step.

This guide will introduce you to the Polyline \ Feature Line extraction command which is divided in four tabs, each of them taking care of one step in the recreation of sharp edges:

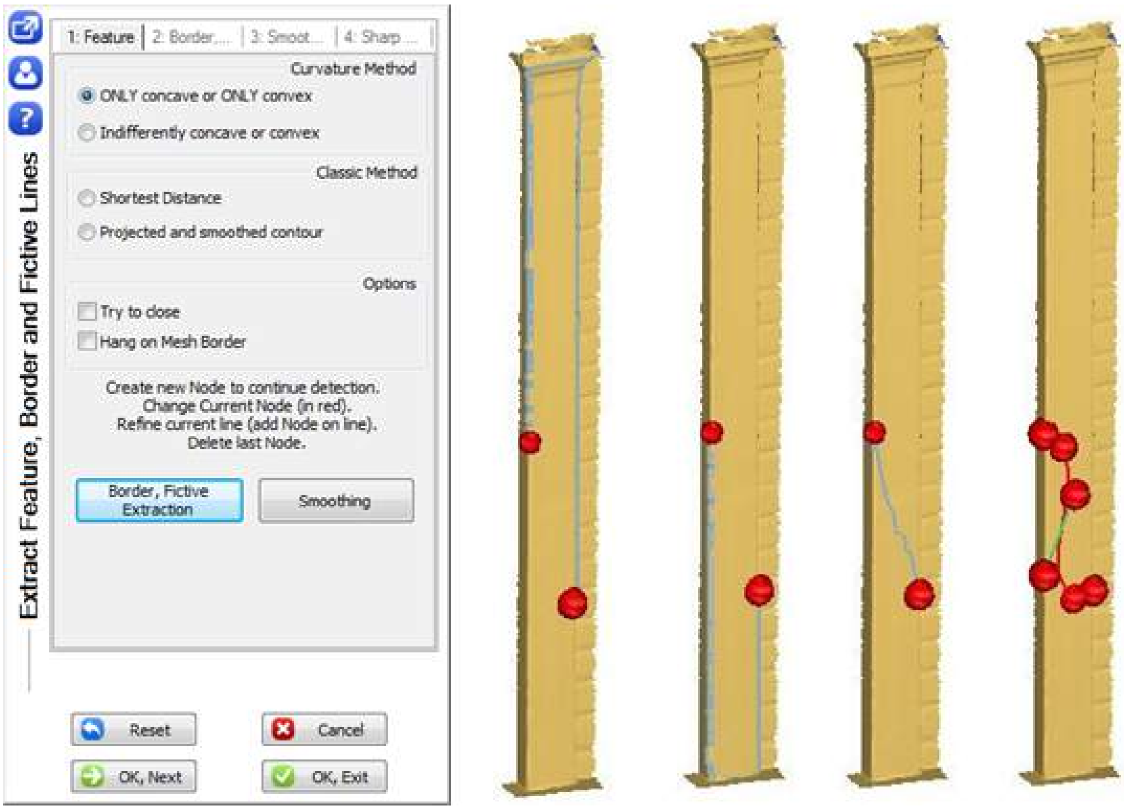
- Feature line extraction
- Border and Fictive lines extraction
- Smoothing lines
- Sharp edge reconstruction

Creating a Feature line

The command Polyline \ Feature Line creates a polyline which follows the characteristics of the mesh you selected before entering the command. From points you chose, a feature line is constructed by connecting these with the shortest path between them subject to one of the following constraints:

- **ONLY concave or ONLY convex**: The line will create a path only concave or only convex. The path cannot be partially concave and partially convex.
- **Indifferently concave or convex**: With this method the path is authorized to be partially concave and partially convex.
- **Shortest Distance**: The line will link the two points in the shortest way possible by following the vertices of the mesh.
- **Projected and smoothed contour**: this method projects a smooth contour onto the mesh

The impact of the method on the created line is visualized in the following figure:



Four methods to extract a feature line

The **Projected and smoothed contour** option is not compatible with the other options: selecting or deselecting this option will reset your chosen control points.

To extract a feature line, we enter the command with a selected mesh (**Mesh**) and click the starting point of our feature line. The method used to constrain the line can be changed for each segment separately. We change the method, if necessary, and click the second point on the mesh, which immediately starts the detection of the feature line segment. In this way we can gradually draw the complete path by adding segments one after the other. The created segment will start and end at the nearest vertex from the points clicked and go through the vertices of the mesh (excepted if **Projected and smoothed contour** is selected).

To preview the different methods as shown in the picture above, proceed as follows: click the first point as shown on the picture below, choose the first method, and click the second point. To see the next method, press the **DEL** key to delete the last point, change the option and click on the endpoint again.

Clicked points appear as red balls. You can at each moment move these points by clicking them and keeping the button pressed. If **Projected and smoothed contour** is selected, points which are snapped onto another line or mesh contour are shown in green.

Note for the 3 first methods

You can add intermediate points in a segment, or add points after the last one. The last point clicked can always be canceled by pressing the key **DEL**. You can click on the first ball to reverse the direction of the line.

Once the feature line is extracted, you can use it as an input to extract the border and fictive lines in the next tab; you can smooth it, or you can simply extract another feature line with **OK, Next**.

The dialog box has several advanced options, useful to improve the result in some cases, of which you will find a detailed documentation in the dedicated Help section.

8.4.2 Exercise: Rebuild a sharp edge using a fictive line

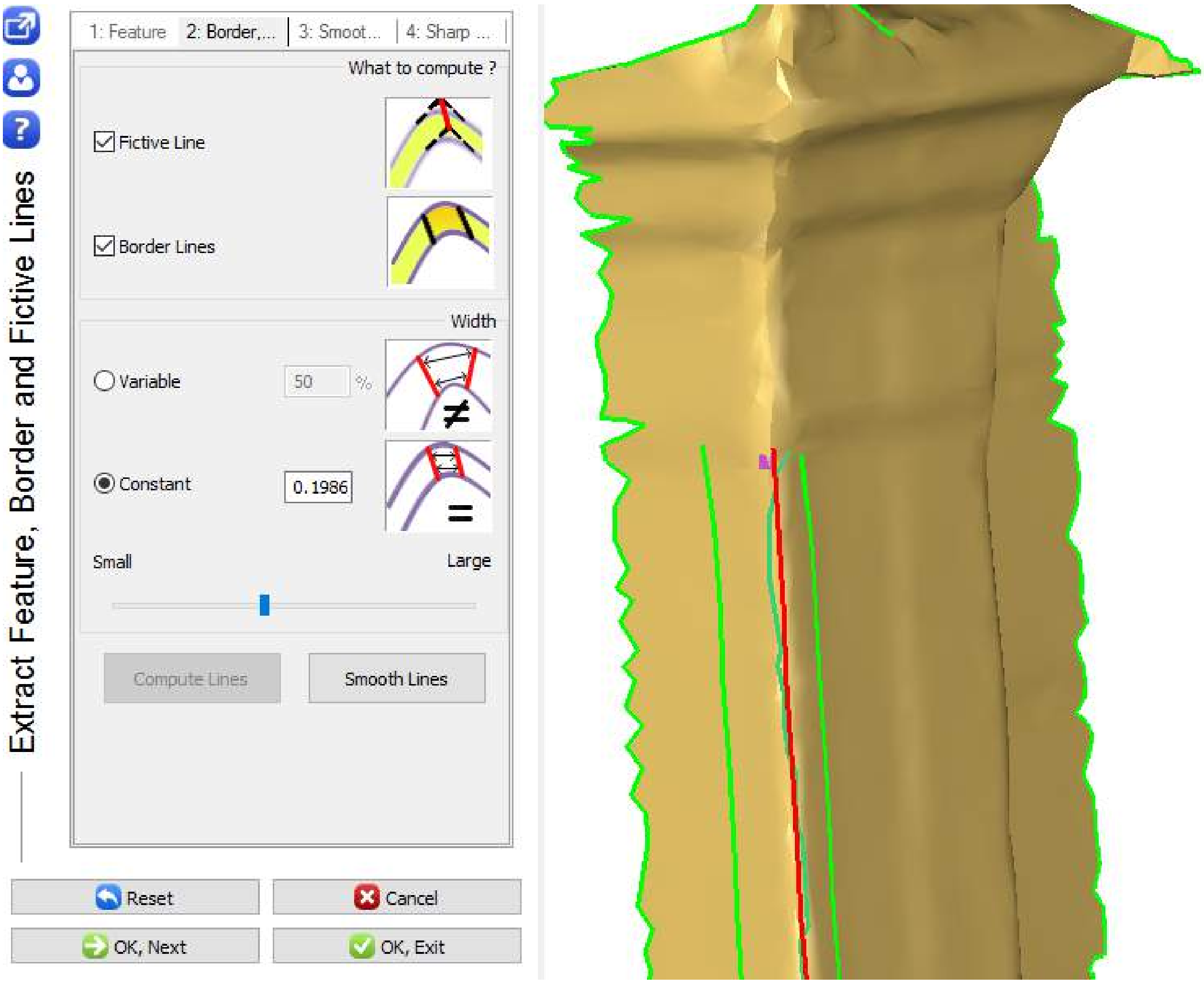
Open the file “FeatureLines.rsh”.

Show the points “Point 1” and “Point 2”. Select the mesh and go to Polyline / Feature Line. Choose the option **ONLY concave or ONLY convex**. Click on the first point near “Point 1” and on the second point near “Point 2”. Check that the feature line lies only on one edge, then click on **Border, Fictive Extraction** to continue with the extraction of the two border lines and the fictive line.

If you select a mesh and a feature line before opening the command, you will directly get to the tab **Border, Fictive**.

Border lines

In this tab, you can choose to extract the fictive line and the border lines. Check both options.



Extract the fictive line and the two border lines

The width is the distance between the two border lines, which can be set as variable or constant. With the variable mode the command tries to find automatically the width. You can use the slider to increase or decrease its value. If you put a high percentage of variability, the distance between the border lines can be very different along the line. If you put a constant width, the border lines will be parallel.

Click on **Compute Lines** to preview the resulting lines. Modify the width and click again on **Compute Lines** to preview the difference.

Choose the constant width and enter a width of **0.20**. Compute line and then click on **Smooth Lines** to smooth the extracted lines.

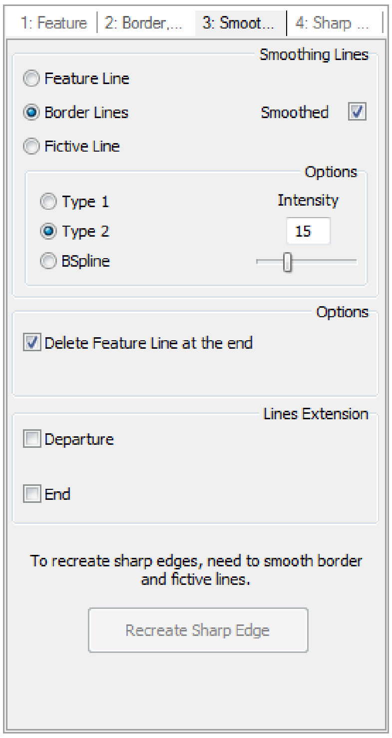
Smooth lines

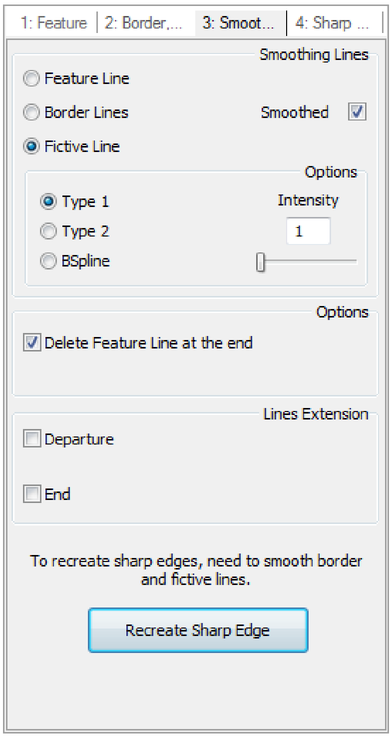
In the smoothing tab, you can choose which lines you would like to smooth.

Check the option **Smoothed** for both Border Lines and Fictive Line (see the following picture for settings). If you wish, you can modify the smoothing parameters by changing the type of smoothing and the intensity (see the paragraph about Smoothing for further information about these parameters).

Various options are available in this tab:

- If you are not interested in the feature line, you can check the option “Delete Feature Line at the end"
- It can occur that the lines computed are not continuous. In this case, an option can be checked to chain the pieces of line together
- It is also possible to extend the lines from one or both ends if you did not click far enough on the edge





Border line Fictive line

We do not smooth a lot the fictive line. If we smooth too intensively, we cut the angles.

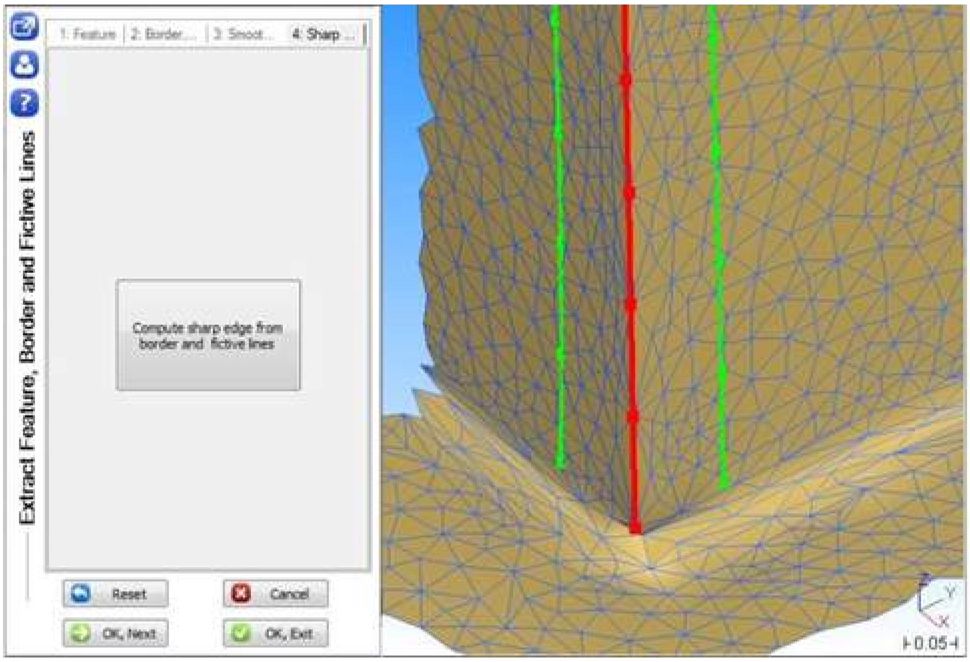
If you have smoothed the border lines and the fictive line, you are able to recreate the sharp edges of the mesh, so click on **Recreate Sharp Edge**.

> Use the next tab only to recreate one single sharp edge. If you have several edges arriving at a same point, you must exit here the command and process the polylines manually with the tools described above in order to recreate the sharp edges with Mesh / Sharp Edges.

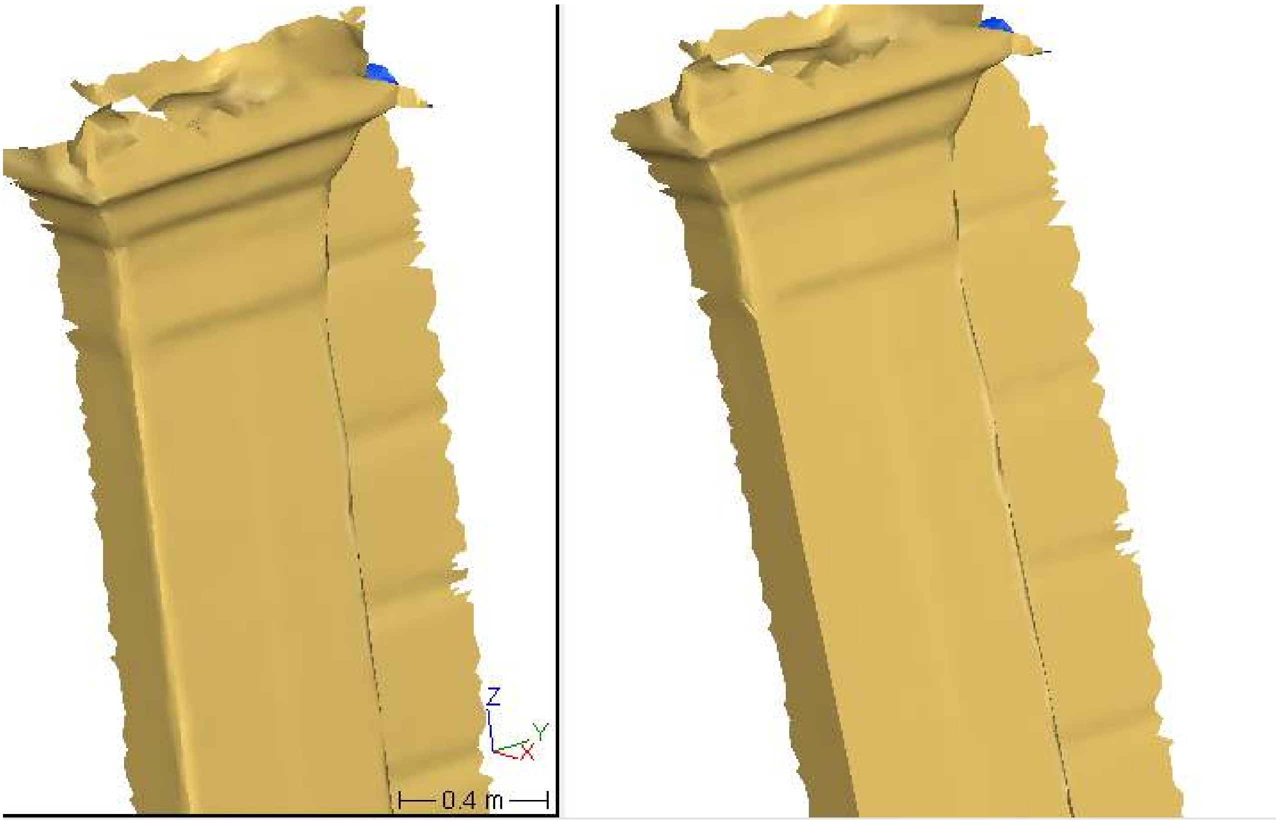
Sharp edge reconstruction

Click on **Compute sharpe edge from border and fictive lines** to preview the results. The mesh is modified in order to respect the given fictive line. The border lines define the area concerned by the re-meshing, around the fictive line.

The edge of the pillar is now perfectly straight.



Rebuild the sharp edge



Compare the original edge with the reconstructed edge

> You can try the Exercise 7 - Meshing a facade point cloud (downloadable online), to see the reconstruction of a vault with the same command Polylines / Feature Line. In this exercise you can also see how to rebuild a sharp edge in a different way. The fictive line is created by intersecting two planes with Create / Intersection, and the edge is rebuilt with Mesh / Sharp Edges.

8.5 Polyline extraction

- Exercise: Extract planar contours from a point cloud
- Exercise: Extract the neutral axis from a tubular shape (mesh or cloud)

8.5.1 Exercise: Extract planar contours from a point cloud

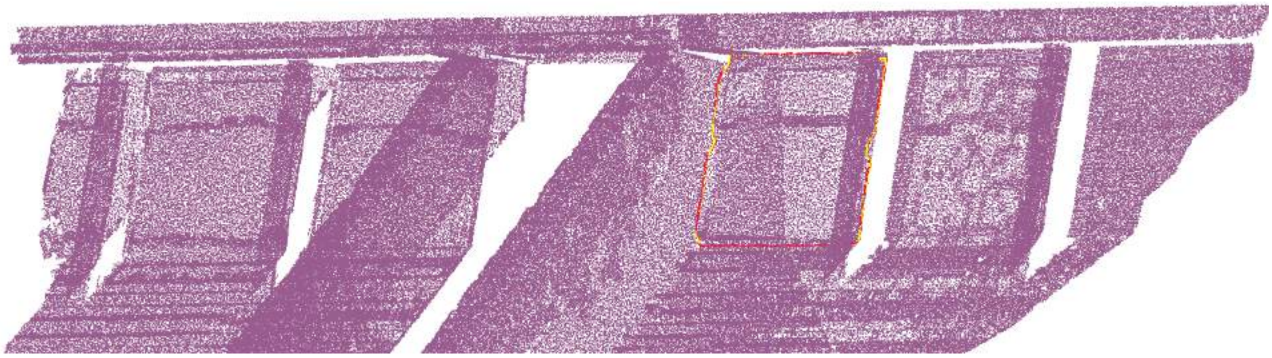
> Open the file "BestFitOnRef.rsh".

Show only the cloud **Aligned Dam**. Launch the command Polyline \ Planar Contours(s).

Click on one point on the cloud as shown on picture below. The software will automatically try to find the plane around the clicked point. Click on a new point while pressing the **CTRL** key. The software computes a new plane from the two points. If the plane is OK, select the option **Extract all contour/hole** and select both **Planar** and **3D**. Then click **OK, Exit** to validate.

Hide the point cloud; you will see all the extracted contours:

- Red contours are 3D contours; they go through the real points of the cloud.
- Yellow contours are 3D contours projected on the extracted plane.



Planar contours extracted from a point cloud

8.5.2 Exercise: Extract the neutral axis from a tubular shape (mesh or cloud)

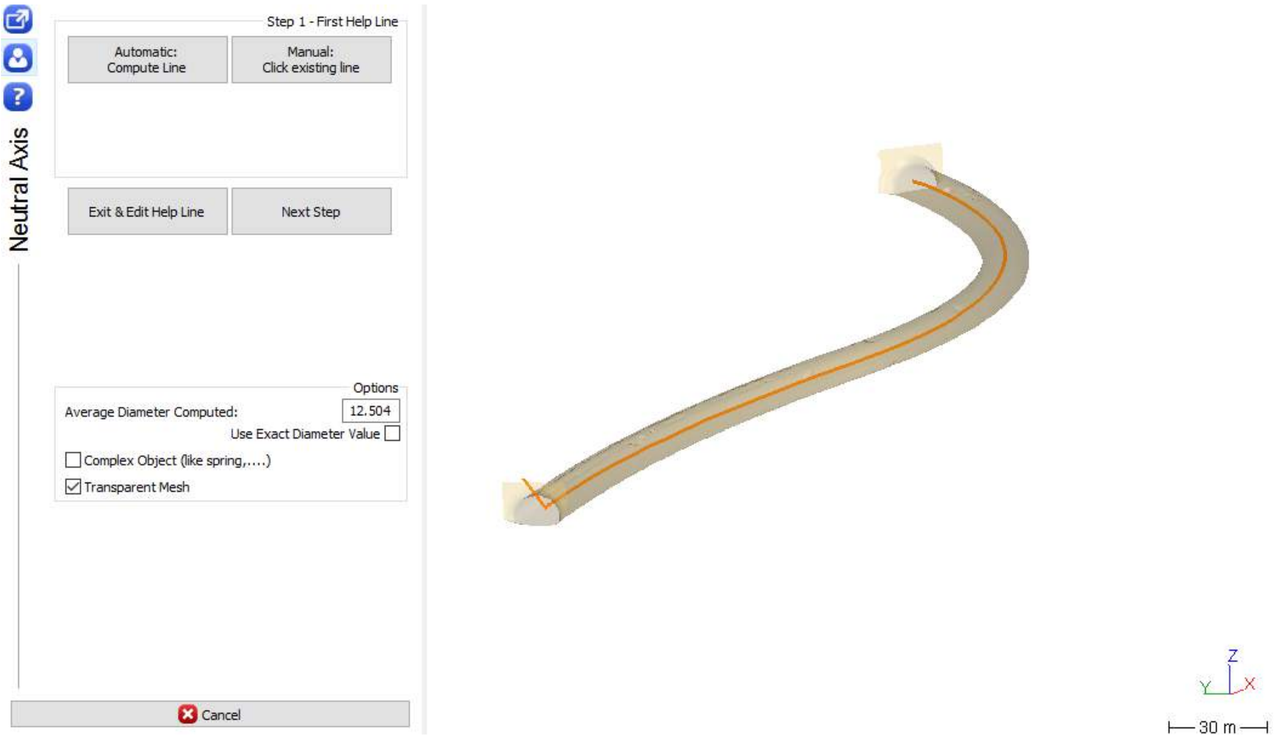
Having access to their neutral axis is helpful while working on tubular shapes; for example, for the inspection of pipes or tunnels. Afterwards, it could be possible to create sections on the shape along its neutral axis. The neutral axis can be extracted on a point cloud or on a mesh.

Open the file “CrossSections.rsh”. It contains a measured mesh of a tunnel named "Measured tunnel". Select it and show it only.

Select it again and go to Polyline \ Neutral axis.

Before computing the neutral axis, we need a “Help Line”, which represents an approximate axis of the shape. This “Help Line” can be automatically computed in the Step 1. You could also draw a polyline following the general direction of the mesh and use it as a Help Line.

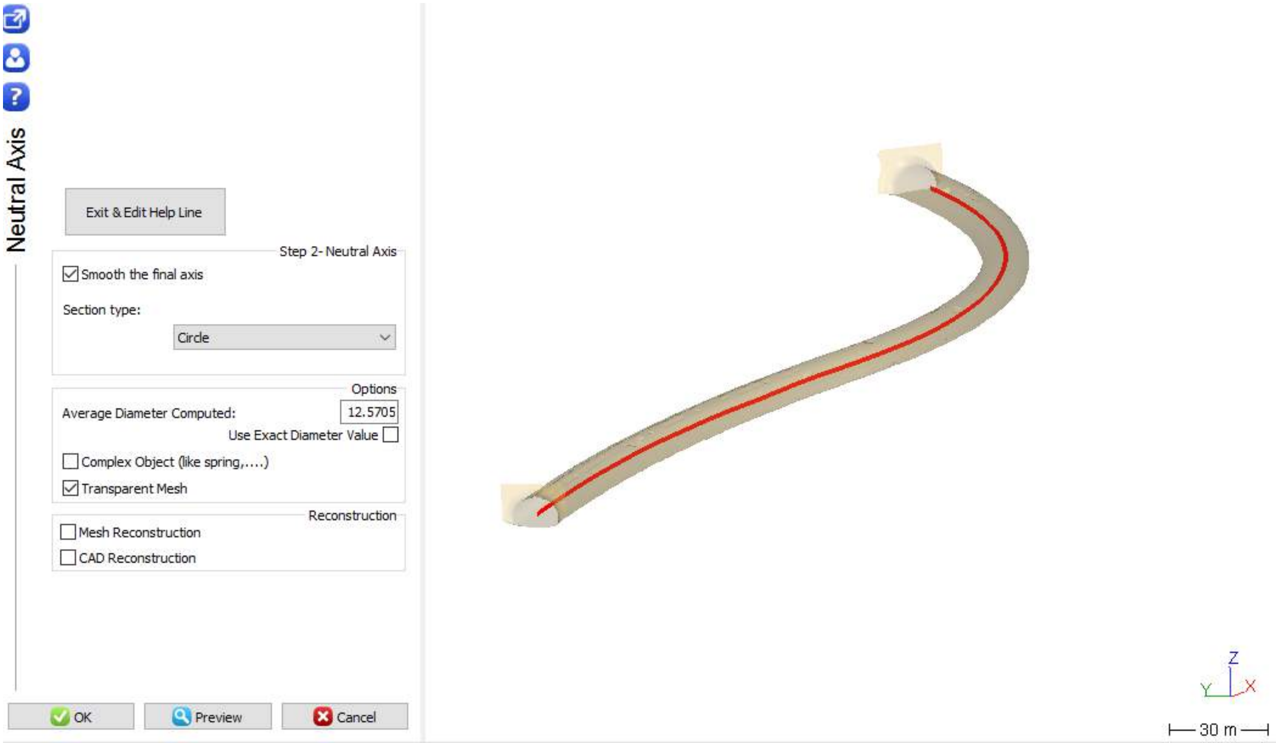
Here click on **Automatic: Compute Line**. Check the option **Transparent mesh** in order to see the axis inside the tunnel.



Compute first approximate of the neutral axis of a tunnel

If the Help line is not totally correct, it is possible to click on **Exit & Edit Help Line** to modify the line, and then launch the command again while selecting the mesh and the line.

Click on **Next Step**. In this second step, the precise neutral axis will be computed based on the Help line. Click on **Preview** to see the final axis computed. You can choose to smooth it or not. Click **OK** to validate the result.



Final result of neutral line extraction on the tunnel

9 Measurement, Inspection and reporting

- Measure volumes
  - Exercise: Measure the volume of a closed mesh
- Make measurements with the mouse
  - Exercise: measurements on a mesh
- Geometric features
  - Exercise: Create a geometric shape
  - Exercise: Compute best shapes from clouds and polylines
- Comparison & Inspection
  - Exercise: Compute inspection between a surface and a cloud
  - Exercise: Adjust inspection colors
  - Exercise: Compute inspection between polylines
- Labels & Reporting
  - Exercise: Create a complete report from a 3D inspection

9.1 Measure volumes

In the software there are 3 commands to compute volumes. They are in the section Measure \ Volume.

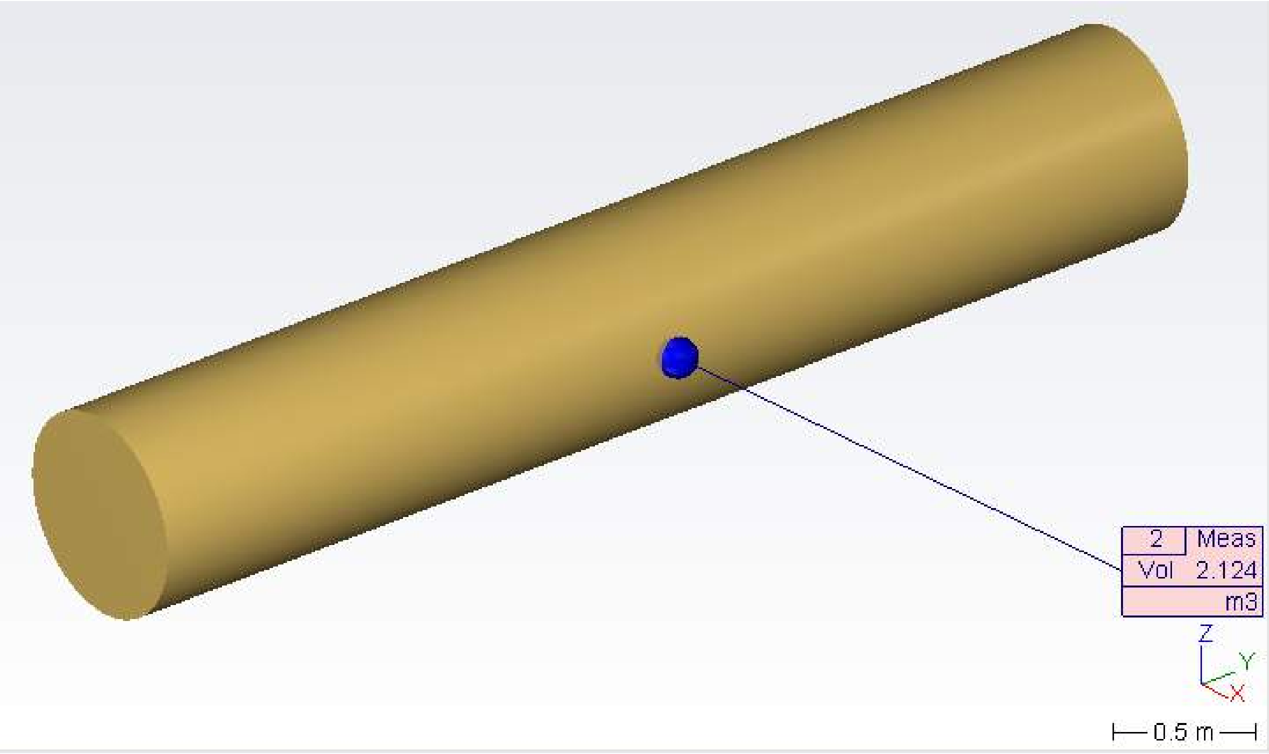
- Exercise: Measure the volume of a closed mesh

9.1.1 Exercise: Measure the volume of a closed mesh

Open the file "VolumeClosed.rsh". It contains the meshes of two sections of a pipe.

Select one mesh and go to Measure \ Volume \ Volume. If the mesh is not closed, a window with a warning message saying **Warning! The volume(s) computed is (are) approximated because the mesh(es) is (are) not closed** is displayed. This message also invites you to see the plugin Surveying if you are interested in computing cubature.

The result is displayed in a label.



Volume of a closed mesh
It is possible to select several meshes before launching the command.

It is good to know that the volume of a mesh can be expressed as a negative value if the normal of the mesh is oriented inside. In this case, you can right-click on the mesh and click on **Reverse** to reverse its normal.

> ⚠ When a mesh is closed, you can also see its volume using the **property** command by right-clicking on the object.

If you wish to compute cubature or volume between two meshes, you can use Measure \ Cubature.

9.2 Make measurements with the mouse

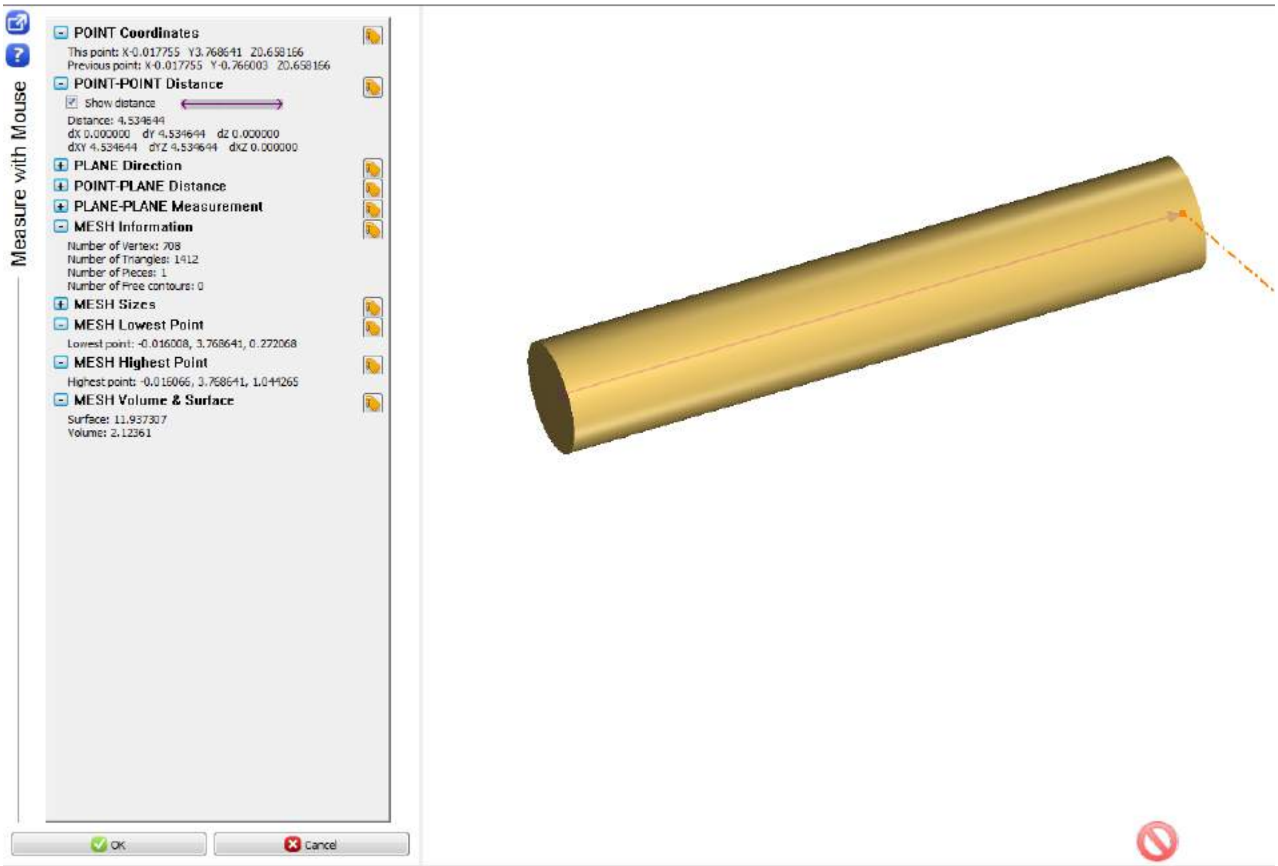
In the software there is a single tool in order to measure coordinates, distances, angles, etc. This command is Measure / Measure with mouse.

- Exercise: measurements on a mesh

9.2.1 Exercise: measurements on a mesh

Open the file “VolumeClosed.rsh”.

Show only the mesh “Straight Pipe” with a transparency of **60%**. Then launch the command Measure / Measure with mouse. Select only the option **Middle/Center** and click on the circle at the extremity of the cylinder. The center of each extremity is detected as in the following picture.



Measurement on a mesh

On the left of the screen, you can see all the information linked to the points you have clicked and to the object you have clicked (here a mesh). Here we have coordinates and distances as well as mesh information (number of triangles, dimensions…). If you click two planes, for example, you will also have angle information.

Near each measure information, there is a small button in order to create the label with the corresponding data. Thus, you can create a label per measure and associate a reference value to get the deviation error. All these labels can be exported later in a report.

9.3 Geometric features

In some cases it is preferable to directly use the geometric shape instead of a discrete version as, for example, a mesh or a polyline. The software offers you, therefore, the possibility to create features as:

- lines
- planes
- circles
- round slots
- square slots
- cylinders
- spheres
- and even Cones

directly using several Methods.

In the following exercises you will see how easy it is to create, extract and to use geometrical features:

- Exercise: Create a geometric shape
- Exercise: Compute best shapes from clouds and polylines

9.3.1 Exercise: Create a geometric shape

In the software several methods exist to define a geometric shape:

- draw
- best shape
- extract shape
- define
- using nominal shape
- from CAD-shape

A detailed description of the available commands and their options can be found in the help of the corresponding Construct menu.

Draw a Circle:

- For example, go to Construct \ Circle \ Draw and click points in the scene. Once three points are selected a circle passing through them will be appearing in the scene (three points define a circle). If you continue clicking points, the created shape will be the circle closest to the input.
- You can constrain parameters of the shape: enable the checkbox **radius** and change its value. You will see that even if you continue clicking points the radius will be fixed to this value. In this way, you can supply additional external information about the shape to the algorithm.

Removing points

As you can add points by clicking on the scene you can also remove them with the **DEL** key.

Define a Circle:

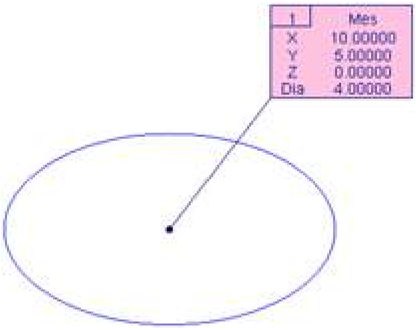
A shape can also be defined in a more mathematical manner, by fixing its parameters directly. For this purpose chose the function Construct \ Circle \ Define and:

- Enter the point X=10, Y=5, Z=0 for **Fixed center**.
- Enter the vector X=0, Y=0, Z=1 for **Fixed axis**.
- Enter 2 for the **Radius**.

Using tools to define parameters

You can use the available tools to fix the Center and the Normal direction of your circle by clicking in the scene.

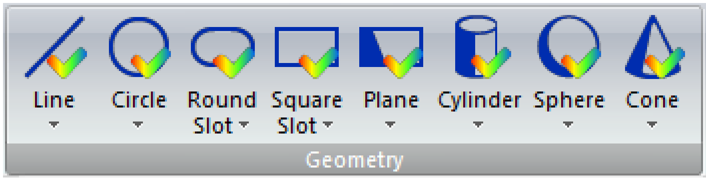
Once the circle is validated, you can check that the circle you created has the desired properties by selecting it and using Measure \ Measure with the mouse to create a label summarizing them.



Measure of the created circle

9.3.2 Exercise: Compute best shapes from clouds and polylines

Beside the creation of geometric shapes by clicking points or by directly defining its parameters, you can extract a shape by fitting it to existing objects (meshes, clouds, points, other shapes, ...).

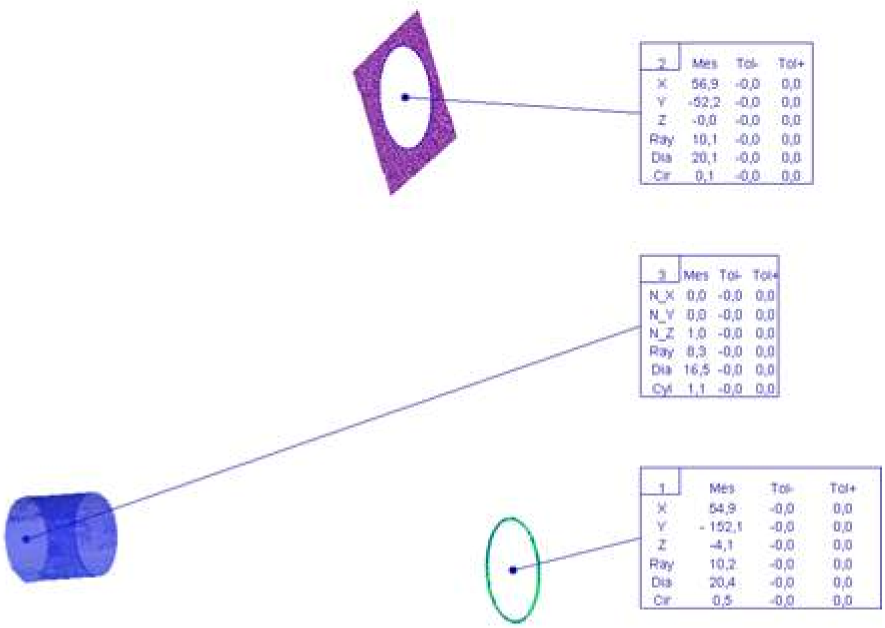


Geometric shapes available in the software

The following exercise will illustrate how to extract shapes and their properties from potentially measured point clouds or polylines.

Open the file “BestShape.rsh”.

- To begin, select the polyline **Circle** and open the dialog Measure \ Circle \ Best Circle. Select the calculation method **All on (Min Std Dev)** since we want to compute simply the best circle fitting the data. Adjust the slider at the middle of the dialog box to eliminate noisy points. Check the box **Create Label** in order to create a label and validate the circle.
- Now, select the cloud **Circle Inner Cloud** and go to Measure \ Circle \ Best Circle. By selecting the calculation method **Inner**, the best circle corresponding to the hole of the point cloud is calculated. Check the box **Create Label** in order to create a label and validate the circle.
- Finally, select the cloud **Cylinder** and go to Measure \ Cylinder \ Best Cylinder. Select the option **Fix radius/diameter**, set the radius to **8.25**, also the option **Fixed axis** and fix the Normal direction to be the Z-axis. You can adjust the slider at the middle of the dialog box to eliminate noisy points. Once you have chosen all the parameters and you are satisfied with the result, you can enable the checkbox **Create Label** in order to create a label and validate the cylinder.



Best shapes computed from point clouds or polylines

9.4 Comparison & Inspection

Quite often you need to compare two objects in order to:

- Inspect your measure with the reference CAD.

- Compare the created model with the original point cloud.
- Check the position of your scan.
- Etc.

In the software you can compare:

- A cloud with a mesh or a surface or a geometric feature.
- A mesh with another mesh, a surface or a geometric feature.
- A polyline with another polyline.
- A set of polylines with another set of polylines.

You can compute inspections only between 2 and only 2 objects (you can use commands Merge Clouds, Group in Compound Mesh or CAD \ Create Compound if you want to group several objects). The one selected first is the “Reference” and the other one is the “Measure”.

- Exercise: Compute inspection between a surface and a cloud
- Exercise: Adjust inspection colors
- Exercise: Compute inspection between polylines

9.4.1 Exercise: Compute inspection between a surface and a cloud

Open the file “BestFitOnRef.rsh”.

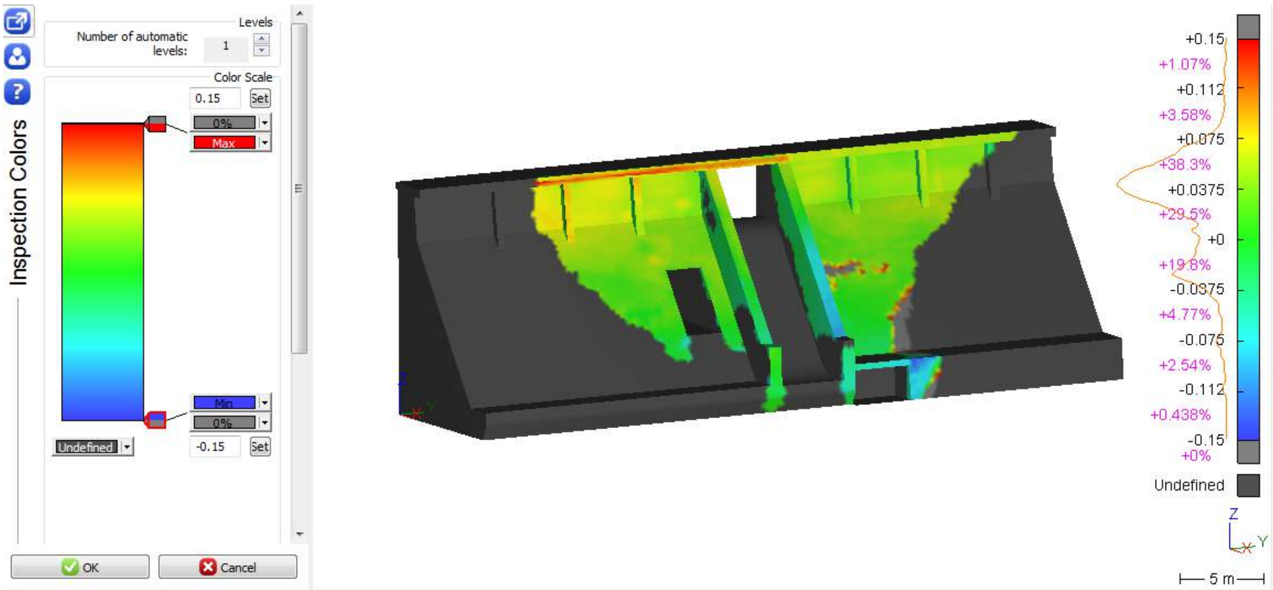
Show only the cloud **Aligned Dam** and the Mesh **Theoritical Dam (good CS)**. Then select the Mesh **first**, as it is the reference, and then with then **CTRL** key select the cloud, and go to Measure / Compare / Inspect .

Choose to apply the color on the reference and select the **3D Inspection** method.

The 2D inspection will project all the points along a given direction (for example Z), while the 3D inspection will look for the closest point on the reference in 3D. In most cases, we compute 3D inspection, except when deviations are only required in one direction

Click **Preview** to compute the inspection. Click **Edit color** to edit thresholds and colors (see Adjust inspection colors for more information). Validate the colors by clicking **Ok**. Click **Ok** again to validate the final result.

A new object called **Compare Theoritical Dam (good CS) / Aligned Dam 1** has been added in the Measure Group.



Comparison between a cloud and a mesh

9.4.2 Exercise: Adjust inspection colors

Each time you have a color mapping, you can customize the color scale according to your needs. There are different kinds of color mapping, they can be:

- a result of an inspection,
- a cloud with intensity values,
- a mesh where curvatures have been highlighted: Measure \ Show curvature on mesh,
- a cloud extracted with all commands in Measure menu allowing to extract a cloud (command 'Best', 'Extract' and 'Using Nominal'), or
- a cloud or a mesh colored along a direction: Measure \ Color Along a Direction

Open the file "BestFitOnRef.rsh"

Select the result you have computed in the previous exercise and go to Measure \ Edit Colors.

As you can see in the result of the previous exercise, the coloring has been automatically divided into 1 part going from blue for the minimum deviation value to red for the maximum deviation value.

You can easily change this representation by modifying the **number of automatic levels**.

You can customize the color scale by moving all cursors on the right of the scale in order to reduce or increase a level. When you select a cursor you can:

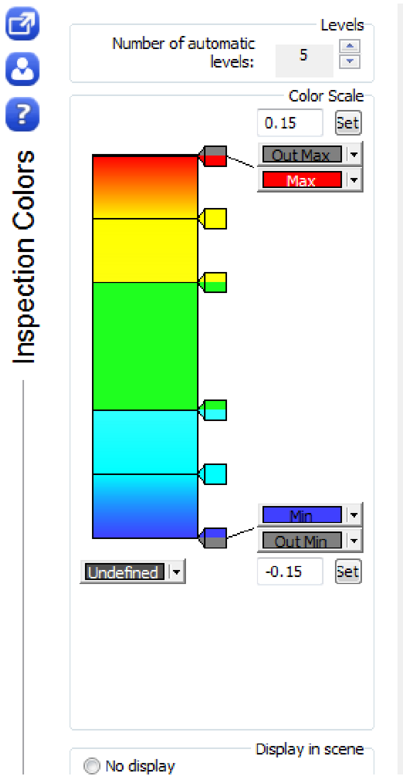
- change colors on both sides of the cursor with button **Cursor max** and **Cursor min,**
- change the cursor value (field just below **Cursor min**), or
- remove it by pressing **DEL** on the keyboard.

Each cursor has two colors:

- One color for the values above the cursor value.
- One color for the values below the cursor value.

Try to reproduce the color scale from the picture below. You have to:

- set the number of levels to **5**,
- set the max value to **0.15** and the min value to **-0.15**,
- set other cursors to **0.1**, **0.05**, **-0.05** and **-0.1**,
- change colors above the cursor **0.1** and below the cursor **-0.1**, and
- select the option **Foreground** in order to display the color scale in the 3D scene.



Edit color mapping

If you select two consecutives cursors with the **CTRL** key, you can directly edit the color of the complete zone between the two cursors.

If you enable the advanced mode, you can save or recall customized color representations.

Click **OK** to validate.

9.4.3 Exercise: Compute inspection between polylines

Open the file “CrossSections.rsh”

It contains:

- the mesh of a measured tunnel
- a theoretical section of the tunnel at a given kilometric point

Select and show these two elements only.

Create a section of the measured mesh

As a first step, we will create a section of the measured mesh in the same plane as the theoretical section:

1. select the mesh and launch Polyline / Planar section,
2. define the plane section position with the  button and select a point on the theoretical section,
3. define the plane orientation with the  button, choose the  option and click on the theoretical section to define the orientation as the section plane,
4. uncheck the option creating sections All over and enter 1 as the umber of sections, and
5. click OK, Exit.

This creates a section named **Section 1** in the Contour Group.

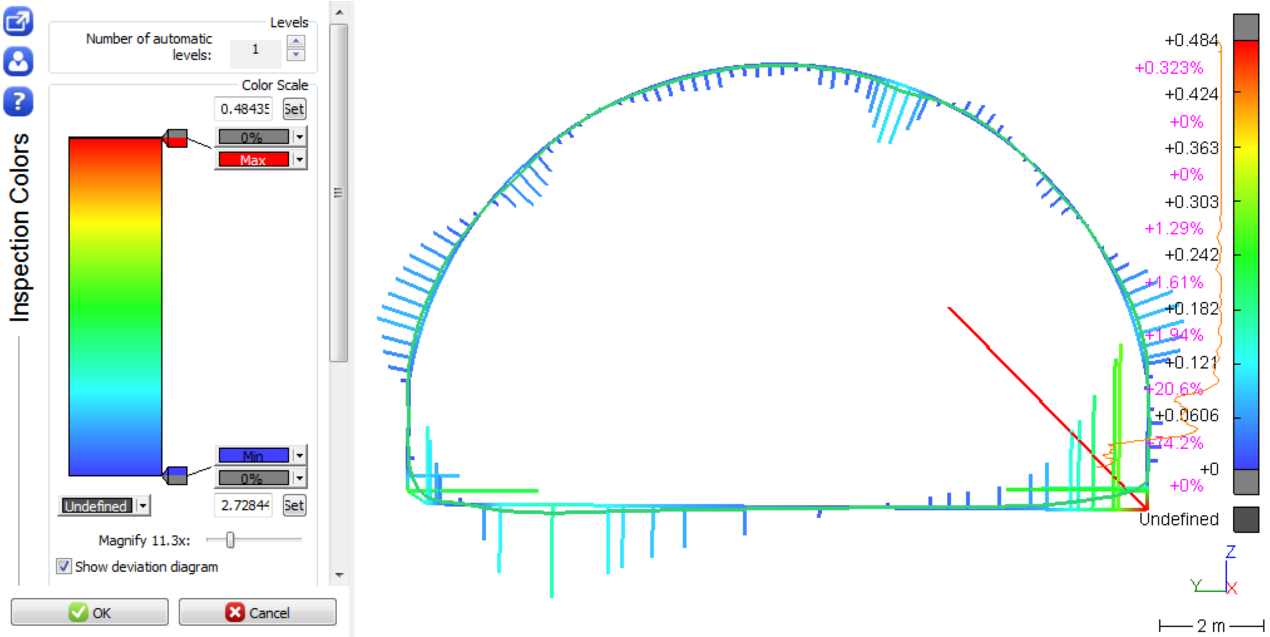
Inspect the polylines

Select and show only the two polylines **Theoretical section** and **Section 1**. Select first **Theoretical section**, as it is the reference, and then select **Section 1** with the **CTRL** key, as it is the measure.

Launch the command Measure / Compare / Inspect.

Choose to apply the color on the object to project **Theoretical section**, then uncheck options to **Ignore points with distance greater than** and **Ignore points projected on edges**. As the two polylines are in the same plane, you can check the option **Unsigned inspection**.

Click **Preview** to compute the inspection, then click on **Edit colors** to magnify the distances and change thresholds and colors (see Adjust inspection colors for more information). Click **OK** to validate the display. Then click **Ok** again to validate the inspection. A new object called **Compare Theoretical section / Section 1** is added in the Compare Inspect group.



Inspection between two polylines

If you select two sets of polylines, the software will automatically create pairs of contours and then compute several inspections of two polylines.

9.5 Labels & Reporting

- Exercise: Create a complete report from a 3D inspection

9.5.1 Exercise: Create a complete report from a 3D inspection

Open the file “BestFitOnRef.rsh".

Show only the inspected mesh **Compare Theoritical Dam (good CS) / Aligned Dam 1** located in the Measure Group and make a **Zoom All**.

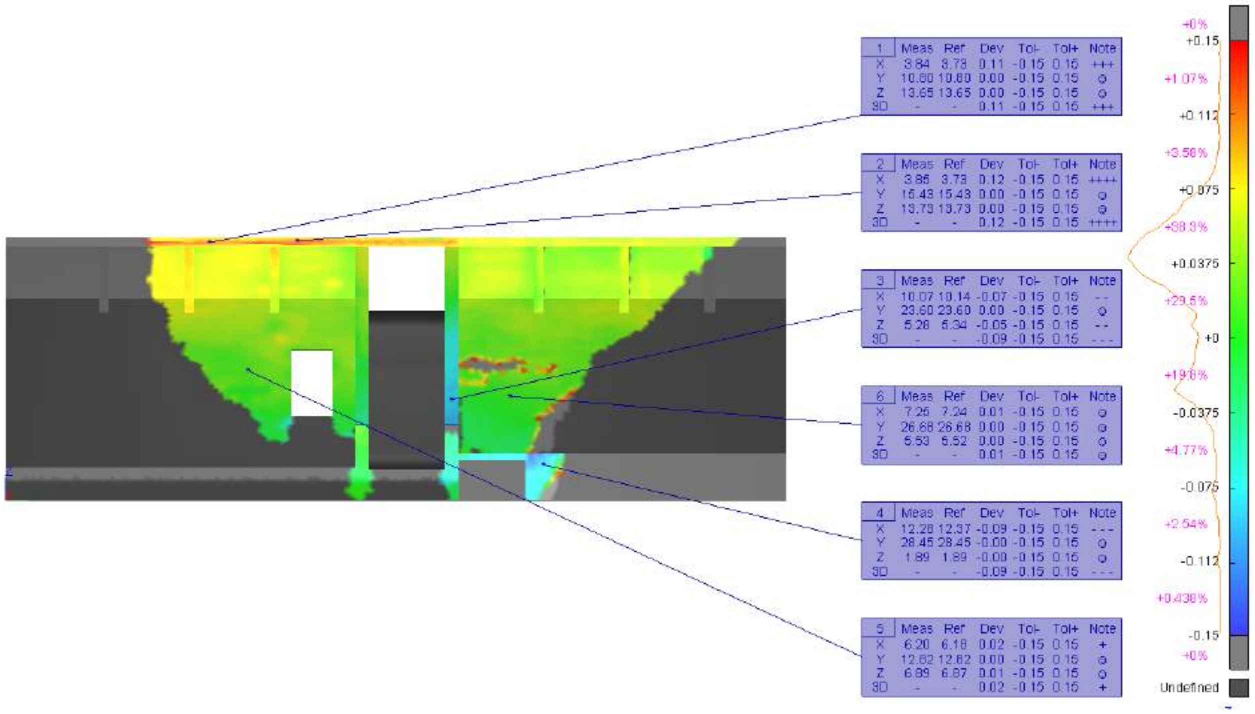
Create labels

First, launch the command Measure \ Label Aspect in order to customize labels aspect. Select the label type **long** and the option **Take automatically a smaller label if necessary**. Then, click **OK**.

Now launch the command Measure \ Create / Edit Label and check the option **Get tolerance from min /max limit values** Then select the option **Point on Selection** in the software's ribbon and click some points on the color mapping. Click, for example, points with different colors (red, light blue, dark blue, green, yellow…).

As you can see in the label, there is a column with some "+" or "-" where the number of "+" or "-" tells you how far you are from the middle of the tolerance. A smile tells you that you are really in the middle of the tolerance. A "!" tells you that you are out of tolerance.

To add a comment, select the option **Modify when clicking**, then enter a comment in the corresponding field in the bottom of the dialog box. Click on the label where you want to add the comment. Then click **OK** to validate.



Labels created from the 3D inspection

If you click many points the function will automatically choose a smaller label size so everything fits in your screen.

Edit labels

Select the label(s) you want to edit and launch the command Measure \ Create / Edit Label. You will be able to change tolerance, comments, etc. You can also set which elements you want to show in the label thanks to the check boxes.

Create View Sets

If you want to add some views in the report, you have to create some view sets.

Show only the color mapping and one label and then go to View \ Create View Set. Enter a name and select the option to store the visibility. Click **OK** to validate.

Now show all the labels, change the view and create a new view set.

Select the first view set in the tree explorer (in the **Other Objects** folder). Do a right-click and select **Show**. It will restore this view and you should see only one label. Now do the same with the second view set. All labels are visible now.

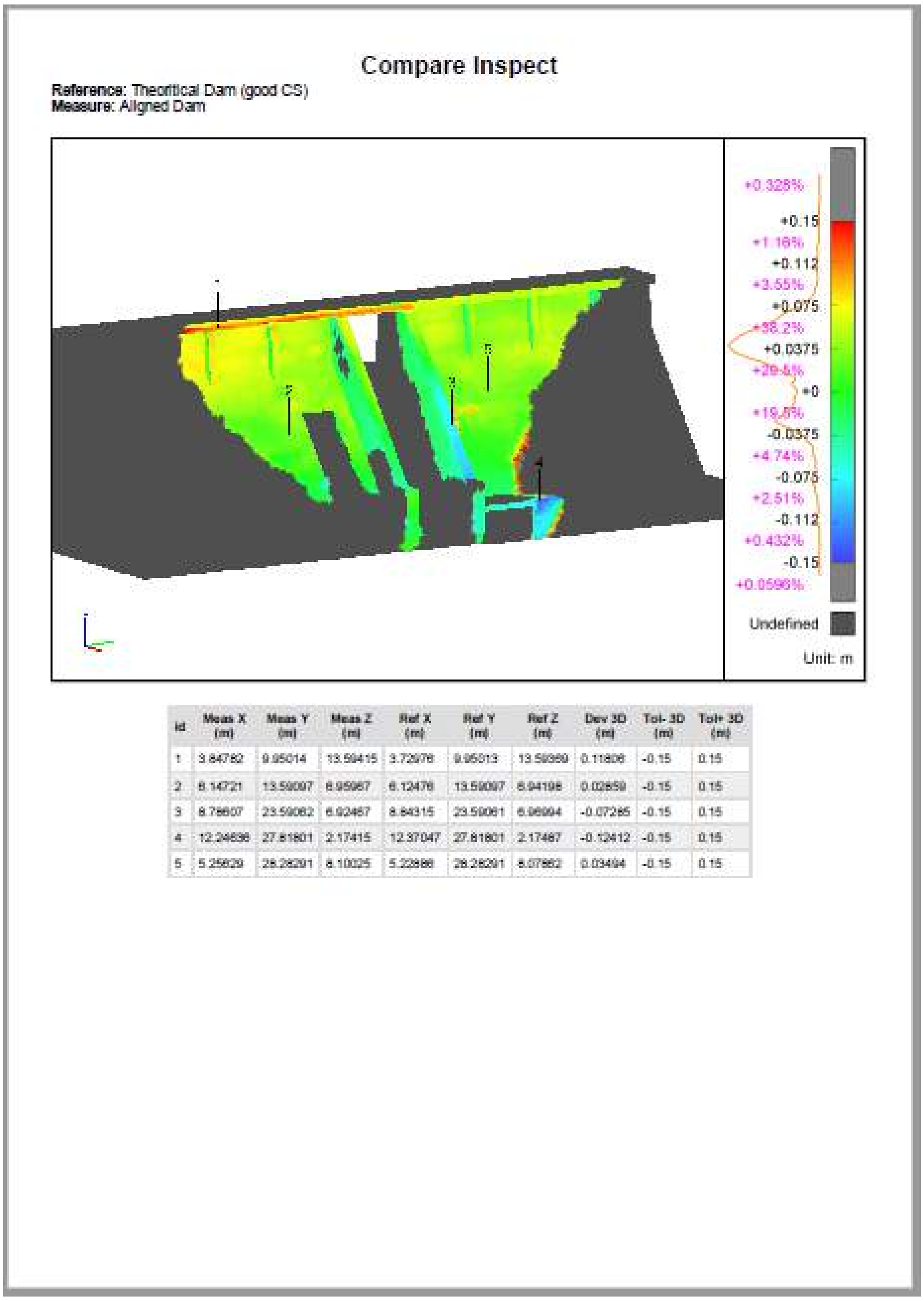
Customize and export a report

Once you have created labels and/or view sets, you can edit a report. Note the labels have been created into Compare Inspect folder.

Launch File \ Report Editor or launch the editor thanks to the magnifying glass corresponding to the report data. The chapter **Compare Inspect** has been automatically generated with the default template assigned to label report data category.

First, define the report layout (**paper format, margins, orientation, header, footer and number of decimals**). For this exercise, remove the cover chapter. You can add or remove unnecessary cells (refer to Template View). Note while inserting an item into a cell, the report editor will make you some suggestions. Otherwise, you can select this data from the Data panel or write it by yourself. When you insert a picture, the image size and ratio are always respected. Consequently, if you want to reduce the image size, you have to reduce the cell width.

- select the scene and set the **mode** on 3D to insert a 3D PDF in your report,
- select the table and filter the colums: show only id, Meas X, Meas Y, Meas Z, Ref X, Ref Y, Ref Z, Dev 3D, Tol- 3D and Tol+ 3D. Align the table to center thanks to the Options panel,
- optionnaly, insert another cell to display another scene (in 2D mode) using a viewset previously defined, and
- click the “**To PDF"** button to create and display a report in .pdf format. Then, you can print the document as usual.



Compare Inspect

Reference: Theoritical Dam (good CS)
Measure: Aligned Dam

id	Meas X (m)	Meas Y (m)	Meas Z (m)	Ref X (m)	Ref Y (m)	Ref Z (m)	Dev 3D (m)	Tol- 3D (m)	Tol+ 3D (m)
1	3.84782	9.95014	13.59415	3.72976	9.95013	13.59369	0.11806	-0.15	0.15
2	6.14721	13.59097	6.95967	6.12476	13.59097	6.94198	0.02859	-0.15	0.15
3	8.78607	23.59062	6.92467	8.84315	23.59061	6.96994	-0.07285	-0.15	0.15
4	12.24636	27.81801	2.17415	12.37047	27.81801	2.17487	-0.12412	-0.15	0.15
5	5.25629	28.28291	8.10025	5.22988	28.28291	8.07862	0.03494	-0.15	0.15

3D Inspection report

10 Surveying

- Cross sections
  - Exercise: Tunnel analysis
- Surface analysis
  - Exercise: Complete analysis of a concrete floor
- Surveying extraction
  - Exercise: Automatic extraction of the center of surveying targets
- Surveying modeling
  - Exercise: Automatic creation of a Digital Terrain Model from a point cloud
  - Exercise: Create the simple model of a building
- Volume and cubature
  - Exercise: Compute cubature between two open meshes
  - Exercise: Measure cubature with a level of water
  - Exercise: Measure the volume of a closed mesh
  - Exercise: Create a stockpile project

10.1 Cross sections

- Exercise: Tunnel analysis

10.1.1 Exercise: Tunnel analysis

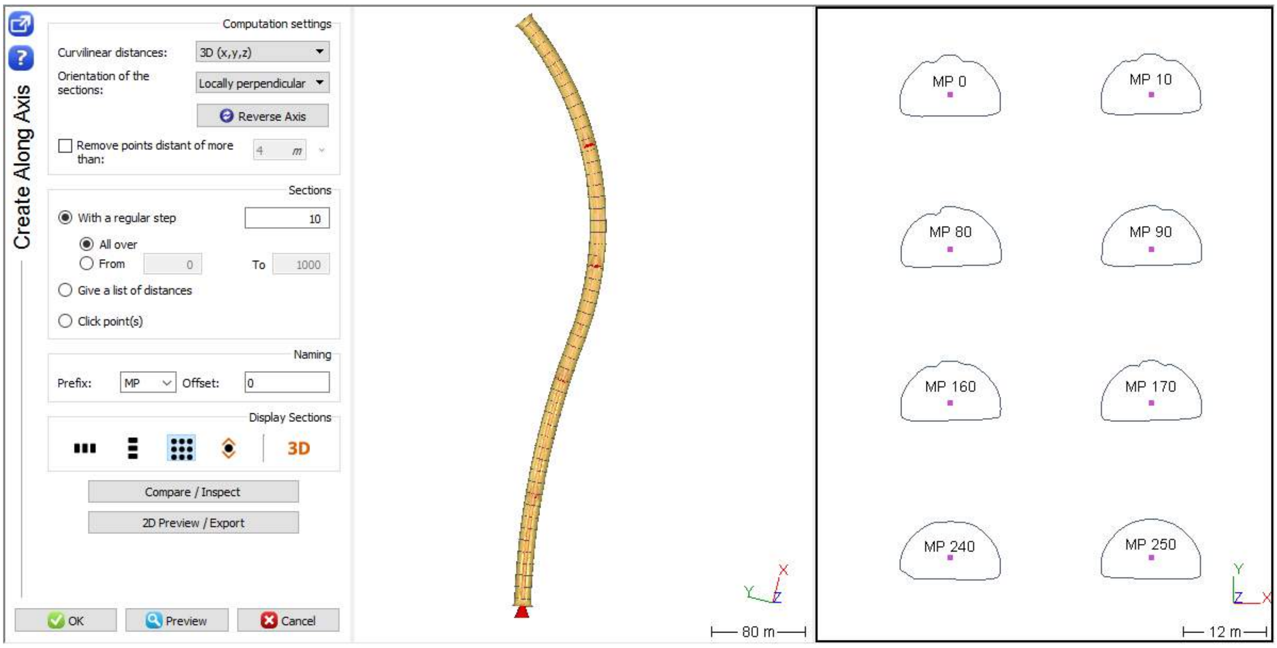
In the surveying field it is common to draw cross sections on a building, on a road or on a structure, in order to inspect it while it is being built or for periodic controls. The following commands have been developed taking into account the specific needs of tunnel inspection. They can of course be used for other applications, having similar needs.

Open the file

Open the file “CrossSections.rsh”. It contains the mesh of a tunnel that has been scanned and the theoretical mesh of the tunnel which has been created by extrusion of the theoretical section along the neutral axis. This file is going to be used through this whole exercise.

Create cross sections

First, we are going to create sections on the measured tunnel along the neutral axis. Select the mesh **Measured tunnel** and the polyline **Neutral axis** and go to Surveying \ Create along axis. You can see arrows appearing on the neutral axis to indicate its direction. It is possible to reverse it with the button **Reverse Axis**. Choose where to create the cross sections along the neutral axis. They can be created all over the axis with a regular step, or only on a certain part with a regular step, or at specific distances. First try the option **All over** with a regular step of **10** meters. The distances to enter are curvilinear distances along the axis: choose **3D (x, y, z)** and **locally perpendicular** to the neutral axis.



Create cross sections on a mesh along the neutral axis

When you click on **Preview**, the created cross sections will be displayed in a 2D layout. You can choose to display them on a line, on a column, in a grid or individually. It is also possible to preview them in 3D with the button **3D**. The scene is then split vertically in two views, with the 3D objects displayed on the left and the 2D sections previewed on the right.
The cross sections are displayed with their name composed of an optional prefix and their curvilinear distance on the axis. In this example we have **MP** as a prefix meaning “Milepost”. You can write your own prefix in the box **Prefix**.

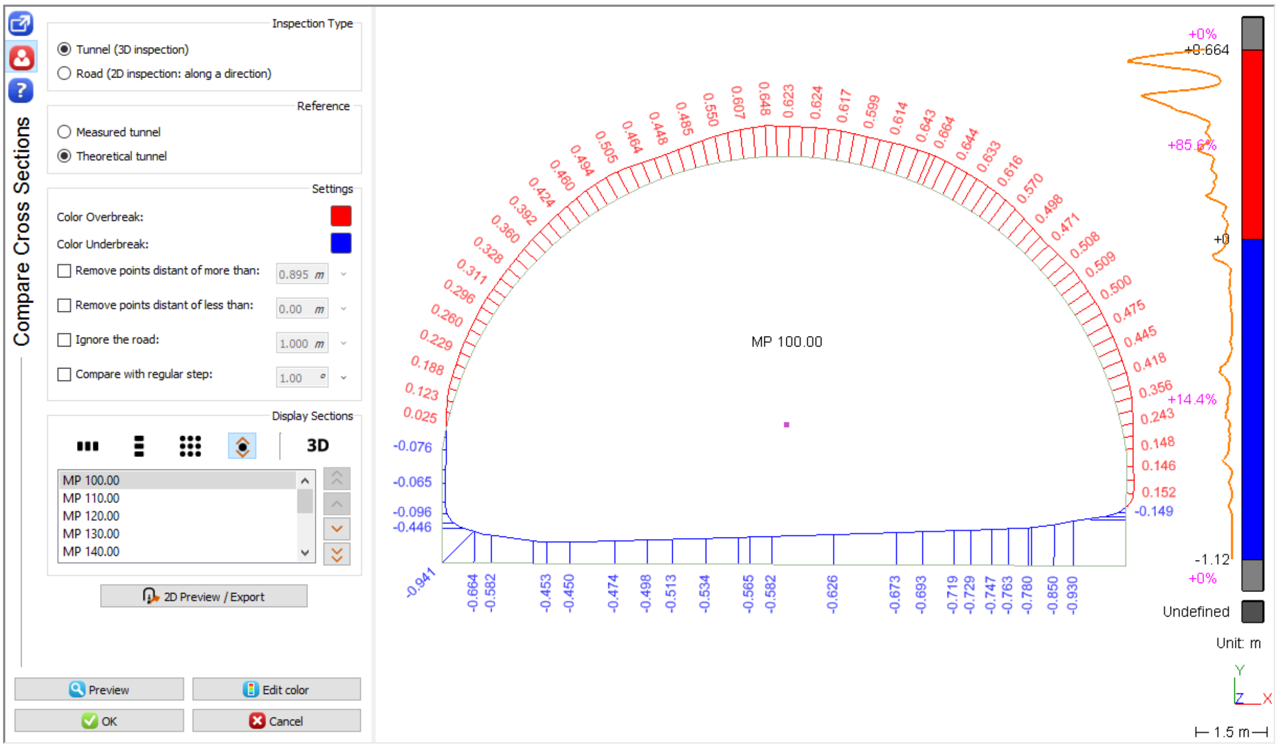
> From this dialog box it is possible to export the created sections in their 2D layout by clicking on **2D Preview / Export**.

Click **OK** to validate the results. You can see a new folder created in the Contour Group named **Cross sections: Measured tunnel**. This folder contains all the sections created, named after the mesh they were created on, the prefix you chose and their distance on the axis.

Compare cross sections

Keep using the file “CrossSections.rsh”. We are going to compare cross sections from the measured tunnel with the theoretical tunnel. Show only the neutral axis and both meshes (the measured tunnel and the theoretical tunnel). Select them and go to Surveying \ Create along axis. This time, create sections **From 100 to 210** in order to compare sections only on a specific part of the tunnel. Choose to create them at a regular step of **10** meters. Click on **Preview** to compute the sections and then on **Compare / Inspect** to go directly to the command Surveying \ Compare / Inspect. You could also click first on **OK** to validate the result and then select the created folder containing all the sections and go to Surveying \ Compare / Inspect.

Set the **inspection type**: it is a tunnel (3D inspection). Choose which mesh is the **reference**, so here choose **Theoretical tunnel**. In advanced mode, several options are available to configure the comparison of the sections. It is possible to ignore deviations greater or smaller than a given value, deviations measured under a certain height to ignore points on the road of the tunnel. You can also measure deviations with a regular step along the section.



Compare cross sections

Deviations can be positive or negative, depending on whether the projected point is outside the reference section or inside. The colors of the deviations can be changed in order to distinguish overbreaks from underbreaks (see the picture above). It is possible to edit the colors or increase the size of the deviations by opening **Edit color**:

- check the option **Show deviation diagram** to display hairs,
- check the option **Show quotation texts** to display them in the 2D layout,
- check **ForeGround** or **BackGround** to display the color scale in order to see the distribution of the deviations and the values corresponding to the colors, and

- check the option **Frame extreme values** to display a frame around the highest values of the deviations for each couple of sections.

As in the previous dialog box, you can choose the layout of the cross sections in their 2D preview, and the display of a 3D view.
Click **OK** to validate the results. The colored polylines resulting of the comparison are automatically added to the folder containing all the cross sections.

Preview and export cross sections in a 2D layout

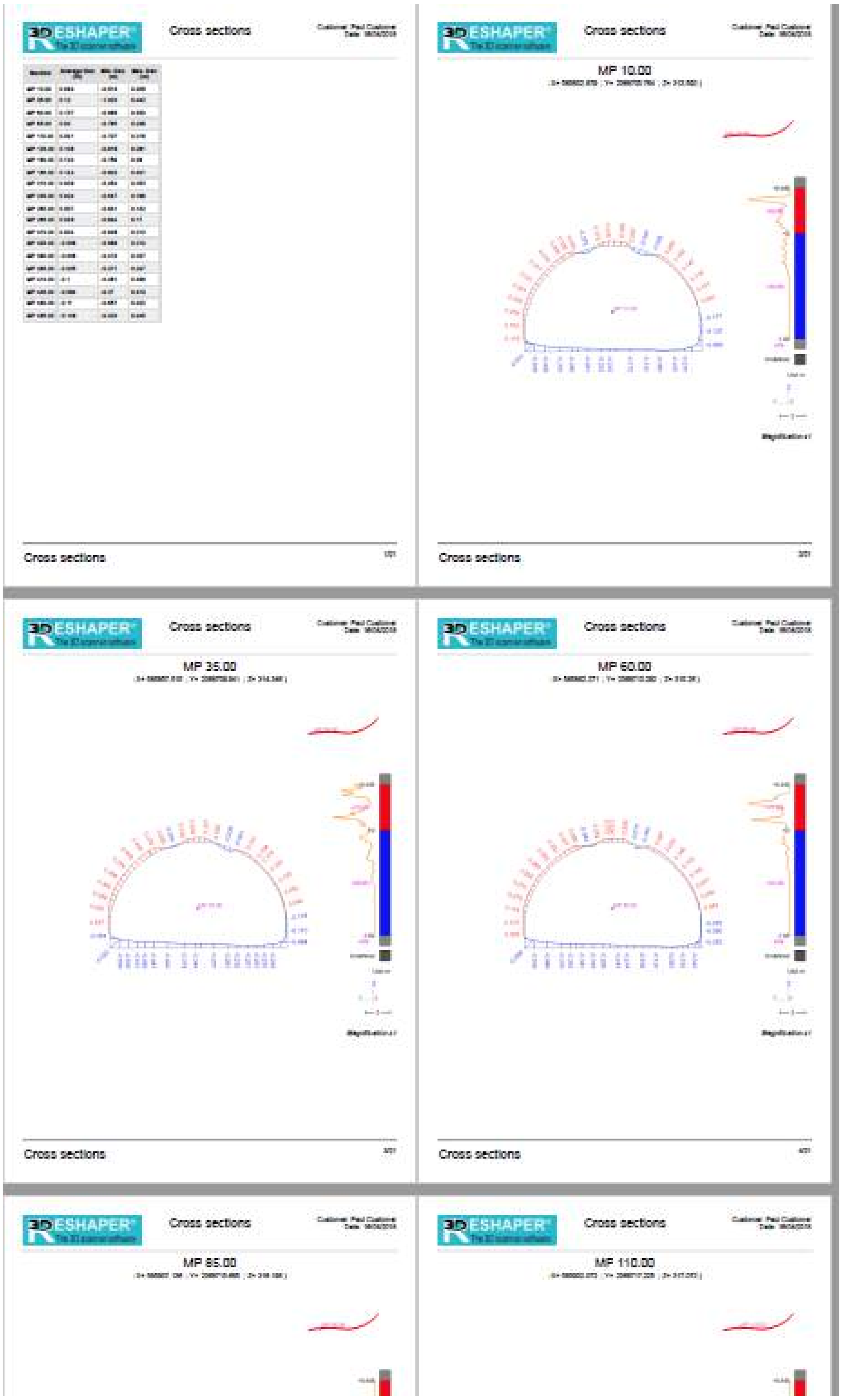
Select the folder containing all the cross sections, the intersection points and the compared polylines and go to Surveying\2D Preview / Export. This command is useful to preview at any time the cross sections in a 2D layout, and also to export them in this layout. The real coordinates of the intersection points between the neutral axis and the plane of the sections can also be displayed, as well as the colored polylines and the corresponding quotations texts.
Everything can be exported in one single DXF file or sent directly in AutoCAD.

Print cross sections in a dedicated template

Launch File\Report Editor to open the report editor. You may have two chapters called **Cross sections** in the default report (because you have just created two cross sections folders in your project). Select one chapter: in the template view, you can read the number of sections inside a dataset. Remove that containing 52 sections and the **Cover** chapter.

1. define the paper format and orientation in layout panel: A4+Portrait. Reduce the number of decimals to apply to distances. Apply header and footer everywhere.
2. complete header and footer with logo, date, customer or company name, title, page number, etc. (refer to Template View). You can modify or create your own fields in Data panel,
3. edit the dataset, go to item n°10. Click on
   
   to unlink this item. Then, remove the side view and modify the main view (manual scale 1:500 and a 1/1 ratio). The ratio enables you to limit a scene height. Thus, you can put several objects in the same page,

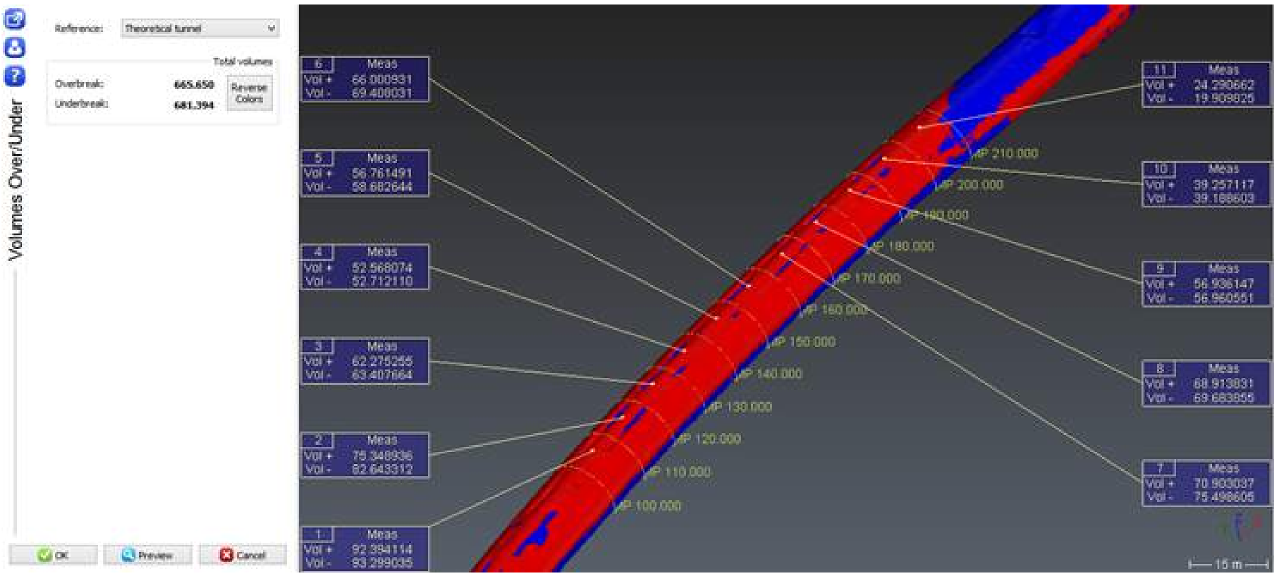
Now click **To PDF** to create the pdf report corresponding to the defined template. Then, open the pdf and print it like any pdf document.



Print cross sections in a dedicated template

Compute the volumes in overbreak and underbreak of a tunnel

Keep using the file “CrossSections.rsh”. We are going to compute the volumes in overbreak and underbreak between the measured tunnel and the theoretical tunnel on specific sections. Select both meshes and the folder containing the sections previously created on them with the command Surveying\Create along axis then go to Surveying\Volumes Over/Under. Choose which mesh is the reference. Here, select the Theoretical tunnel. Click on **Preview** to preview the results. The two meshes are entirely compared to display a 3D mesh colored with two colors: red for overbreak areas and blue for underbreak areas. The volumes are computed for each part of the tunnels defined by two couples of cross sections. The method used to compute the volumes is the interpolation between cross sections. Subsections are automatically computed between the couples of cross sections selected to compute a precise result. Through the advanced parameters it is possible to reduce or increase the number of subsections by moving the slider on **Low** or **High**. The default slider is on **Medium** to have the best ratio between precision and response time. A label is tied to each part, showing the volumes of overbreak (Vol+) and underbreak (Vol-). The total volumes of all the parts are displayed in the dialog box. The labels are inserted in a new folder in the Measure Group.



Compute volumes of overbreak and underbreak

10.2 Surface analysis

- Exercise: Complete analysis of a concrete floor

10.2.1 Exercise: Complete analysis of a concrete floor

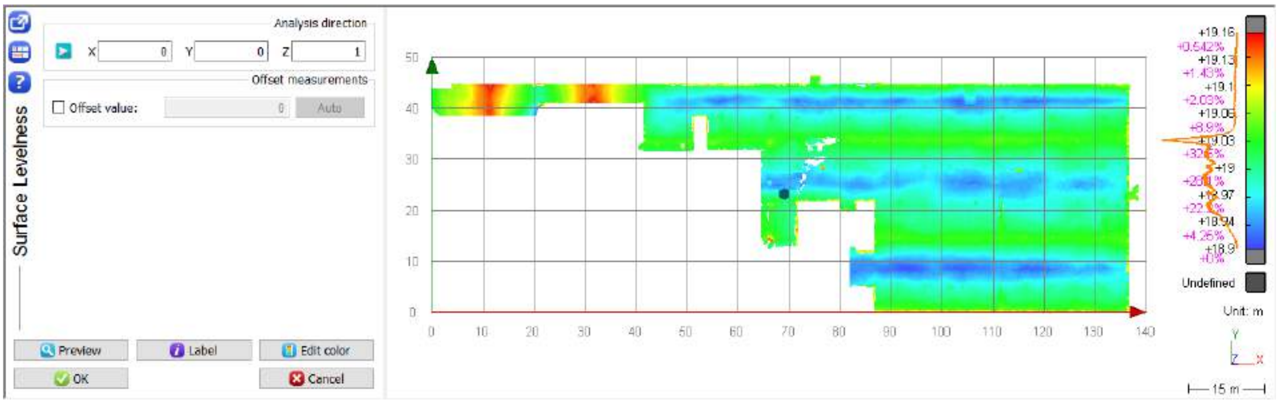
In this command Surveying\Surface Levelness, you will find complete tools to inspect a surface. You can check the levelness of a floor, the verticality of a wall, the flatness of a road, the local slopes on a terrain.

Check the levelness of a floor

Open the file “SectionsBuildingPlan.rsh”. It contains one cloud of the walls and one cloud of the floor.

Show only the cloud **Floor**. Select it and go to Surveying\Surface levelness. Choose the **Z** direction to check the levelness, uncheck **Offset value** and click on **Preview**. The points of the cloud are colored along their Z coordinate. You can now see the lowest points in dark blue and the highest points in red.

In this command and all commands from Surveying\Surface analysis, you can create labels on specific points.

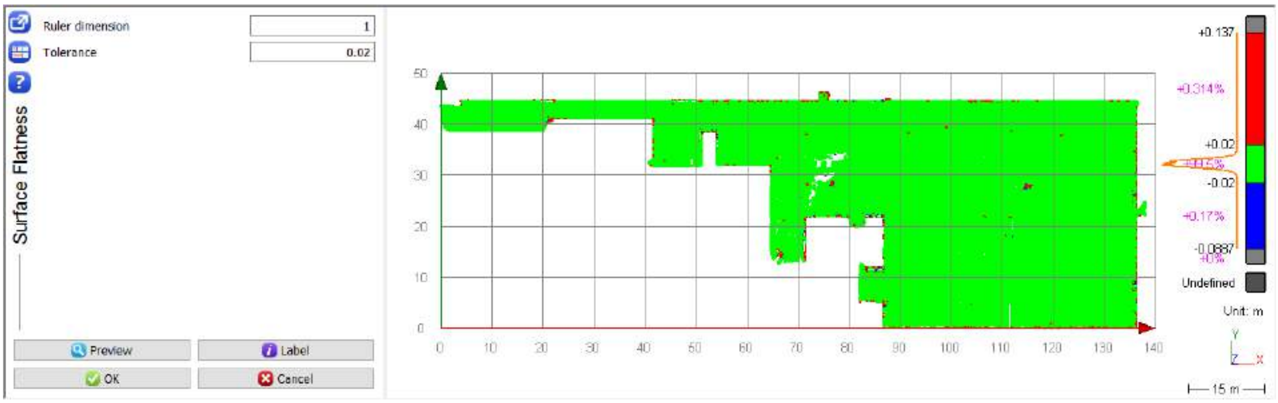


Check the levelness of a concrete floor

Check the flatness of a surface

This command allows checking if there are bumps of holes in a planar surface. This check can be done on a horizontal surface like a floor, but also on any other surface. Select again the cloud **Floor** and go to Surveying\Surface flatness. Give the parameters to check the flatness: set **1**m for the Ruler dimension and **0.02**m for the tolerance. This means that if you put a 1 meter long ruler on the floor, you expect no point from the floor to be further than 2cm from the ruler.

Click **Preview**. We can see here that 99.5% of the floor matches the tolerance. This means this floor is flat regarding this tolerance. There are only a few red and blue points where the posts and walls are.

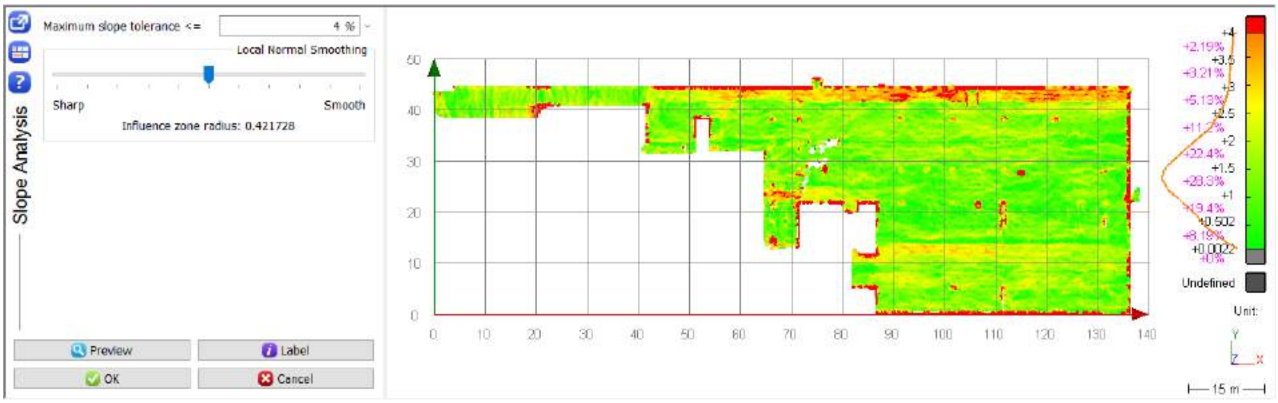


Check the flatness of a concrete floor

Check the local slope on a surface

The last command allows measuring the local slope on each point of a cloud or a mesh. Select again the cloud **Floor** and go to Surveying\Slope analysis. Change the unit of the slope to percentage (%) and set **4**% as the **maximum slope tolerance**. In this way you will check that the entire floor does not contain a zone with a slope higher than 4%. Set the **local normal smoothing** cursor in the middle to compute the slope on zones that are approximately 40cm wide. Click **Preview** to preview the results.

We can see that red zones are only where there are walls and posts.



Check the slopes on a concrete floor

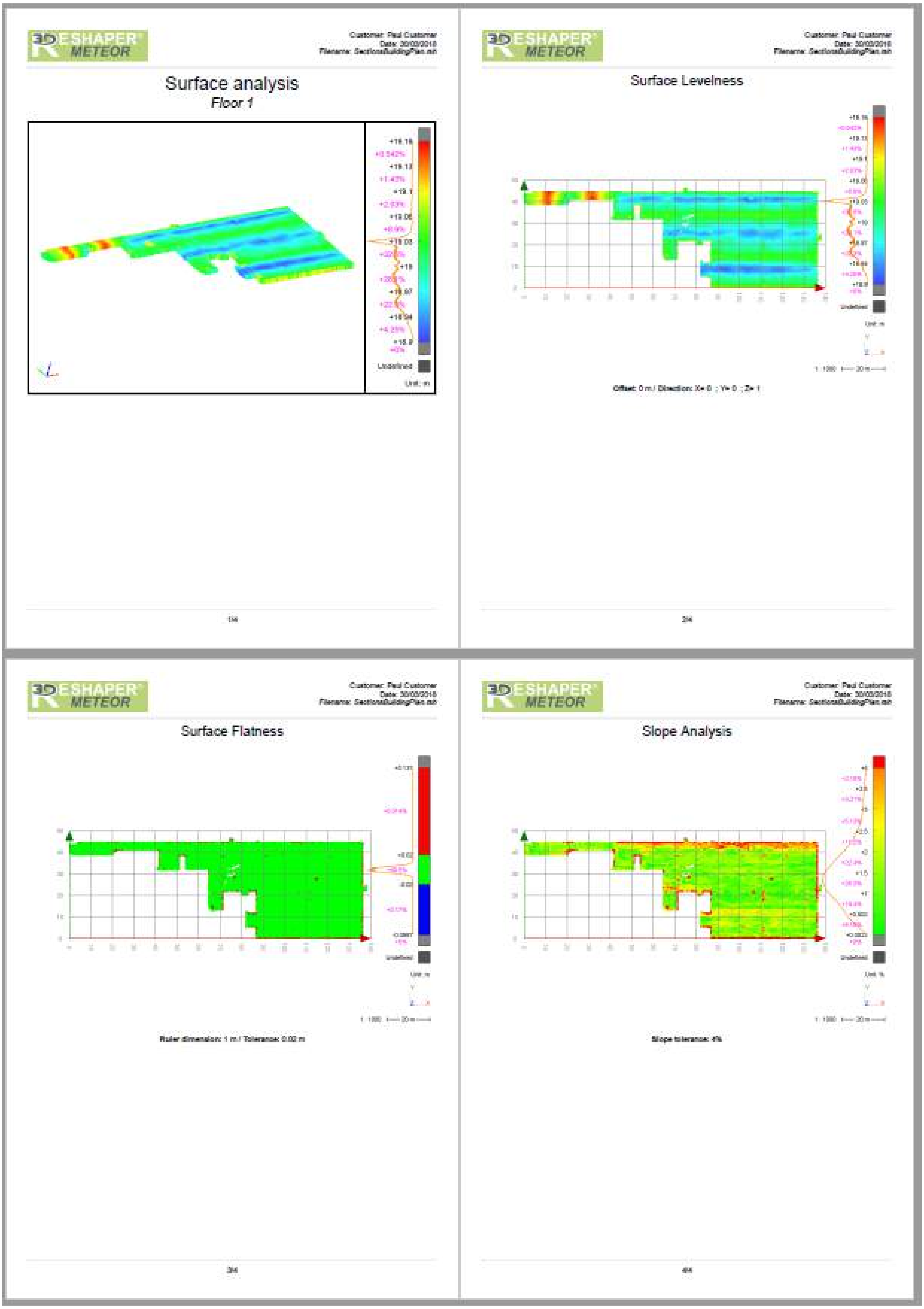
Make a report

Now, you can create a pdf report with these analysis. Launch File \ Report Editor. 4 chapters have been added to the report:

- cover
- surface Flatness
- surface Levelness
- slope Analysis

Click **New Chapter** to create an empty one called Title page. Remove the chapter called Cover. Drag and drop the chapters so as to sort out your report as following: Title page; Surface levelness; Surface Flatness; Slope Analysis.

1. Choose a portrait **orientation** inside the **layout panel.** Apply header and footer for all the pages.
2. Add your company logo in the header top left cell: import your logo using the plus icon in front of **Environnment data** (**data panel**). Then, insert it in the top left cell.
3. In the header top right cell: add the current date by dragging and dropping from the **data panel** an automatic value. Repeat the same workflow for the customer name and the filename. Align these texts to the right using the text toolbar.
4. In the footer: keep one cell, add the **current page** and the **total page**. Align center these texts using the text toolbar.
5. In the body area: keep one cell, transform the cell into a **text area** and write the title report. Format it using an appropriate style.
6. Add a cell below and insert the levelness viewset in **3D Mode**.
7. In other chapters: add a cell containing the **chapter title**, format the texts and remove the cells containing table if you haven't created labels previously.
8. Modify 2D scenes so as to set the **scale** to 1:1000. Choose to display the grid.
9. Optionnaly, export the chapters as new templates using **...** icon.
10. Finally, click on **To PDF** to generate the report.


Customer: Paul Customer
Date: 30/03/2018
Filename: SectionsBuildingPlan.rsh
Surface analysis
Floor 1
Undefined
Unit: m
Surface Levelness
Offset: 0 m / Direction: X= 0 ; Y= 0 ; Z= 1
Surface Flatness
Ruler dimension: 1 m / Tolerance: 0.02 m
Slope Analysis
Slope tolerance: 4%

Report

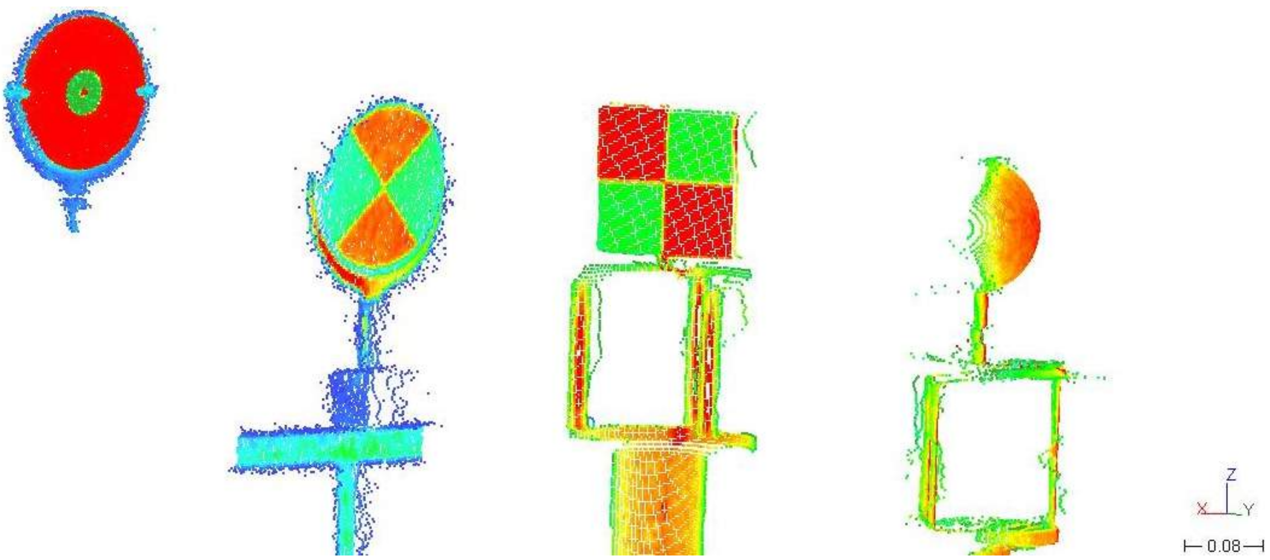
10.3 Surveying extraction

- Exercise: Automatic extraction of the center of surveying targets

10.3.1 Exercise: Automatic extraction of the center of surveying targets

Open the file “EnterPoints.rsh”.

This file contains the point cloud of a round black and white target. The point cloud is displayed in the “Inspection” representation in order to show the difference of intensity. Go to Surveying \ Extract targets. The upper ribbon to click on a point appears, with the option **Surveying Target** already checked. Select the option **With checkered pattern** in the type list. Click a point on the target closed to the center. The center will be automatically computed and a new point created.



Automatic extraction of target center

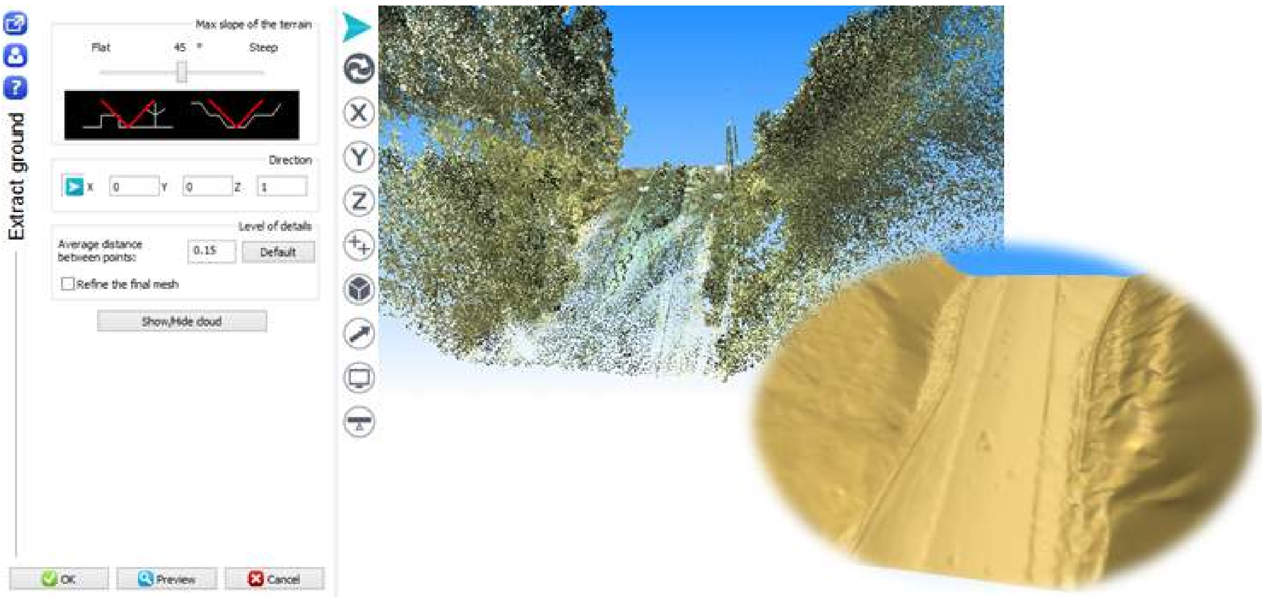
The target extraction works on black and white targets or blue and white targets only when the “Inspection” representative mode is active, because the center is computed by inspecting the difference of intensity in the point cloud. Extraction of spherical targets works on the geometry of the cloud.

10.4 Surveying modeling

- Exercise: Automatic creation of a Digital Terrain Model from a point cloud
- Exercise: Create the simple model of a building

10.4.1 Exercise: Automatic creation of a Digital Terrain Model from a point cloud

Open a new project in 3DReshaper. Import the file "ExtractGround.nsd". You can directly drag and drop it in the 3D scene, or import it with the function File \ Import cloud(s). It is the scan of a road going through a forest, so there is a large amount of vegetation to remove in order to keep only points on the ground. Select the point cloud and launch the command Surveying \ Ground Extractor. Set the maximum slope that can be seen in the terrain to **45°**. Choose **Z axis** as the direction for the computation. The average distance between points will give you the level of details in the final mesh. This value is computed by default, but you can set it to **0.15**. Check the option **Refine the final mesh** in order to improve the result.
Click **Preview** to get the resulting mesh. You can show or hide the cloud in order to better see the result. Through the **Advanced parameters** it is also possible to extract the points that are on the ground, or the points that are not on the ground. Click **OK** to create the mesh.



Automatic creation of a terrain model

10.4.2 Exercise: Create the simple model of a building

The aim of this exercise is to see how to create the model of a building without complex meshing, but only by extracting planes and joining them together automatically.

Open the file “AlignTargets.rsh”. This file contains two point clouds of a building, also used for the alignment exercises.

Show only the cloud **Fusion Cloud**. You can let the representation mode as **Inspection** or turn it to **Smooth** to not see the colors over the point cloud.
Select the cloud and launch the command Surveying\Building Extractor.

3DReshaper does pre-processing to compute the best extraction tolerance for the planes over the whole point cloud. You can see this parameter at the top of the dialog box, and you can modify it with the cursor if needed.
Make sure the option **Automatic contour** is checked, and then you can begin the extraction.

Click on a wall on the building. You will see a thick contour showing which plane has been found. Press Enter to validate this contour, a planar mesh is now created. Click on the next wall and press Enter to validate the contour. A second planar mesh is created and the two faces are automatically connected to each other.

Continue like this to model the entire building.

Create a model based on planar faces

10.5 Volume and cubature

In 3DReshaper, three kinds of volumes can be measured:

- Volumes of a closed object
- Volumes over and under a certain level of liquid
- Volumes of embankment and excavation between two meshes

Different configurations can be used:

- If you select one closed mesh before launching the command Measure/Volume, you will be able to compute either the volume of the object, or the volumes over and under a water level.
- If you select several closed meshes, a window appears showing the volume of each mesh. Labels are also created.
- If you select an open mesh, you will not be able to compute its volume, but only the volumes over and under a water level.
- If you select two open meshes, you will be able to compute the volumes of embankment and excavation between them.

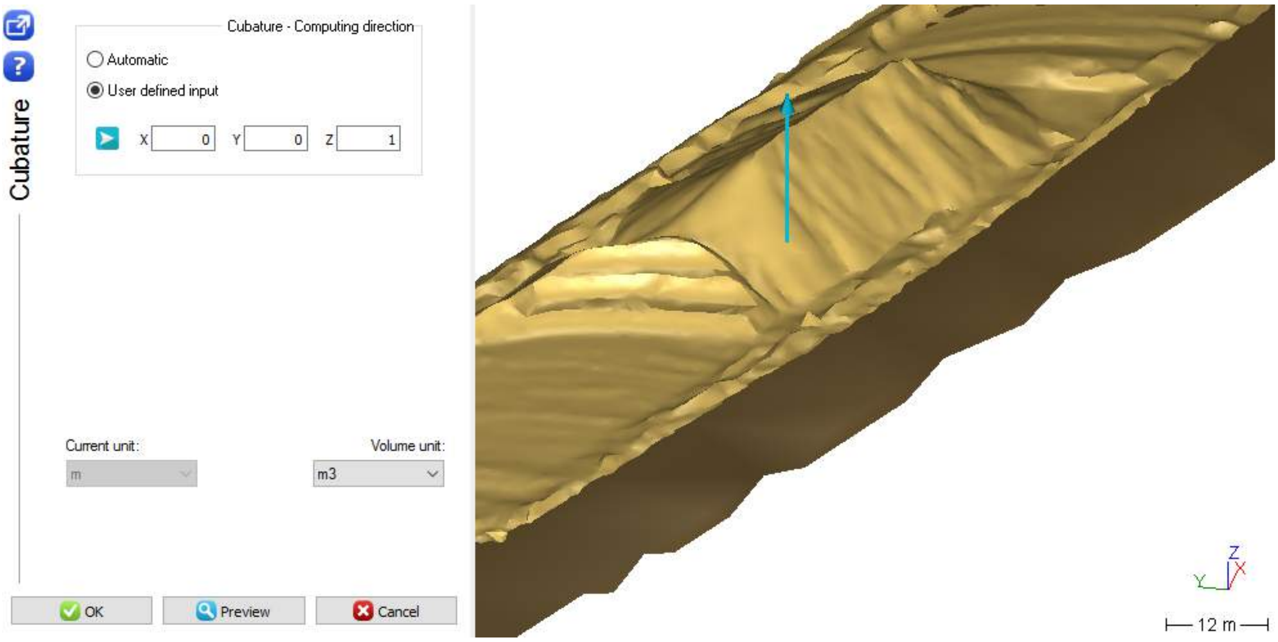
- Exercise: Compute cubature between two open meshes
- Exercise: Measure cubature with a level of water
- Exercise: Measure the volume of a closed mesh
- Exercise: Create a stockpile project

10.5.1 Exercise: Compute cubature between two open meshes

With the command Measure \ Volume \ Cubature, it is possible to compute embankment and excavation between two open meshes according to a given axis.
The two meshes must have only one hole corresponding to the external border. Small holes on their surface will lead to inaccurate or wrong results.

Open the file “Cubature.rsh”. It contains the mesh of a stockpile and the mesh of a reference ground.

Select both and go to Measure \ Volume \ Cubature.
Choose the direction for the cubature computation. Here, choose **Z** direction as the reference plane. We consider it represents the ground. You can also try the automatic direction. In this case, 3DReshaper tries to find an appropriate direction to see both entire surfaces.



Compute cubature between two open meshes

When you click **Preview**, a window opens, showing the volume of excavation, the volume of embankment, and the difference between them.

```
Volume output

The volumes are approximated because some parts of the surface are outside of the free borders.
Volume on red ball side: -> above Reference Ground and below Stockpile: = 13 268,391 m³
Volume on green ball side: -> above Stockpile and below Reference Ground: = 6,188 m³


Difference of the two volumes: 13 262,204 m³
If the reference surface is Stockpile, Excavation volume of 13 268,391 m³ and embankment volume
of 6,188 m³
If the reference surface is Reference Ground, Excavation volume of 6,188 m³ and embankment
volume of 13 268,391 m³
```

> Note that you can choose the unit for the results.

When you close the window, two labels are created to indicate the volumes between the two meshes:

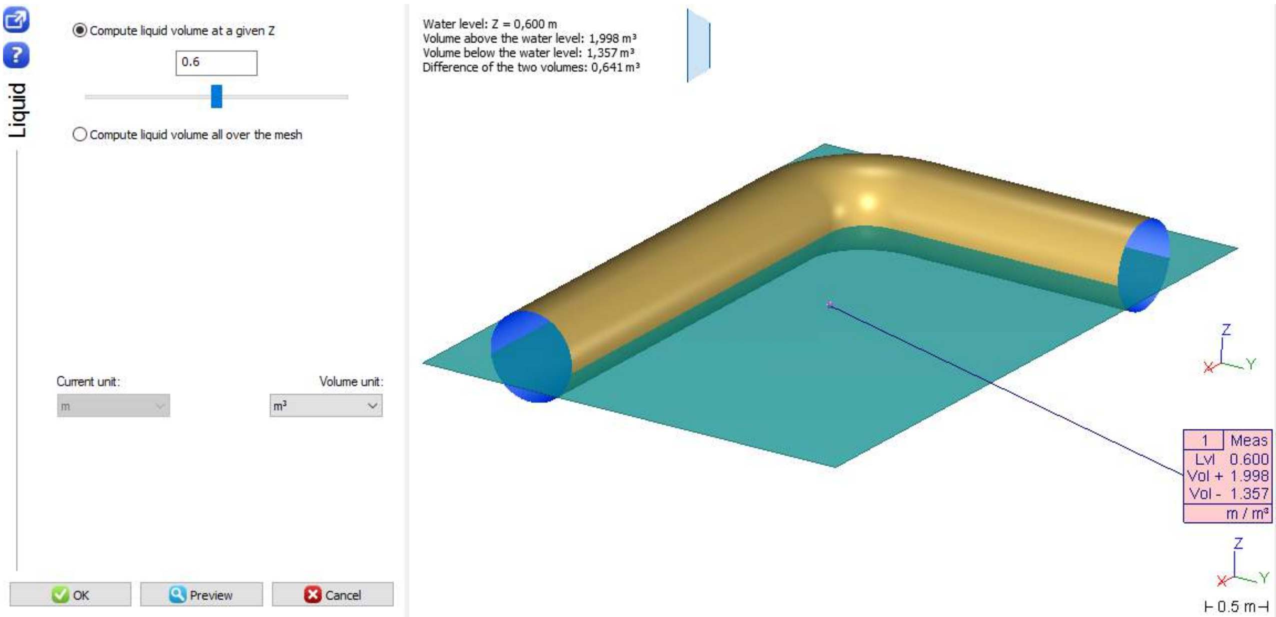
- One showing the volume above Reference Ground and below Stockpile.
- One showing the volume above Stockpile and below Reference Ground.

> You can check the online exercises to test the difference between the cubature computation with a rough mesh and with a refined mesh of this stockpile.

10.5.2 Exercise: Measure cubature with a level of water

Open the file “VolumeClosed.rsh”.

Select the bent pipe and go to Measure \ Volume \ Liquid. This command can be used to compute volumes over and under a level of liquid inside a closed or an open mesh.
You can see that a plane representing the water level is displayed in the scene. If you validate the result, it will be inserted in the Geometric Group. This plane will, of course, always be horizontal regarding the local coordinate system.
The plane is automatically placed in the middle of the object. You can adjust its height with the slider or enter a value manually. The maximum value of the slider corresponds to the highest point of the mesh; the minimum value corresponds to the lowest point of the mesh.
Put the height on **0.60** for example and click **Preview**. A window opens showing the level at which the volume was measured and the volumes computed. A label is also created in the scene indicating that information. Click **OK** to validate computation and keep the label.



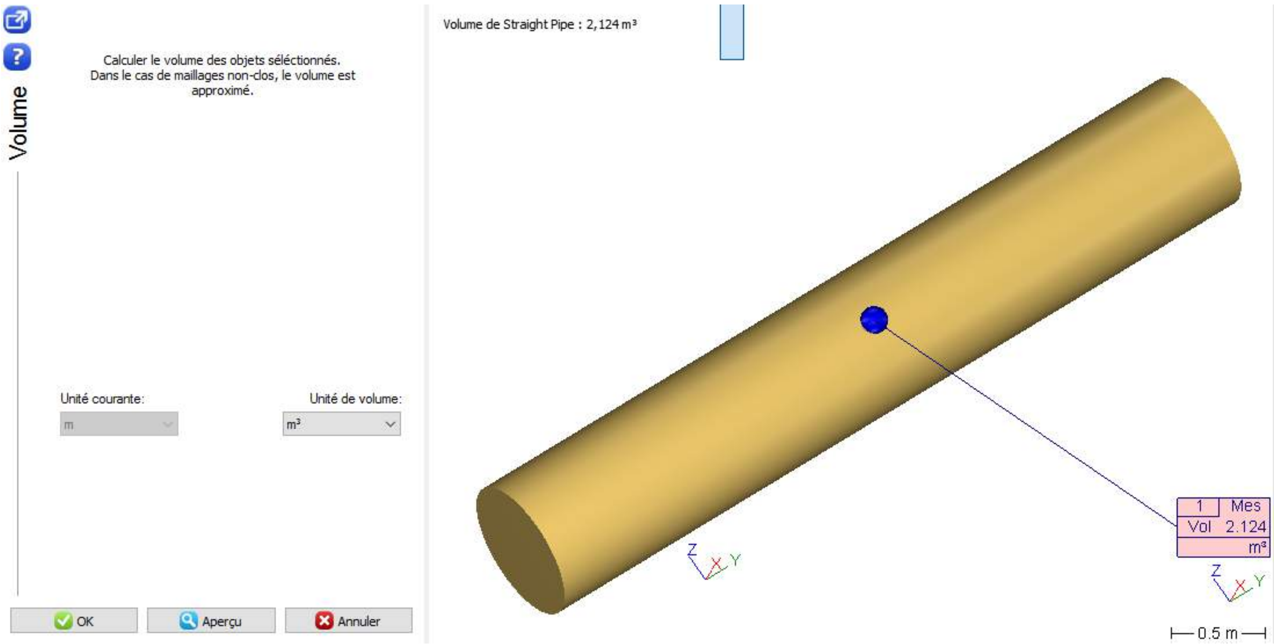
Compute cubature with a level of water

You can try the command with the closed Bent Pipe and with the open Bent Pipe. In this case, if the volumes are computed for the same level of liquid, it makes no difference whether the mesh is closed or not because the pipe is closed vertically.
With the option **Compute liquid volume all over the mesh**, it is also possible to compute volumes between several levels of liquid, distributed along the Z axis with a regular step.

10.5.3 Exercise: Measure the volume of a closed mesh

Open the file “VolumeClosed.rsh”. It contains the meshes of two sections of a pipe.

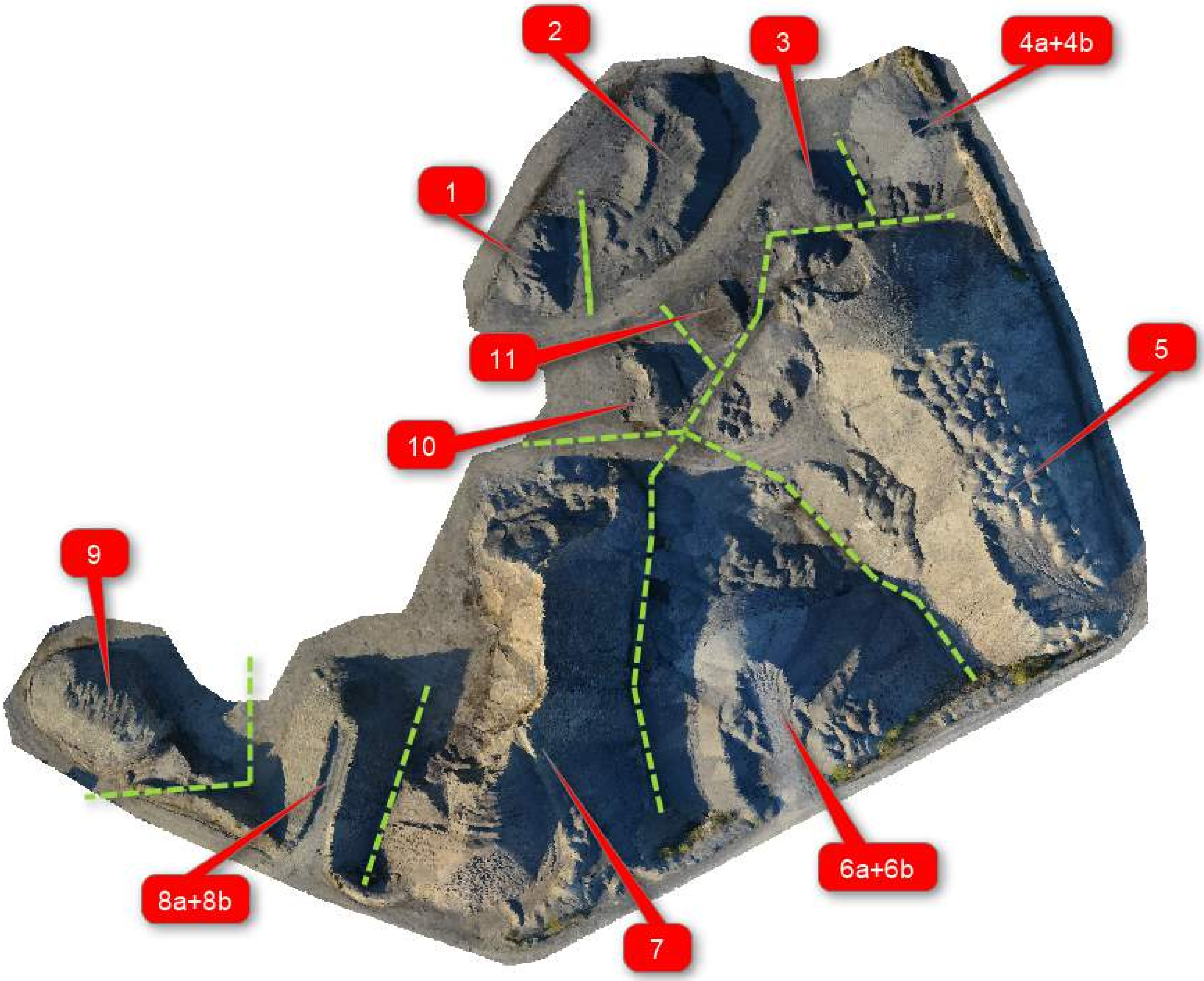
Select one mesh and go to Surveying/ Volume / Volume. Click **Volume** and **Preview** to compute the volume of the object.
A window opens, showing the volume of the object. A label containing the volume information is created in the scene and attached to the center of the object.



Volume of a closed mesh

See also, in the chapter “Measure Volumes” in the “Measurement, Inspection and Reporting” section.

10.5.4 Exercise: Create a stockpile project



Open the file

Open the file “Stockpile.rsh”. It contains a cloud with several gravel stockpiles. This file is going to be use through this whole exercise. Select **Stockpile cloud** and launch the command Surveying \ Create / Edit Stockpile Project.

Tip & Trick

To help you draw the contours, vertices have been added to this sample. If necessary, you can display them (**Help cloud**).

Create stockpiles

Stockpiles n°1, 2 and 3

Click on **New Stockpile** and define main characteristics of stockpile n°1:

- material name: 10001 (for instance a material ID),
- material nature: gravel, and
- grain-size: 4/6.

Note that you are free to modify or to add your own fields. You can also modify the stockpile color by clicking on the colored square. Click **OK**. If necessary, you will be allowed to modify later all stockpile definition items thanks to shortcuts within the command.

Then, you can draw its contour (refer to the stockpile map, to the help cloud and to fig.1). Close and validate the contour with **ENTER**.

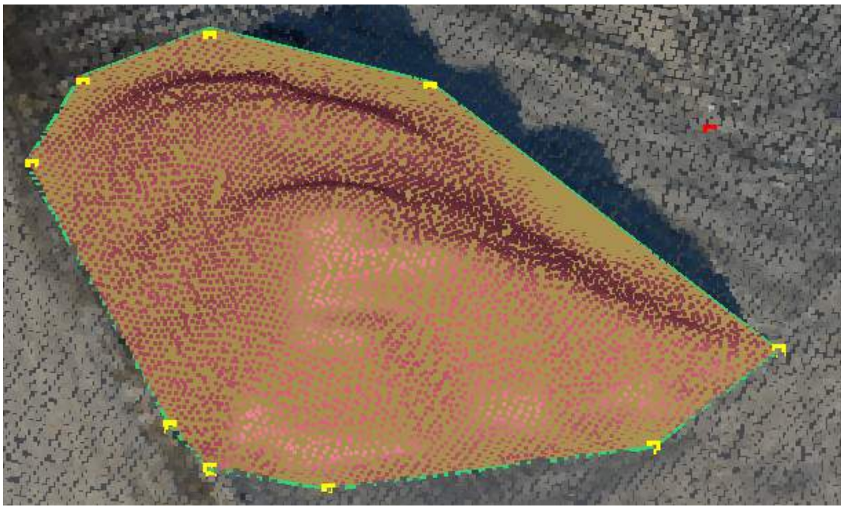


Figure 1: stockpile n°1

Select the **Best plane from contour** method to mesh the initial ground below the stockpile. If you have clicked exactly on the yellow points, you will be able to check the Cut Volume. Click **Preview.** You can use the bulbs to display/hide elements (this can help to find out the best computation method for the ground). Here, you don't have to apply a spike elimination. Click **Reset** to start again the stockpile contour, if necessary.

Click **OK, Next** and repeat the same workflow for stockpiles n°2 and 3, using the parameters in the chart below.

Stockpile name	Material name	Material nature	Grain-size	Method	Spike elimination	Cut Volume
1	10001	gravel	4/6	Best plane from contour	None	422m^3
2	10001	gravel	4/6	Best plane from contour	None	2365m^3
3	10002	gravel	4/6	From contour points	None	293m^3

> **Note**
>
> Some stockpiles are along embankments. Consequently, you have to find out the initial ground. Sometimes, you can reconstruct the embankment shape by splitting the stockpile (n°4, 6, 8). Sometimes you can't (n°5 and 7) but you have to choose the less inaccurate method.

Stockpile n°4

For this stockpile 4, you have to proceed in 2 steps by computing first the main part 4a above the ground, and then create another stockpile 4b corresponding to the part above the embankment. In both cases you should start drawing the contour by clicking points corresponding to the border between parts, that is to say points corresponding to the low and invisible embankment edge. Choose the **From contour points** method to compute the 2 parts. Note the gap size between both parts depends on the cloud resolution.

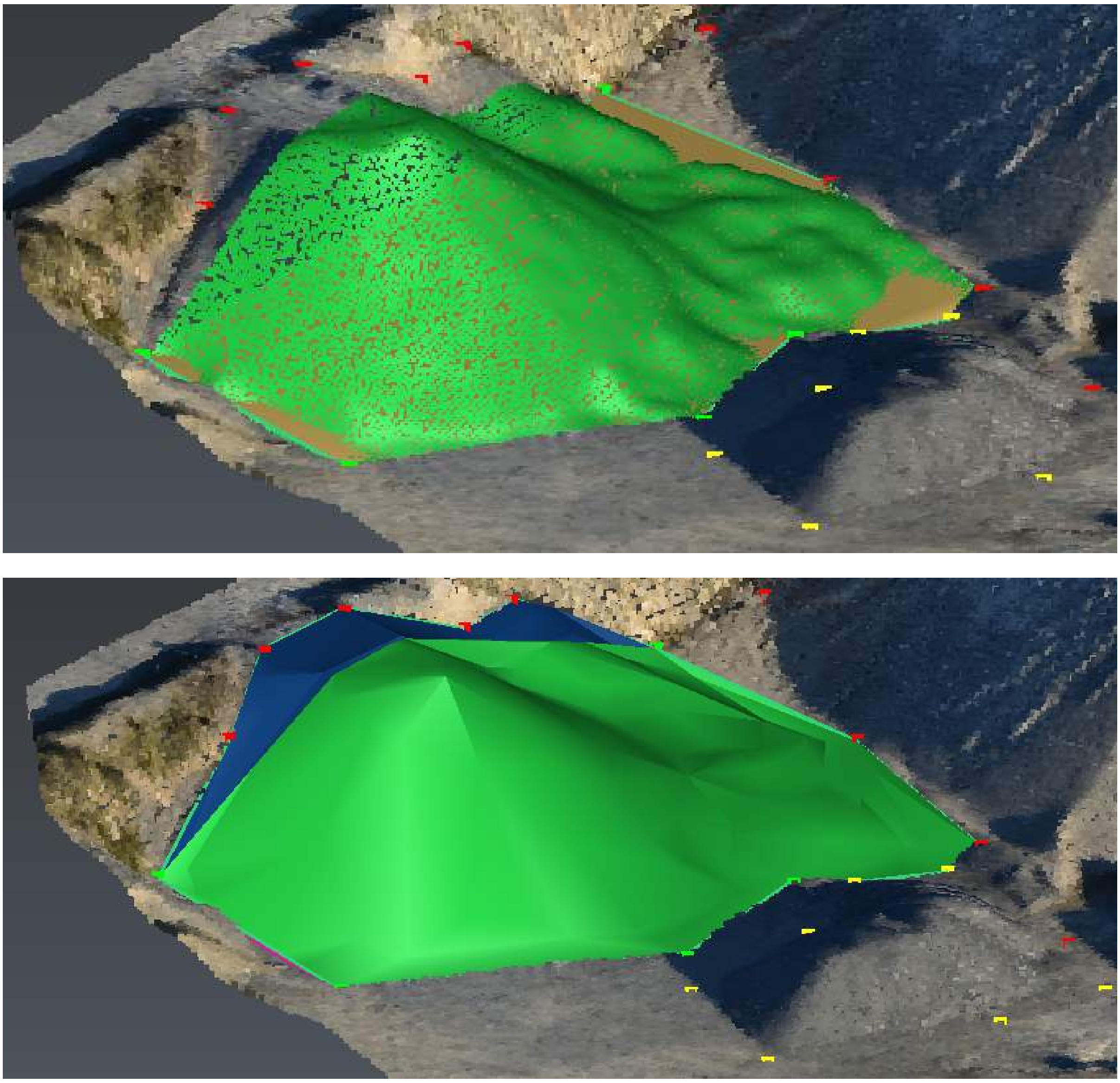


Figure 2: stockpile n°4 (a+b)

Stockpile name	Material name	Material nature	Grain-size	Method	Spike elimination	Cut Volume
4a	10003	gravel	4/10	From contour points	None	$1878m^3$
4b	10003	gravel	4/10	From contour points	None	$187m^3$

Tip & Trick

In fact, you can proceed in the same way for stockpile n°5 and 7. However, you have to reconstruct the low embankment edge before. For this exercise, show both help lines.

Other stockpiles

Repeat the same workflow for each stockpile. You can also go directly to the next section.

Stockpile name	Material name	Material nature	Grain-size	Method	Spike elimination	Cut Volume
5	20001	stone	100 /200	Horizontal plane from lowest point	Medium	$91367m^3$
6a	20002	stone	0/150	Horizontal plane from lowest point	Medium	$22821.5m^3$ *
6b	20002	stone	0/150	From contour points	Strong	$1390m^3$ *
7	20003	stone	0/80	Horizontal plane from lowest point	Medium	$50295.5m^3$
8a	10004	gravel	0/31.5	Horizontal plane from lowest point	Medium	$5775m^3$
8b	10004	gravel	0/31.5	From contour points	Medium	$314m^3$
9	10005	gravel	4/10	Horizontal plane from lowest point	None	$5083m^3$
10	00001	sand	0/2	Best plane from contour	None	$304m^3$
11	00002	sand	0/4	Best plane from contour	None	$79m^3$

*****: refer to the section Clean noisy points.

Clean noisy points

In some cases (for instance n°6), you may have to remove noisy points. Click **Cancel** to exit the command. Select the **Main cloud** and launch Cloud \ Clean / Separate Cloud(s).

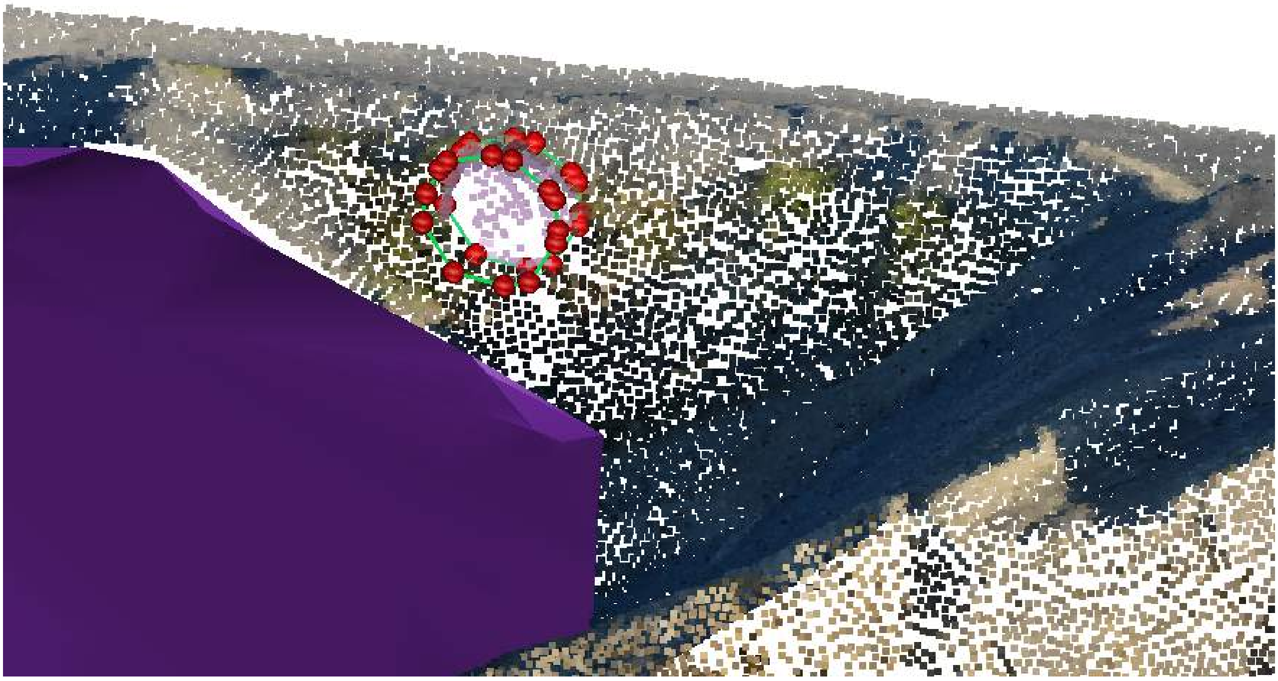
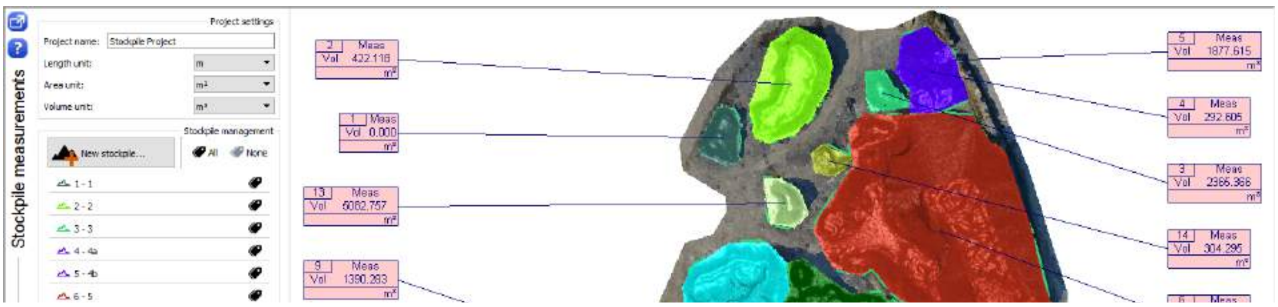


Figure 3: remove noisy points

> **Note**
>
> Noisy points must be removed before the stockpile computation. If the stockpile project has been created, you have to modify the **Main cloud** inside the project folder. Otherwise, you can simply modify the input cloud.

Add Labels

Finally, add labels for all stockpiles and define a single color for the stockpiles which have been divided into two parts. Choose a new color or copy/paste for instance the HTML color code.



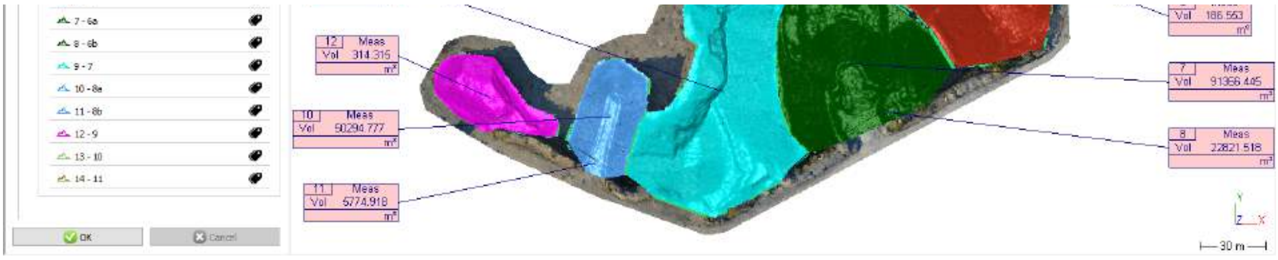
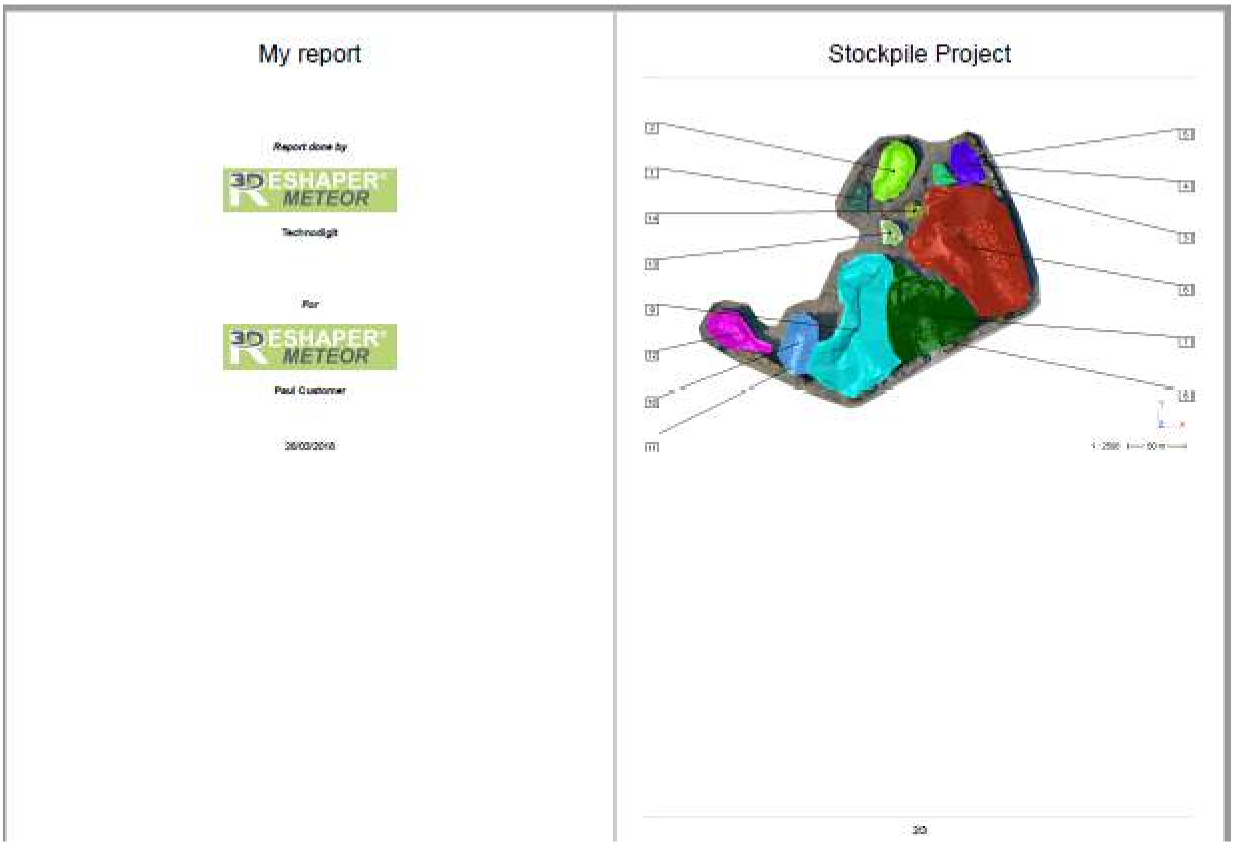


Figure 4: colored map with labels

Create a report

Launch File \ Report Editor. Actually, you must see the default template for stockpile projects. At this step, you must have two chapters called Cover and Stockpile project data. You are going to customize the second chapter:

- keep one item in header and footer . Drag and drop **Project name** from the group **DATA** to the header . Drag and drop **Current page** and **Total page** from the group **DATA** to the footer. Align these texts to the center with the tool bar,
- remove the text item which is inside the body,
- select the viewset and set the scale to 1:2500 in the group **OPTIONS**,
- select the chart and filter the columns: show the numbers, the colors, the stockpile names, the material names, the material natures, the grain-sizes and the cut volumes,
- align to center the table and reduce the number of decimals (click somewhere in the template view to display the **LAYOUT** panel), and
- click **To PDF** to create your report.



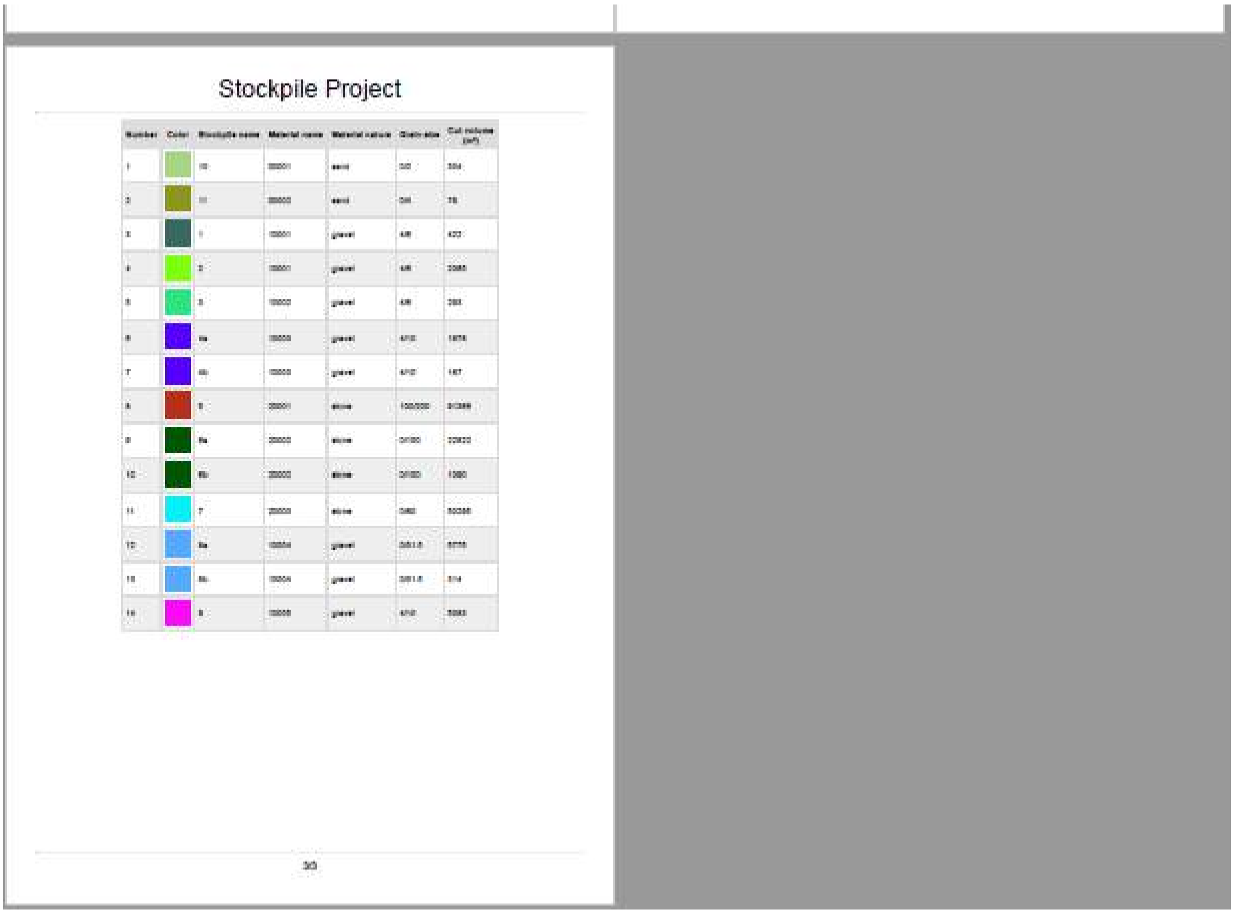


Figure 4: customized template

11 Tank

11.1 Introduction

The Tank module is dedicated to tank analysis. It is aimed to above ground tanks, with vertical cylinder design, single shell, with or without roof.

This module brings a full workflow designed to follow main requests from API 653. Nevertheless, it provides generic tools (3D inspection, color maps, sections, etc.) so that it can be versatile enough and enable the inspection using different standards or considering other types of tanks (horizontal cylindrical shapes, for example).

This module can be added to the standard configuration of 3DReshaper.

The following pages will guide you through the workflow for a complete analysis of a tank. Note that you always need to create a mesh of the tank before being able to use the tools from the Tank Module. You can use the tools from the Base license to clean the scans and create an accurate mesh.

The tank used in these exercises results from an inside scan of a tank. This module can, obviously, also be used on tanks scanned only from the outside.

- Define the project
    - Start the project
    - Compute the best cylinder
    - Separate the shell
- 3D Inspection
    - Compute the inspection
    - Unroll the color map
- 2D Inspection
    - Roundness
    - Verticality
- Settlements
    - Differential settlements
    - Localized settlements
- Create a Tank Inspection Report

11.2 Define the project

Exercise: Define the project and prepare the data for the inspections

- Start the project
- Compute the best cylinder
- Separate the shell

11.2.1 Start the project

Open the file “TankInspection.rsh”.

This file contains the mesh of a tank. It will be used through all the following exercises.

Select the mesh **Tank mesh** and launch Tank \ Create / Edit project. First, give a name to the project or take the standard name **Tank project**.

You can then define a specific orientation point for the tank using the Entering point procedure.

Place it on the top of the manhole.

The elevation marker will automatically be on the lowest point. You can place it on the bottom of the mesh.

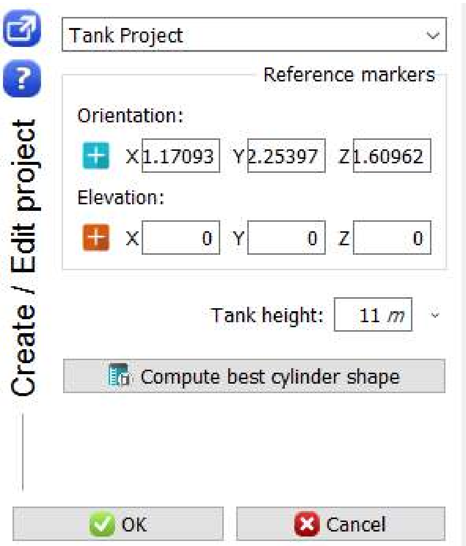
Both markers can be used as references in several functions from the Tank module.

Finally, enter the theoretical height of the tank: **11 m**.

Click **OK** to create the project. A new folder has been created in the tree, containing several objects:

- **Tank Info**: this element cannot be displayed in the 3D scene. It is a property sheet containing definitions and results from the project.
- **Tank mesh**: the initial mesh chosen for the project.
- **Tank Project Orientation Reference** and **Tank Project Elevation Reference**: the two markers.

As the project is defined, all next computed results will automatically be inserted inside this same folder. On top of this, you will not need to select your mesh again before using the next functions. As soon as a tank project is defined, commands will automatically use the mesh of the tank as input. Nevertheless, it is always possible to launch a command using a selected mesh or object like other commands in 3DReshaper.



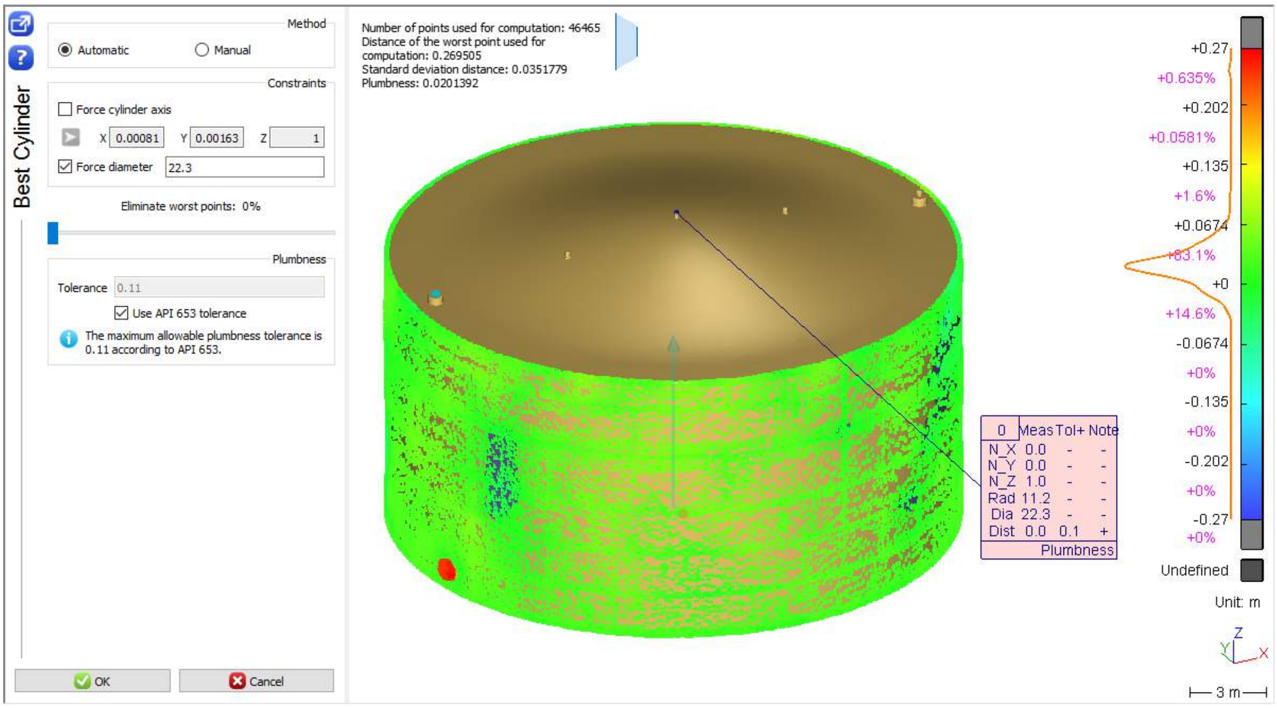
Creation of a Tank project

11.2.2 Compute the best cylinder

Without selecting anything, click on Tank \ Best Cylinder. A cylinder best fitting the tank will be computed.

If the **Automatic** method does not give a result good enough, you could switch to the **Manual** method and click several seed points for the extraction of the best cylinder.

It is possible to give in some constraints for this cylinder, if the nominal dimensions of the tank are known. To force the diameter of the cylinder for example, check the option “Force diameter” and enter **22.3 m**.



Compute a best cylinder on the tank

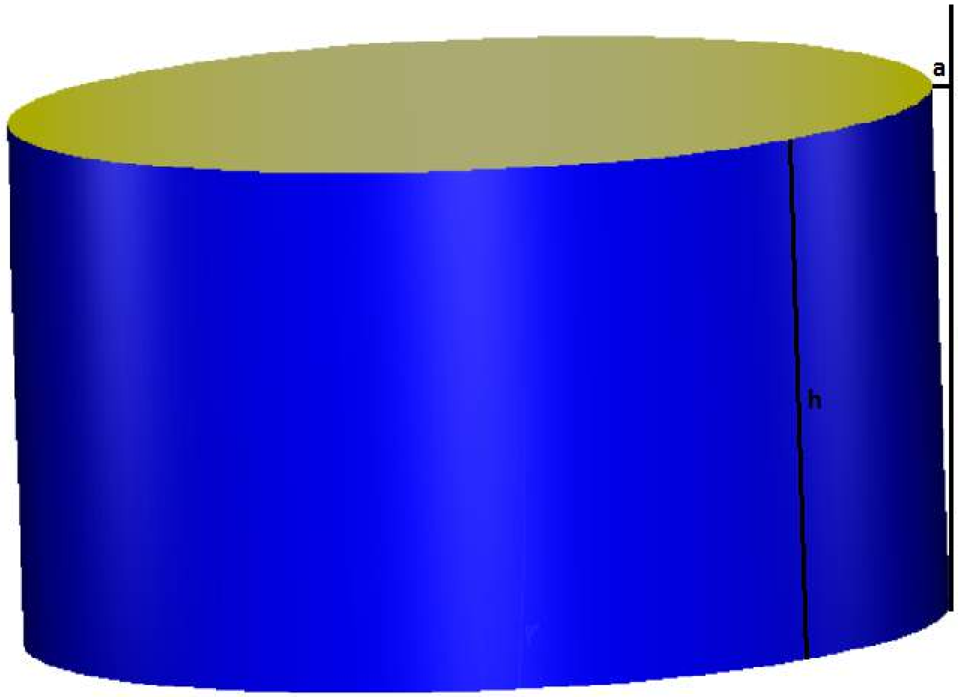
It is possible to eliminate the worst points, for example if the mesh is not perfectly clean. Here we can keep the computation with all points.

The color scale at the right shows how well the Tank matches the computed cylinder.

A small window on the left also gives some additional information on the computation.

In the created label, we have the coordinates of the main axis of the cylinder. This function also gives us the plumbness value of the tank, as described in the API 653.

The plumbness value is the distance between the tank axis and a vertical axis, at the theoretical height of the tank. The API 653 defines that plumbness shall not exceed 1% of the total tank height, with a maximum of 5 inches.



Definition of the plumbness (a) of a tank

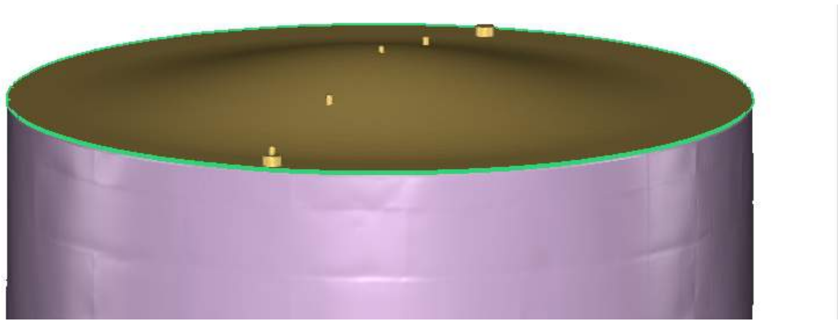
> ⚠ The use of API 653 tolerance is not mandatory. If you want to enter another tolerance, uncheck the corresponding checkbox and enter your new value.

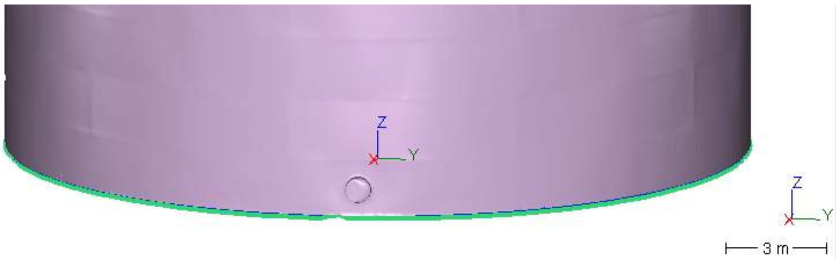
Validate the result with **OK**. We now have a new folder **Best Cylinder** in the Tank Project. It contains the best cylinder, the central axis of the cylinder and the label giving the plumbness value.

11.2.3 Separate the shell

For some inspections, it is useful to split the tank in several parts in order to consider only the shell or only the bottom plate, for example.

Show only the initial mesh and go to Tank \ Separate Shell. The computation is done automatically; it can take a few seconds. We now have a new folder **Separate shell** containing two polylines representing the top contour and the bottom contour of the tank, as well as the three separated parts of the tank (Shell, Top, and Bottom).





The tank is separated in 3 meshes: the shell, the top and the bottom
The project is now fully ready to start the complete analysis of the tank.

11.3 3D Inspection

The tank can be compared in 3D to the best cylinder computed previously in order to detect potential deformations on its surface. This inspection is usually relevant only on the shell of the tank.

Exercise: Run a 3D inspection on the tank shell

- Compute the inspection
- Unroll the color map

11.3.1 Compute the inspection

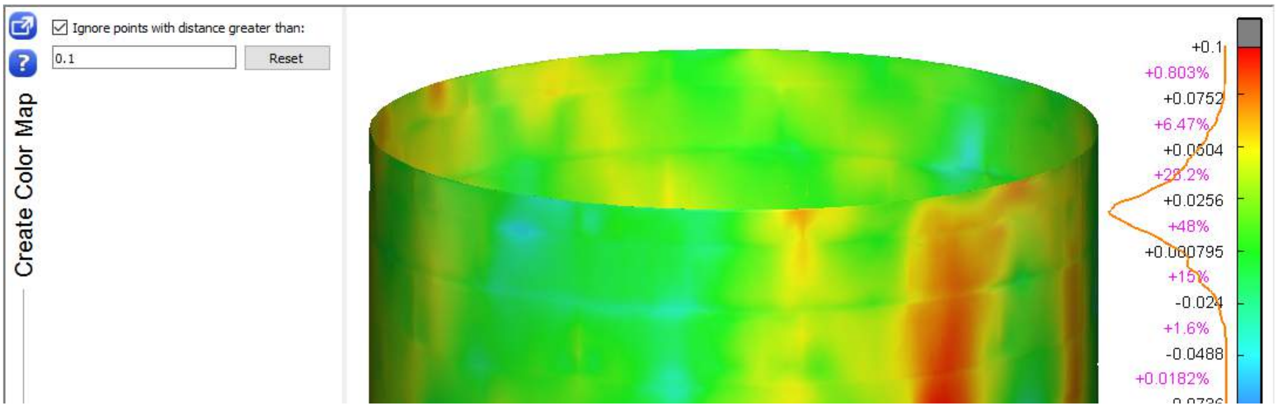
Show only the mesh called “Tank mesh Shell”. Select it and launch Tank\Create Color Map.

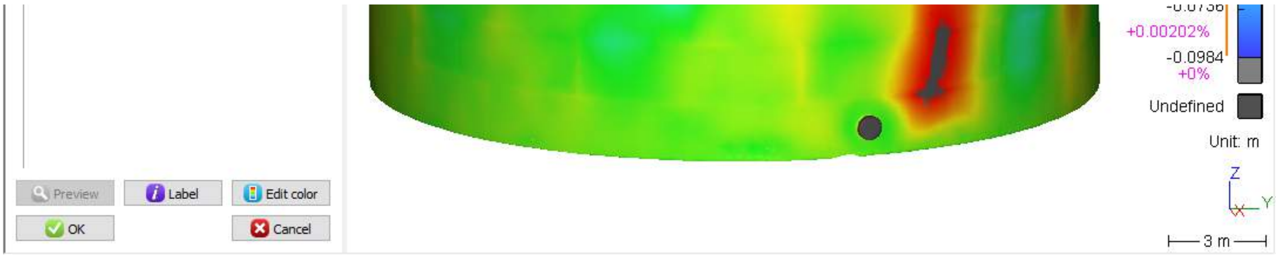
This command computes the deviations from the shell compared to the best cylinder. The deviations are displayed thanks to colors applied to the mesh.

The only input parameter required is a distance used to remove too distant points from the computation. By default, this value is initialized with the distance of the worst point used to compute the best cylinder, so that the result will be visually the same as the one that you had when computing the best cylinder.

Change this distance to **0.10 m** in order to ignore the manhole and click **Preview** to replay the computation. We now see better where could be some problematic areas on the shell.

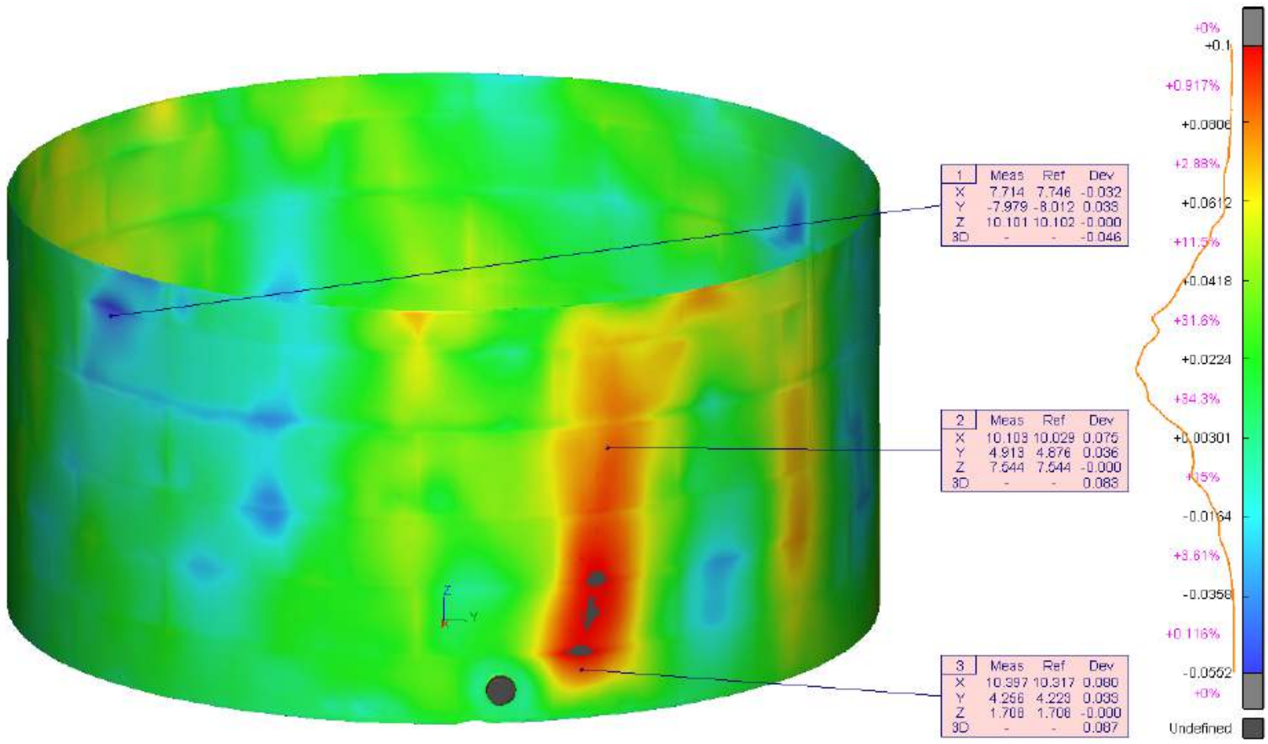
The color scale shows the maximum and the minimum distances as well as the distribution of the deviations between these two extreme values.





Compute the 3D deviations on the tank shell

Click on the "Label" button to create some labels on specific points on the shell. The labels give the 3D coordinates of the measured point and of the reference point as well as the deviations in the three axis and in 3D.



Create labels on specific points

Click **OK** to validate the results. The colored mesh and the labels are created in a new folder “3D Inspection”.

11.3.2 Unroll the color map

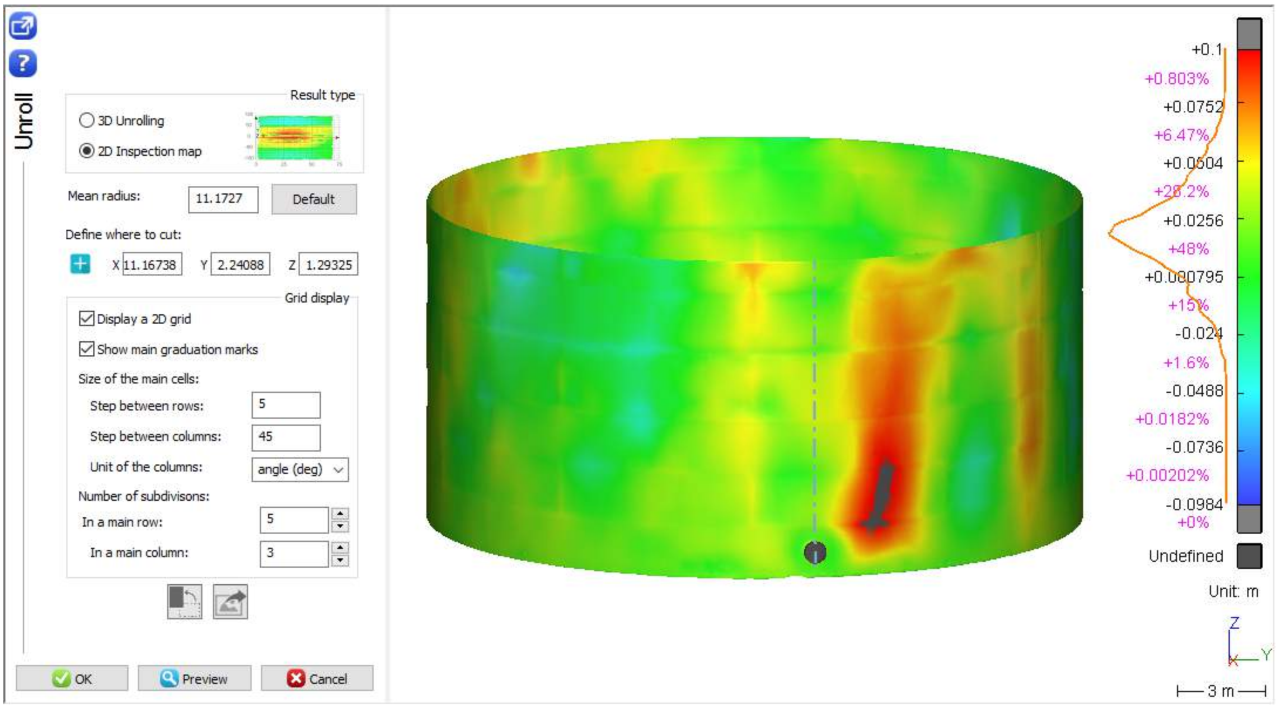
This colored mesh can now be unrolled to show a 2D inspection map of the shell.

Show only the mesh **Compare Tank mesh Best Cylinder / Tank mesh Shell** and launch Tank\Unroll.

The shell can be unrolled in 3D to keep the shape of the bumps and hollows in the surface (from weld seams, for example), or unrolled in 2D to simply get a flat color map. Choose the option **2D Inspection map** .

The shell will be unrolled along its center axis, using a fixed radius. The command automatically takes the center axis and the radius of the best cylinder computed previously.

The shell will be cut in its height, where the Orientation Marker has been defined during the project creation.

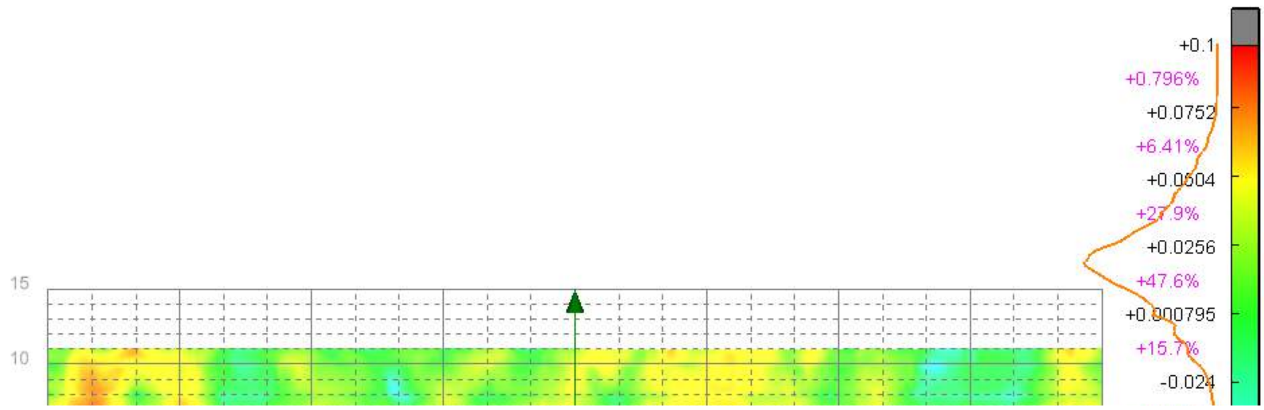


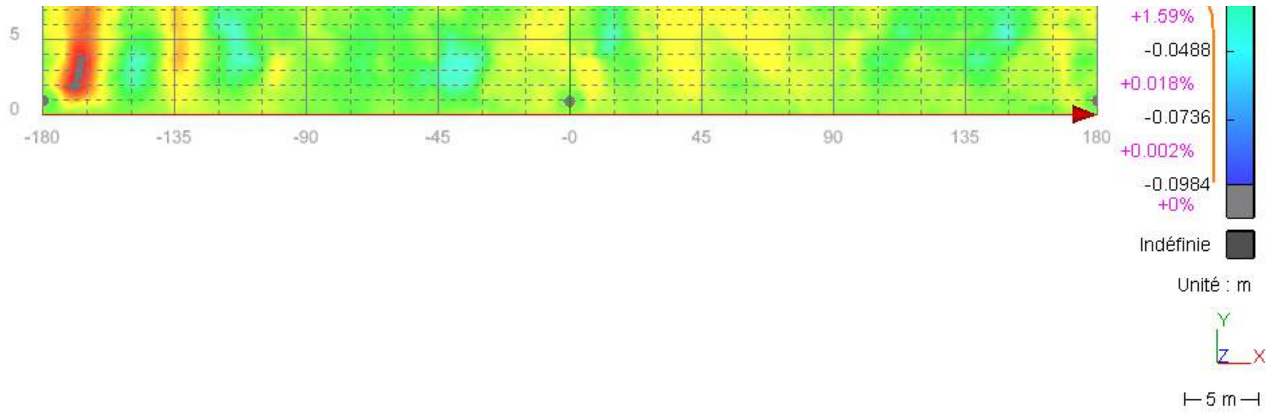
Parameters of the best cylinder are used to unroll the shell of the tank

Check the options to display a 2D grid over the result and show the main graduation marks. Set rows every **5 m** and columns every **45°**. Choose **5** subdivisions for the rows and **3** subdivisions for the columns.

Click **Preview** to compute the result.

> ⚠ The value of the column can be given as an angle or as a curvilinear distance around the tank.





Unrolled inspection map of the tank shell

After validating with **OK**, a folder “Unroll” is added to the tree, containing the unrolled shell and the 2D grid. It is possible to show or hide the 2D grid, like any other object.

11.4 2D Inspection

As we have seen previously, the shell of the tank can be fully inspected in 3D. It is also possible to inspect it with the help of horizontal or vertical sections in order to check the roundness and the verticality of the tank.

Exercise: Check the roundness and the verticality of the tank thanks to profiles

- Roundness
- Verticality

11.4.1 Roundness

Show only the mesh “Tank mesh Shell”. Select it and launch the function Tank\Roundness.

The purpose is to compute sections on the shell as well as on the theoretical shape (the best cylinder) at different heights, and to compare them.

First choose the direction of the sections. They can be created perpendicularly to the axis of the best cylinder computed previously, or can be created perfectly horizontally. Choose the first option “ **Perpendicular to the best cylinder axis**”.

You can choose to use tolerances for the radii according to API 653 or use your own tolerance. Here, keep the API tolerances.

Then, define where to create the sections. You can create sections at a regular step all over the tank, or only between given heights. You could also give a list of specific heights where to create sections, or choose the option “Click point(s)” to visually click on the tank where a section is needed.

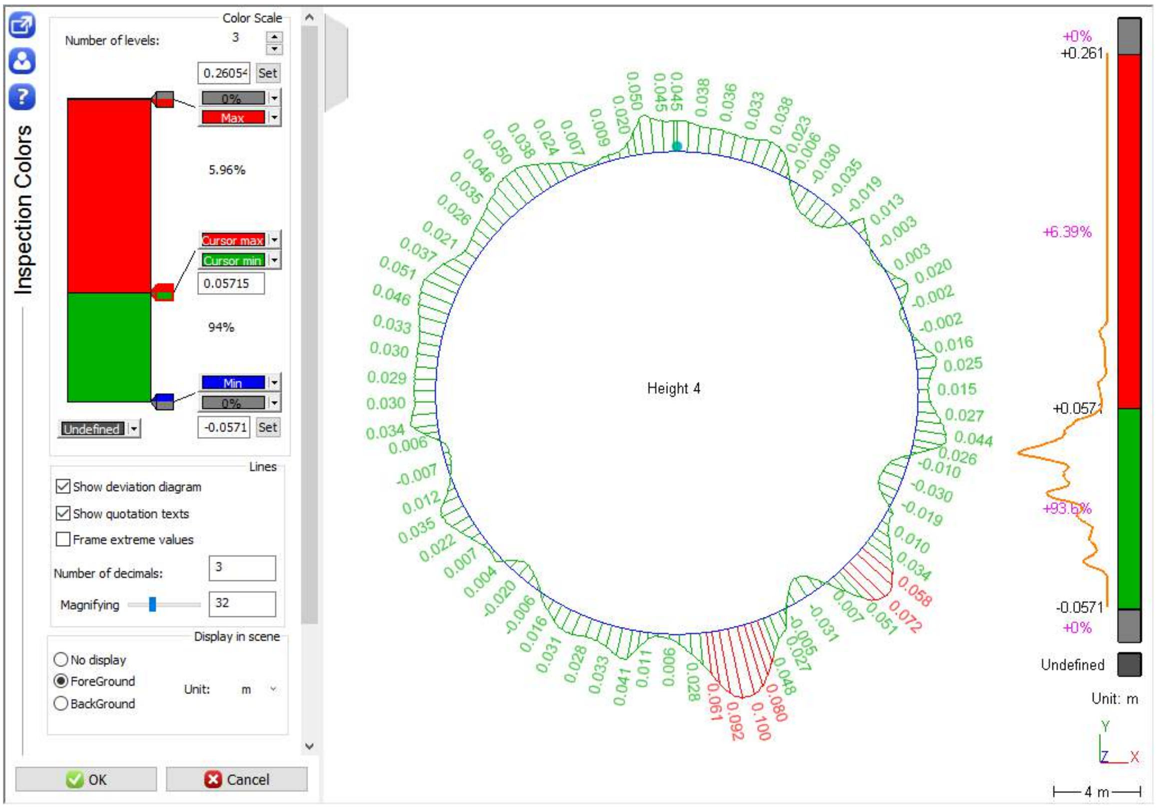
Choose the options “With a regular step” and “All over” and give a step of **1 m**. Click **Preview** to display the resulting sections.

You can now manage the display of the sections to visualize them easily. In **Display Sections** choose the option “In a grid” to display the sections in 2D in a grid. You can then zoom on one and pan the scene to go from one to another.

You can also choose the option “Section by section” to visualize only one section at a time. You then have other buttons in the dialog box to switch from one section to the next one.

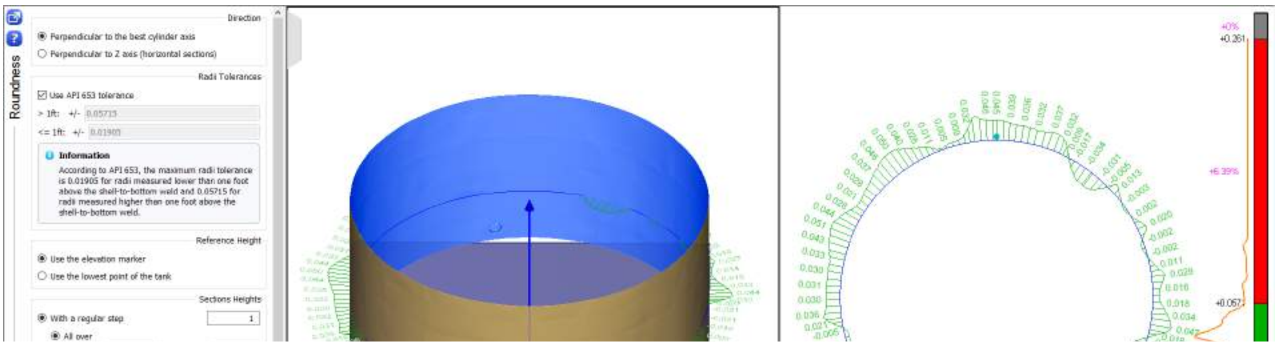
Choose to show only the section at height 4 (4 m) and click **Edit Color**. Here, it is possible to adapt the color scale if needed, and also to magnify the deviations in order to see even the smallest ones. Set the cursor on 32 for example.

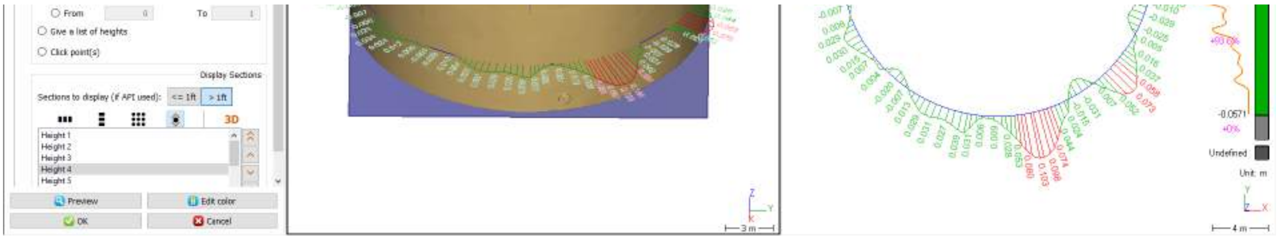
Then click **OK** to validate and come back to the previous command.



Magnify the deviations in order to see even the smallest ones

> At any time, it is possible to check the option “3D" to visualize the sections in 3D on the tank.





Display the section in 2D and in 3D on the tank

Click **OK** to validate the results. A new folder called “Roundness” is created in the tree. It contains all the sections per height (on the tank, on the cylinder and the result of the comparison) and the corresponding points on the center axis.

11.4.2 Verticality

A similar method can be used to check the verticality of the tank, by creating vertical sections on the tank.

Show only the mesh “Tank mesh Shell”. Select it and launch the function Tank\Verticality.

First choose the direction of the sections. They can be created parallel to the axis of the best cylinder computed previously, or can be created perfectly vertically. Choose the second option “**Parallel to Z axis (vertical sections)**”.

> ⚠ You can choose to use the tolerance defined by the API 653 or give in your own tolerances.

Then define where to create the sections. You can create sections at a regular step all around the tank, or only between given angles. You could also give a list of specific angles where to create sections, or choose the option **Click point(s)** to visually click on the tank where a section is needed.

Choose the options **Regular angle between sections** and **All over** and give a step of **10°**. Click **Preview** to display the resulting sections.

The first section is done on the Orientation Marker defined when creating the project.

You can now manage the display of the sections to visualize them easily. In **Display sections** choose the option **In a grid** to display the sections in 2D in a grid. You can then zoom on one and pan the scene to go from one to another.

You can also choose the option **Section by section** to visualize only one section at a time. You then have other buttons in the dialog box to switch from one section to the next one.

Click **Edit Color** to modify the color scale if needed. It is also possible to magnify the deviations in order to see even the smallest ones. Set the cursor on 32 for example.

Then click **OK** to validate and come back to the previous command.

Magnify the deviations on vertical sections

At any time it is possible to check the option **3D** to visualize the sections in 3D on the tank.

Click **OK** to validate the results. A new folder called "*Verticality*" is created in the tree. It contains all the sections per angle (on the tank, on the cylinder and the result of the comparison).

11.5 Settlements

We have seen previously how to inspect the shell of the tank. In the Tank Module, functions also let you measure the settlement of the tank, using different methods.

Exercise: Measure different settlements on the tank

- Differential settlements
- Localized settlements

11.5.1 Differential settlements

In this exercise, we will see how to measure differential settlement from a polyline using the method described by API 653. The polyline must represent the edge between the shell and the bottom plate of the tank.

It is also possible to compute differential settlement using existing geometric points. This method is used when measuring fix points on the outside surface of the tank. See the help files of the command for more details.

Show only the “**Tank mesh**”. Find the polyline “**Tank mesh Contour bottom**” in the “**Separate shell**” folder and launch Tank\Differential Settlements.

You can choose to use the tolerances defined in the API 653 by checking the very first option.

The Orientation Marker defined when creating the project is used by default as the start point. You can define a new point if needed.

Then set how many points are needed to measure the settlement. The default number of points (8 here) is the minimum number requested by the standard. Enter **50** for the number of points. The points are automatically created on the polyline in the 3D preview and the computation is automatically done.

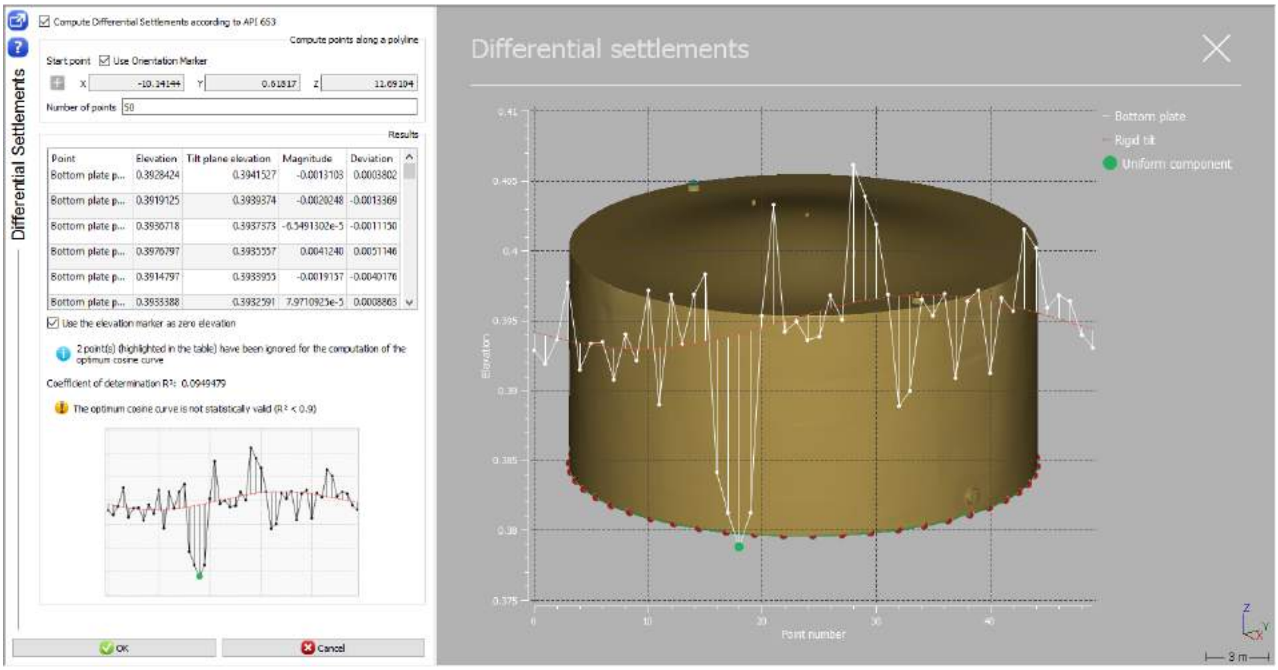
The results are displayed in the dialog box. A table is showing the results in each point:

- **Elevation**: the elevation of the point,
- **Tilt plane elevation**: the value in the best cosine curve at the index of the point,
- **Magnitude** of the differential settlement (Ui): the difference between the elevation and the tilt plane elevation of the point,
- **Deviation** (Si): Si = Ui - (1/2Ui-1 + 1/2Ui+1)

It is possible to use the Elevation Marker defined at the creation of the project as the reference elevation by checking the option “Use the elevation marker as zero elevation”. Check it.

Two points are removed from best cosine curve computation in order to improve the coefficient of determination R².

You can then click on the graph in the dialog box to enlarge it over the 3D scene and be able to display information on each points when moving the mouse cursor on them.



Results from the differential settlement

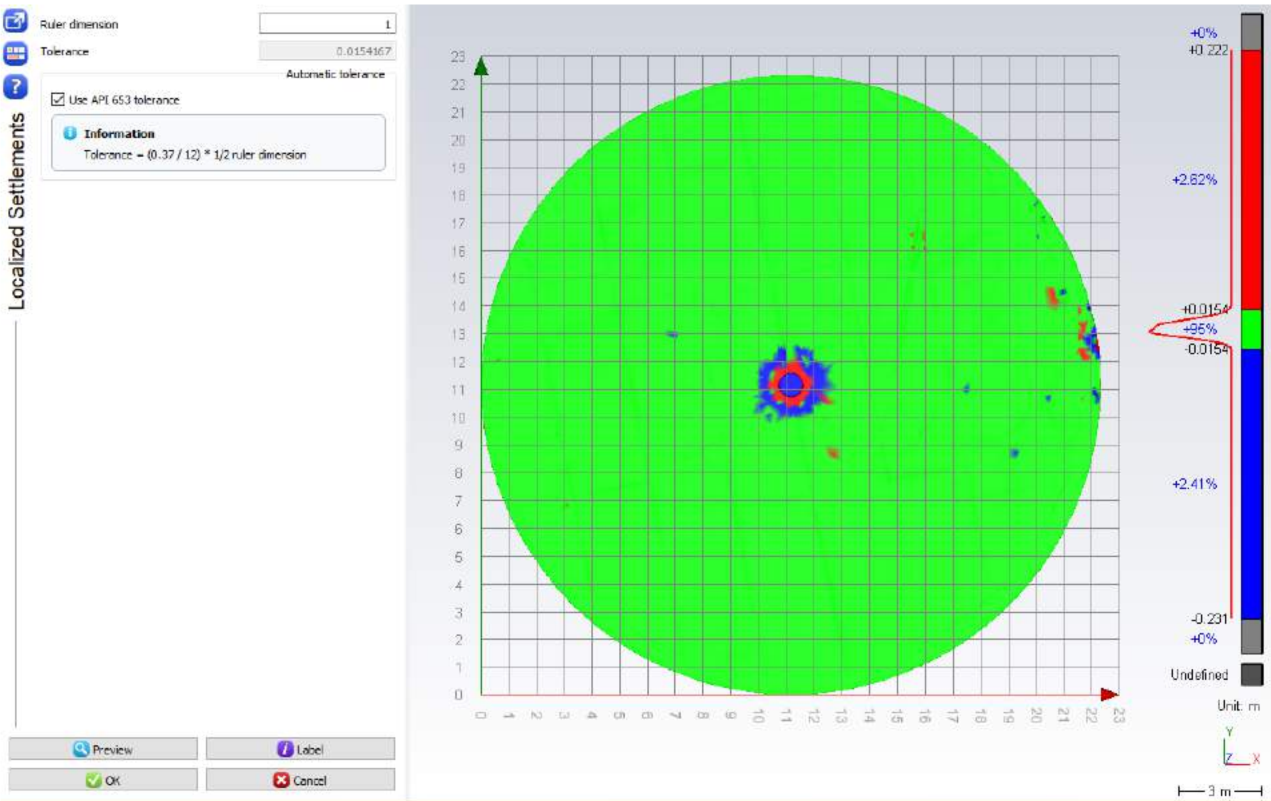
Click **OK** to validate the results. A new folder called "*Differential Settlements*" is created in the tree. It contains the 50 points used for the computation.

11.5.2 Localized settlements

If the tank has been scanned from the inside, it is also possible to measure the localized settlements on the tank bottom.

Show only the **Tank mesh Bottom** from the **Separate Shell folder**. Select it and launch Tank\Localized Settlements. This function will highlight bumps and hollows on the bottom of the tank, according to a ruler dimension and a given tolerance.

Set **1 m** for the ruler dimension and check the option **Use API 653 tolerance**. The tolerance is then computed automatically. Click on **Preview**. We can see that the bottom respects the API standards, apart from the center (where there is a lower part), the right side and some small areas.



Localized settlements on the bottom plate

It is possible to create labels on specific points to know the deviation in specific areas of the bottom.

Click **OK** to validate the results. A new folder called "*Localized Settlements*" is created in the tree. It contains a copy of the bottom plate mesh, colorized to show the deviations.

11.6 Create a Tank Inspection Report

> At the end of the workflow, all the results computed during the previous inspections can be included in one single PDF report or exported in a CSV file.

Without selecting anything, launch File \ Report Editor. A default report, with all available contents, is automatically created. First, define the main layout options in Layout panel. For instance, choose an A4 Portrait layout.

Then, you can remove some chapters and complete the report as you want. You can:

- Give a title to the project and to chapters.
- Enter the name of your customer and of your company.
- Insert the logo of both companies or any other image.
- Add and modify texts. You can drag and drop automatic fields from the Data panel.
- Add a global view as a 2D or a 3D scene.
- Edit all the views (or specific ones) inside a dataset. For example, you may increase the scale for a specific section.
- Apply another template to a chapter. Note you can save the current chapter as a new template for next tank inspections.
- Sort out the chapters.
- ...

Some results can also be exported in CSV files using the Data panel. For instance, the results computed during the differential settlements, the verticality and the roundness. Use the export icon

, or **To CSV**, to export a table into a .csv file.

Section Height	Avg. Radius	Min. Radius	Max. Radius	Min. Acc. Radius	Max. Acc. Radius
1 m	11.18152 m	11.14482 m	11.41054 m	11.09285 m	11.20715 m
2 m	11.17488 m	11.13229 m	11.24908 m	11.09285 m	11.20715 m
3 m	11.17255 m	11.1198 m	11.25257 m	11.09285 m	11.20715 m
4 m	11.17158 m	11.11522 m	11.25257 m	11.09285 m	11.20715 m
5 m	11.17334 m	11.12415 m	11.24446 m	11.09285 m	11.20715 m
6 m	11.17237 m	11.11878 m	11.24008 m	11.09285 m	11.20715 m
7 m	11.17379 m	11.12695 m	11.23605 m	11.09285 m	11.20715 m
8 m	11.1707 m	11.12261 m	11.22859 m	11.09285 m	11.20715 m
9 m	11.17338 m	11.11324 m	11.22596 m	11.09285 m	11.20715 m
10 m	11.17227 m	11.10987 m	11.21779 m	11.09285 m	11.20715 m
11 m	11.17378 m	11.12377 m	11.21957 m	11.09285 m	11.20715 m

Figure 1: csv export

Click **To PDF** to generate the report. This can take a few seconds especially in case of inserted 3D pdf.

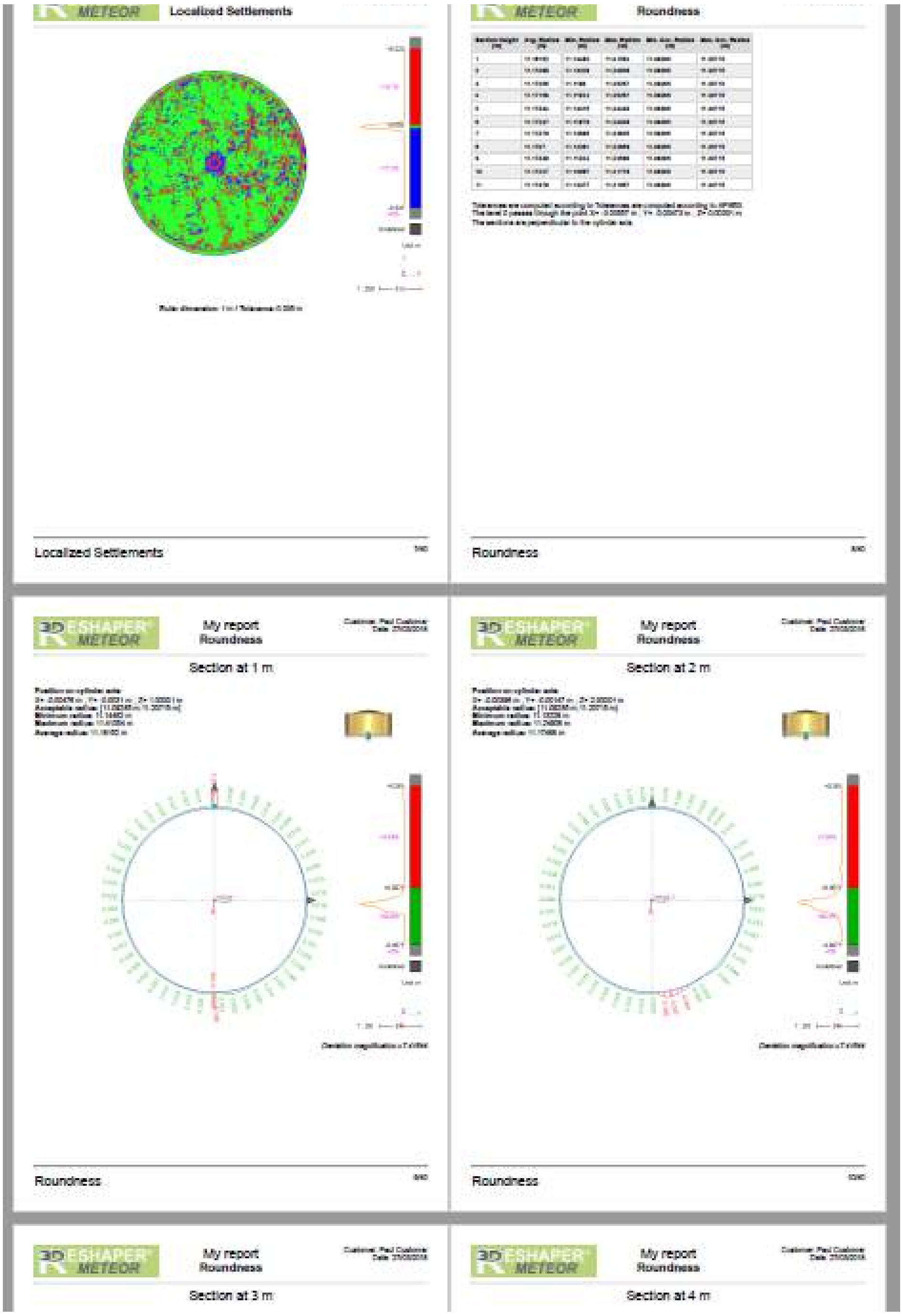


Figure 2: report example

12 Image

- Texture Mapping
  - Exercise: Texture a mesh with reference points
  - Exercise: Export textures from an RSH file
  - Exercise: Texture a mesh with camera parameters, adjust textures and export
  - Exercise: Texture a building mesh with an ortho-image
- Ortho-image
  - Exercise: create an ortho-image and import it in AutoCAD
  - Exercise: send an ortho-image to AutoCAD
- Virtual visits
  - Exercise: Create a video with a camera path
  - Exercise: Create a video with a camera scenario

12.1 Texture Mapping

In the software, you can map a picture on the corresponding 3D model. Depending on the data you have, there are two possibilities:

- Use couples of points (points on the 3D mesh and points on the picture).
- Use camera parameters (position, orientation, focal length, pixel size…).

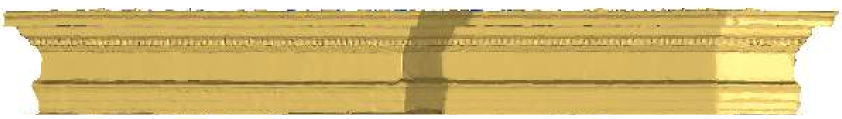
If you do not know all the camera parameters (for example, you know the position but not the orientation), you can click reference points and enter the information you have; they will constrain the mapping.

- Exercise: Texture a mesh with reference points
- Exercise: Export textures from an RSH file
- Exercise: Texture a mesh with camera parameters, adjust textures and export
- Exercise: Texture a building mesh with an ortho-image

12.1.1 Exercise: Texture a mesh with reference points

Open the file “TextureRefPoint.rsh”.

Select the mesh “MonumentBeforeTexture” and go to Image\Texture From Pictures\From Reference Points.



The monument to texture with reference points

The view will be automatically divided into two parts:

- On the left, the mesh to texture
- On the right, the texture to map

Click reference points

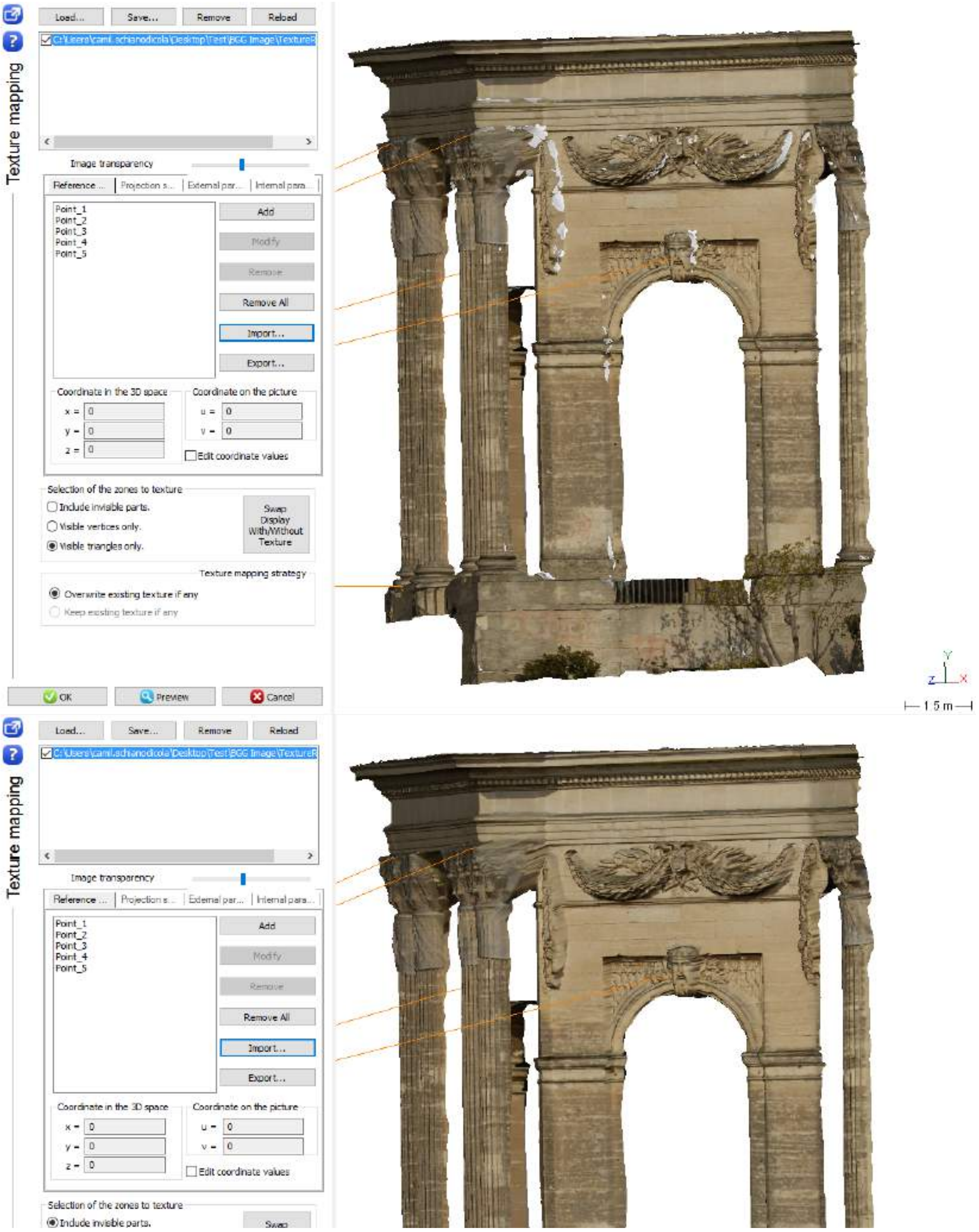
Classical picture

Click the **Load** button and then select the picture “TextureRefPoint.jpg” and click **Open** (don’t load camera parameters form incam file). The picture has been added to the texture list in the dialog box. Click now the **Add** button in order to create the first couple of points. Then, click on a point on the mesh and then the corresponding point on the picture (or vice versa). You can click angles or details in order to be more accurate. Once you have clicked the two points, you can click again the **Add** button in order to enter another couple. Click the **Modify** (resp. **Remove**) button in order to change (resp. remove) the selected couple in the list. If you want to empty the points list, click the button **Remove all**. All the couples should be sufficiently distant in order to map correctly the texture.
Once you have entered two couples of points, you can see the position of the camera in the 3D scene.
At any time, when you think that your reference point definition is correct, you can apply the texture on the model by clicking the **Preview** button. After the preview, you can continue to enter other couples of points, and you can use the button **Swap Display With/Without Texture** to make the selection easier.
If you have some difficulties to enter the points, you can empty your list of points and then press the **Import** button and select the file "TextureRefPoint.txt". You will have a sample with 5 couples of points. Note that in most of cases, 3-4 couples are sufficient. But if you have a distorted picture (for example taken with a “fish-eye” lens), you will have to enter more points (probably 10 couples).

In addition to the camera position, the software needs to know which triangles have to be textured according to its visibility from the point of view. The problem is that the definition of “visible” is not always clear because:

- Some triangles can be partially hidden by other surfaces.
- Sometimes the model is not correctly measured and you get some hidden triangles because of noisy parts.

In this example, select the option **Visible triangles only** and then click **Preview**. If you turn the 3D view a little bit, you will see some non-textured triangles because at least one of the three vertices is not visible from the camera point of view. Then select the option **Include invisible parts** and click **Preview** again. All triangles will be textured.



Once the result of your preview is correct, you can validate with the "OK" button.

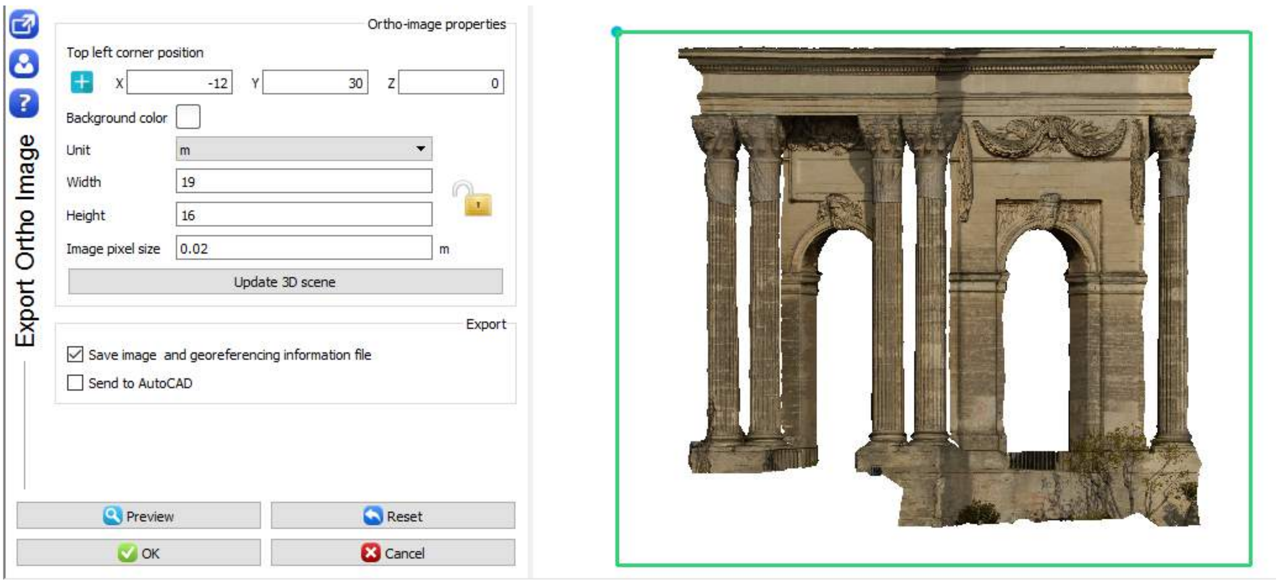
Picture with distortion

If you want to practice more, you can redo the exercise with the picture “TextureRefPoint-Distortion.jpg”. You will have to enter more couples of points in order to compute the lens distortion. You will have to click points all over the picture, including in the corners. You can import the file “TextureRefPoint-Distortion.txt” in order to have a sample.

> ⚠ You can texture a model with several pictures

Create an ortho-image

Once the mesh is textured, you can for example create an ortho-image. You just have to show your textured mesh, check that the orthographic mode is enabled (and not the perspective mode) and then set the view as you want (for example you can press the Z key in order to see along the Z axis), and go to Image\Create Ortho-Image.



Export an ortho-image

Enter (-12, 30, -6) for the top-left position. Set the background to white. Set the width to 19m, the height to 16m and the pixel size to 0.02m. Click **OK** and save the image. The created image has a resolution of 950x800 pixels.

A .txt file (at the same place than the picture and with the same name) is created to save georeferencing information (position of corners, pixel size, the view direction, etc.) so that the image can be easily imported in another software.

12.1.2 Exercise: Export textures from an RSH file

Open the file “TextureParam&CameraPath.rsh”.

Select the mesh “CliffTextured” and go to Image \ Texture From Pictures \ From reference points.

Select the picture “1032.jpg” in the texture list and then click the button **Save…** and export it as “TextureCamParam1032.jpg” in the samples directory. Repeat for the two others pictures.

12.1.3 Exercise: Texture a mesh with camera parameters, adjust textures and export

In this exercise, we will use camera parameters in order to do a very accurate texture mapping. There are two kinds of parameters:

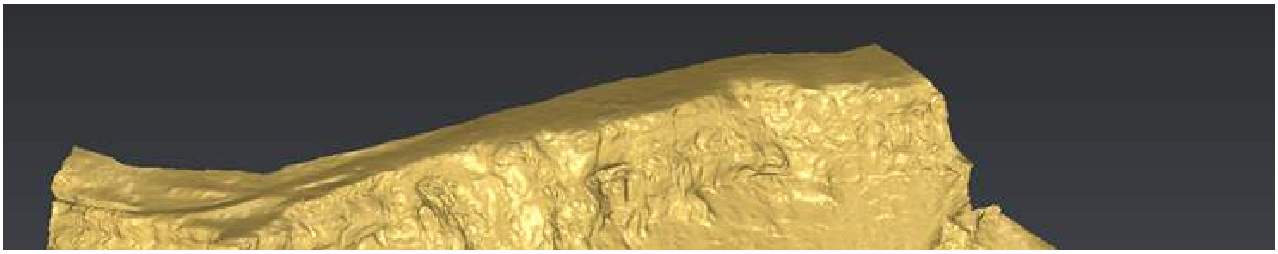
- External parameters (different for each picture)
- Internal parameters (different for each camera)

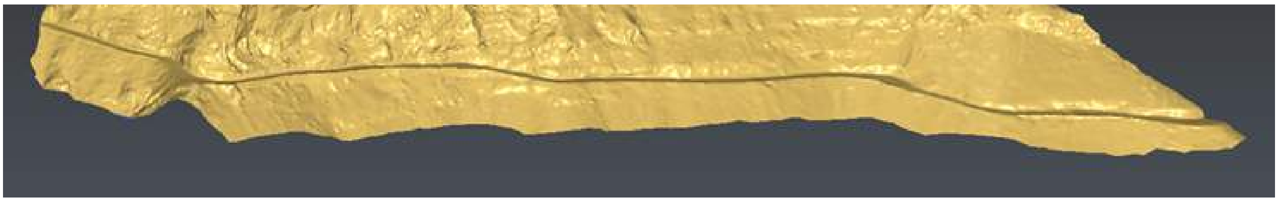
Open the file and launch the command

Open the file “TextureParam&CameraPath.rsh”

Show only the mesh “MeshToTexture”, select it and go to Image\Textures From Pictures\From Reference Points.

Click the button **Load** in order to open the textures from the previous exercice and select files “TextureCamParam1029.jpg”, “TextureCamParam1030.jpg” and “TextureCamParam1032.jpg” and click **OK** . If some .incam files exist with the same name in the same directory, you will be asked if you want to import automatically camera parameters. As there are .excam files with the same name in the same directory, you can see that the **External parameters** tab has been automatically filled.





The mesh to texture with camera parameters

Set projection type

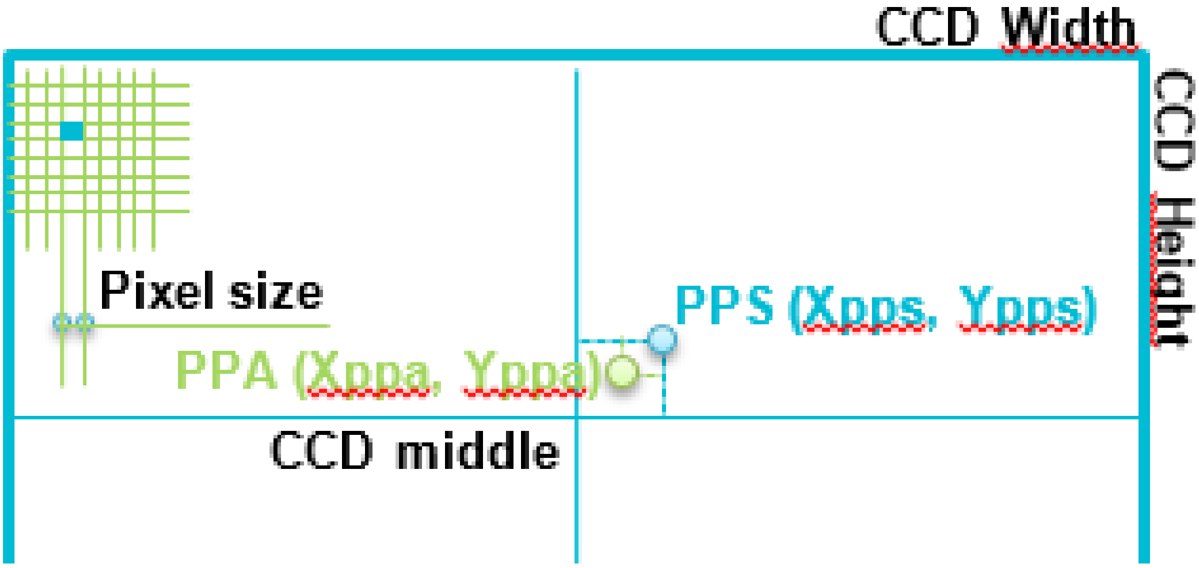
Go to the tab **projection settings** and select the type **Automatic** in the list in order to detect automatically the best projection type. You can also select **Perspective** as the picture has been taken with a classical camera (no distortion and not a panoramic picture).

Enter camera internal parameters

As the internal camera geometry is the same for all the images, we will first enter these parameters and we will save them inside a file in order to prevent repeating the input for each image. Select the first texture in the list, go to the **Internal parameters** tab and select the option **Make parameters editable and enter constraints.**
All the values must be expressed in the same unit as the 3D model. If your model is in meters and your internal camera geometry is defined in millimeters, you must multiply all the values by 0.001. So if your focal length is equal to 50mm, you must enter 50E-3.

Internal parameters are:

- The focal length (for perspective and fish-eye images)
- The CCD or the pixel size (for perspective and fish-eye images)
- The lens misalignment (for perspective images only):
    - Principal point of symmetry - PPS (offset from the CCD center)
    - Principal point of autocollimation - PPA (offset from the PPS)
- The radial distortion (for perspective images only)



Camera internal parameters

Open the file "TextureCamParamCalibrationFile.txt" in a text editor (like Notepad).

"TextureCamParamCalibrationFile.txt"

; H1 calibration File
; Date: 17.03.2009 - RHONE

; Focal length [mm]
C= 35.124

; CCD dimensions [mm]
X= 48.96
Y= 36.72

; Pixel size [um]
Ps= 9

; Principal Point [mm] PPA related to PPS (assumed to be Xpps=0 Ypps=0)
Xppa= -0.025
Yppa= -0.298

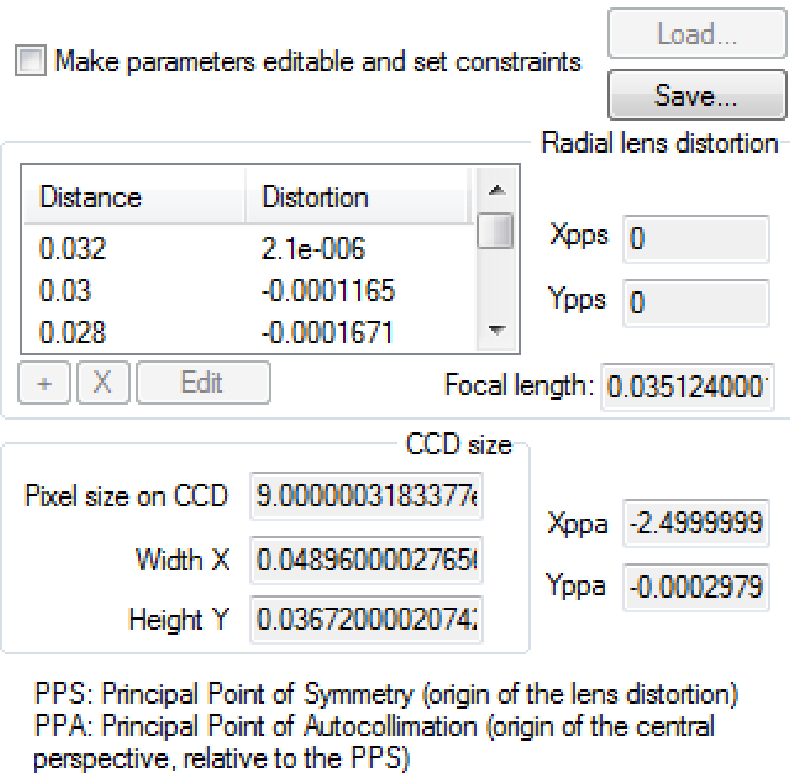
Xpps= 0.000
Ypps= 0.000

; Radial distortion
; Distance Distor
; [mm] [um]
0 0.0
2 35.6
4 64.7
6 88.0
8 103.1
10 107.0
12 97.6
14 74.2
16 38.1
18 -7.7

```
20    -58.1
22    -107.3
24    -148.7
26    -172.2
28    -167.1
30    -116.5
32    2.1
; end
```

In blue you have all the values you will have to enter in the software. As the model is in meters, you must pay attention to all units. Enter successfully all the values inside the dialog box.

- Enter the focal length. You must enter 35.124e-3 or 0.035124.
- As the pixel size is in µm, you must enter 9e-6.
- As the PPS is (0.000, 0.000), nothing to do.
- Enter -0.025e-3 for Xppa and -0.298e-3 for Yppa.
- For each line of the radial distortion, you must click the button **+** to create a new entry. Pay extremely attention to the unit:
    - The values of the first column must be multiplied by 0.001 or you can just add e-3 at the end.
    - The values of the second column must be multiplied by 0.000001 or you can just add e-6 at the end.



Interface allowing to enter camera internal parameters

When all the camera geometry parameters are entered, you can click the button **Save** to store it on the disk. For example, you can call the file “MyCamDef.incam”. Then select the second texture in the list and in the **Internal parameters** tab select the option **Make parameters editable and set constraints**, click the **Load** button and select the file “MyCamDef.incam”. Repeat this process for all the textures (a message will ask you if you want to apply the same settings to all pictures, say yes).

Enter camera external parameters

Open the file “TextureCamParamPosition.txt” in a text editor (like Notepad).

```
TextureCamParamPosition.txt

Sequence Omega Phi Kappa
type    ID       X           Y          Z     OMEGA[X]  PHI[Y] KAPPA[Z] [GRAD]
 OPK  1029 1741505.310 2298619.433   696.784 -42.9810  52.8317 158.7793
 OPK  1030 1741457.621 2298668.477   696.503 -42.0097  56.2967 156.7910
 OPK  1032 1741376.607 2298772.582   696.442 -39.7197  55.2205 154.9949

Sequence Phi Omega Kappa

type    ID       X           Y            Z      PHI[Y]    OMEGA[X]  KAPPA[Z]
[GRAD]
POK    1029 1741505.310 2298619.433    696.784   60.5220  -27.7245  124.8091
POK    1030 1741457.621 2298668.477    696.503   63.4225  -25.4050  122.3794
POK    1032 1741376.607 2298772.582    696.442   61.6217  -24.6692  123.0326
```

Select the first texture in the list (1029), go to the **External parameters** tab and select the option **Edit camera origin parameter and set constraints**”. Enter 1741505.310 for X, 2298619.433 for Y and 696.784 for Z. Then select the option **Edit camera orientation parameters and set constraints**, and select **Aerial OPK photogrammetry** and **Grad** (as it is mentioned in the TXT file with camera positions). Then, enter -42.981 for Omega/X, 52.8317 for Phi/Y and 158.7793 for Kappa/Z. Then repeat the same process for the other textures with data from the TXT file.
Once all parameters (internal and external) are entered for all textures, you can click the **Preview** button to obtain the result of your texture mapping. If you observe in detail the result, you should see that some triangles are not textured because they are not visible from any point of view corresponding to the camera positions:

- Some parts represent a big surface and can be considered as “normal” zones, in particular on the left part of the picture 1029.
- Some parts represent a very small surface (1 or 2 triangles) in the very deep holes of the rock

We will modify the parameters so that all the small isolated parts are textured:

- For the images 1029 and 1032, you can select the option “Visible vertices only”.
- For the image 1030 you can take the option “Include invisible parts”.

Click the **Preview** button again to see the difference.

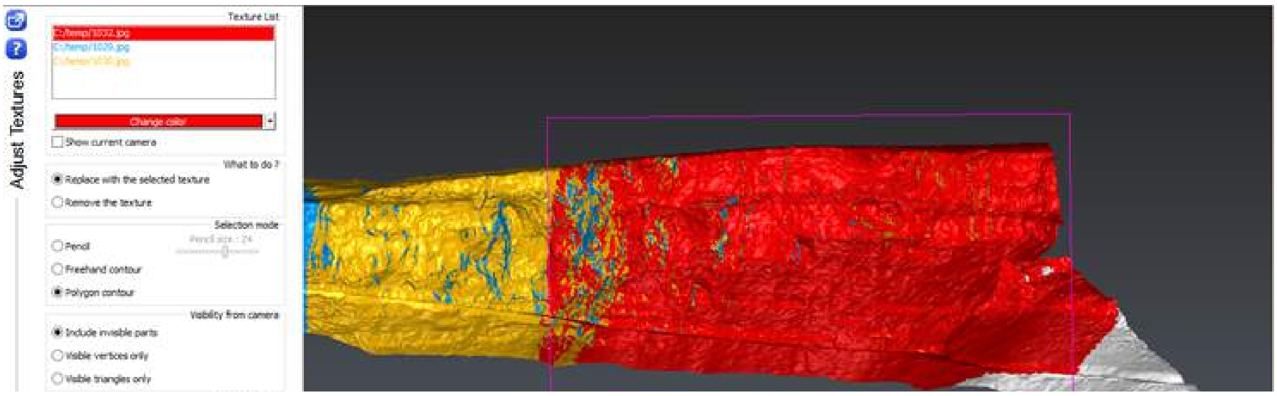
The cliff textured with a perspective view

Adjust textures

By default, when a triangle can be textured by several pictures, a choice is made according to two parameters:

- The distance between the camera position and the triangle.
- The angle between the camera orientation and the triangle normal.

However, in some cases, you may want to select manually the texture to apply. To do this, select the textured mesh and go to Image\Adjust Textures. Each texture is replaced by a color in order to better visualize where each texture is mapped on the mesh. Select the texture 1032, the option **Replaced with the selected texture**, **Polygon contour** and **Visible vertices only**. Then draw a polygon by clicking four points in the 3D scene like in the following pitcure (validate with a double click).



Adjust textures

The selected area will take the color of the texture 1032 (red in the Figure 5). Then select options **Remove the texture** and **Pencil**, and then "erase" the triangles on the right. They will become white, means that they will not be textured. Click **OK** to validate.

Export a textured mesh

Now that the texture is applied and adjusted, we can export the mesh. You just have to select it, then go to File\Export\Export Selected Mesh(es). OBJ format allows export of textured meshes. Note that some software are not able to handle files with big coordinates, so maybe you will have to translate your mesh near the origin.

> You can also save your file as a RSH file and then open it in the free viewer.

12.1.4 Exercise: Texture a building mesh with an ortho-image

In this exercise, we will texture the outer walls of a building in several steps, using:

- an ortho-image,
- a material (repetitive picture), and
- a point cloud with real colors.

Open the file

Open the file “TexturePhotomodel.rsh”. It contains a mesh and a point cloud with real colors. A texture, an ortho-image and its reference file are in the sample folder ("TexturePhotomodel.tif", "TexturePhotomodel.txt", "TextureMaterial.png").

Texture the main frontage with an ortho-image

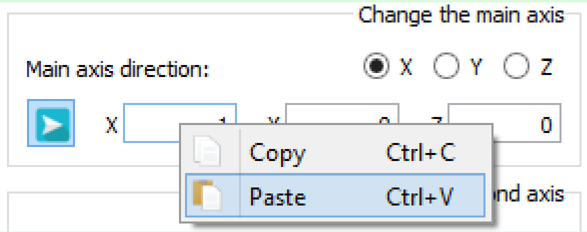
It is possible to texture a mesh with an ortho-image in Z direction only. As we want first to texture a facade in X direction, we need a UCS (User Coordinate System). The origin of the UCS will correspond to the top-left corner of the picture, and Z axis of the UCS will be parallel to the direction of the ortho-image (in most cases perpendicular to the wall to be textured). All that information to define the UCS are in the reference file "TexturePhotomodel.txt".

Show only the mesh and launch Construct / User Coordinate System.

- specify the **Origin** of the UCS with the **Pixel Top Left** coordinates from the reference file,
- choose the **Main axis direction** to be **X** axis and specify its new coordinates using **Horizontal axis Top Right** from the reference file, and
- choose the **Second axis direction** to be **Z** axis and specify its new coordinates using **-(Normal axis)** from the reference file.

Tip & Trick

New coordinates X Y Z of a vector can be copied and paste from the reference file the 3 at once.



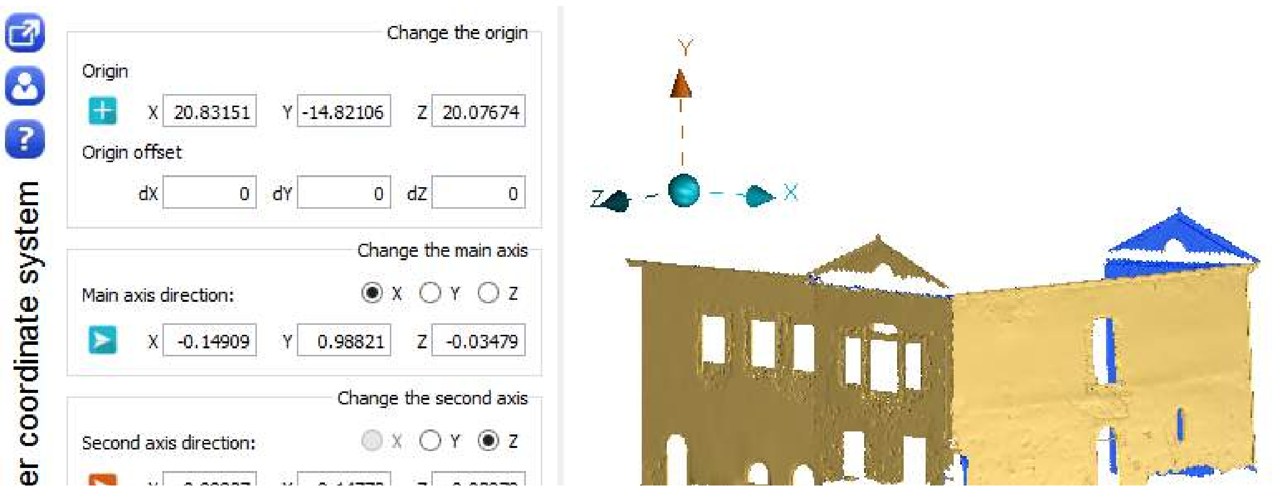


Fig.1 Define an appropriate UCS

Once the UCS is created, select the mesh and launch Image / Texture From Pictures / Texture Ortho-image . Load the picture "TexturePhotomodel.tif". Since the UCS corresponds to the ortho-image, we do not have any translation or rotation to apply. Simply set 0 for the **image coordinates** and for the **image orientation**. Then set the **pixel size** on object to 0.005 (**Pixel Size** in the reference file). Choose the option **Include invisible parts** and click **OK** to validate.

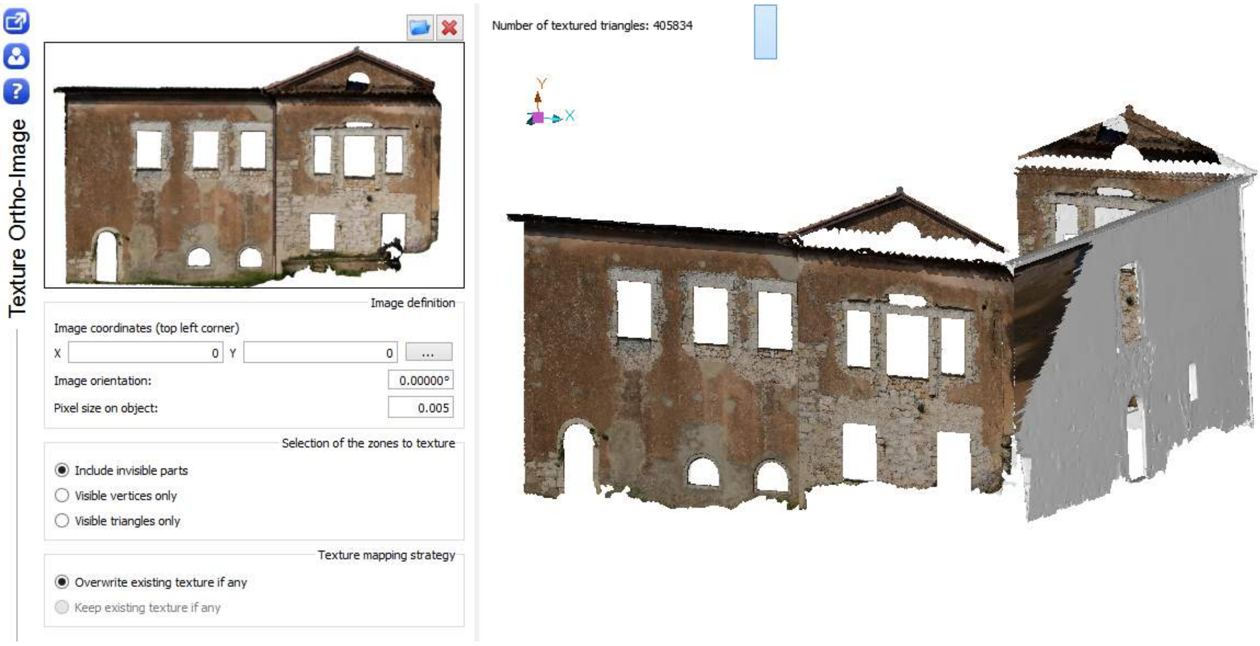
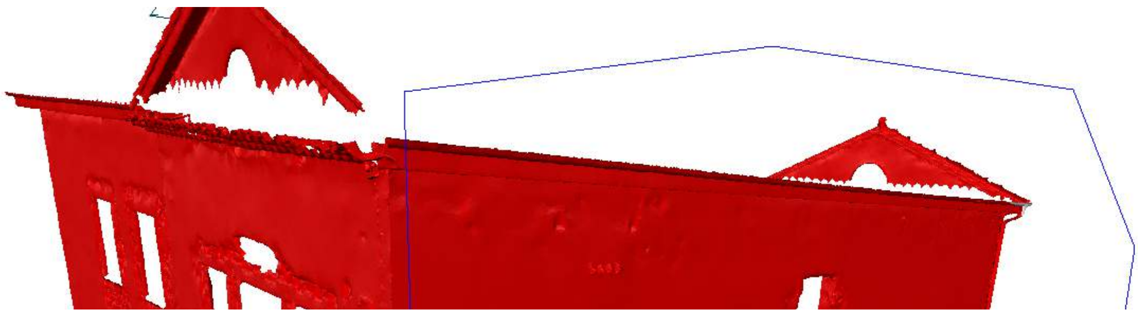


Fig.2 Texture Ortho-Image command

Now, the first side is well-textured. However, the ortho-image overlays other walls. We are going to correct it in the next steps.

Adjust texture on the side wall

Before texturing other faces of the building, we will first remove the texture coming from the ortho-image from walls where it should not be applied. Select the mesh and launch Image / Adjust textures. Choose **Erase the selected texture** to remove the texture on the other sides. You can begin to select the faces with a **Polygon contour** and continue with the **Pencil**. These tools never select invisible faces. In order to complete this work, you have to rotate the view and select the remaining faces. You may also activate the perspective mode. Click **OK** to validate the command.



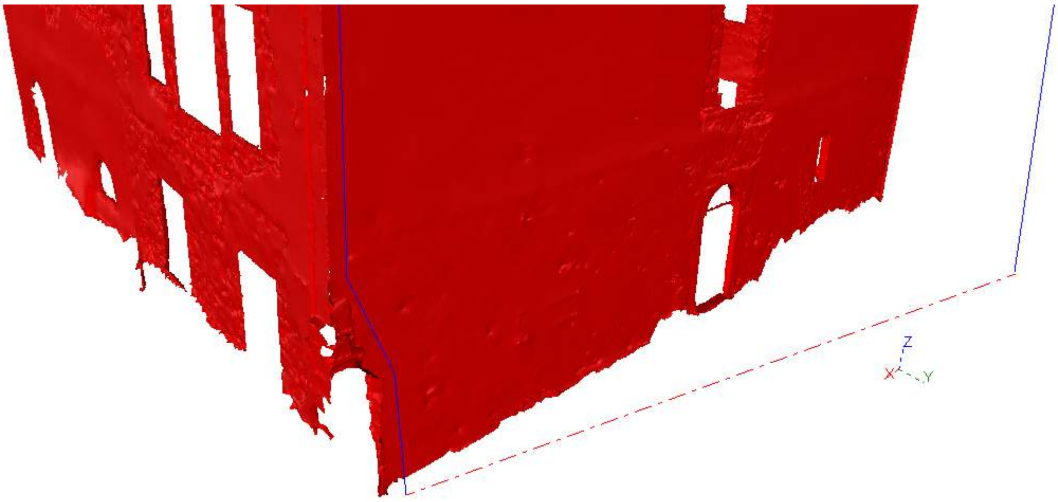


Fig.3 Erase a texture (first step)

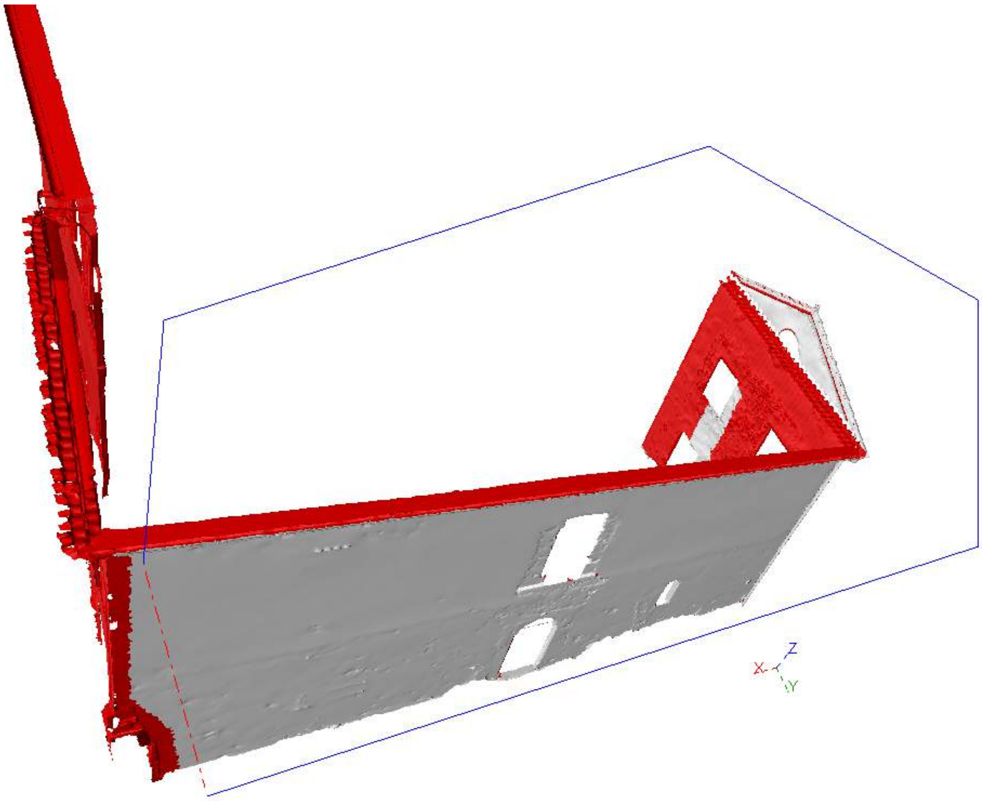


Fig.4 Erase a texture (second step)

Texture the side wall with a material

The next step will be to texture the second wall using a material. Select the mesh and launch Image / Texture material. Add the file "TextureMaterial.png". Choose to Keep the proportions and select Keep existing texture if any. Set the View direction to X. Changing the **numbers of repetitions** can improve the aspect of the texture: the picture is very stretched with 1 repetition whereas the result is better with 10. Try several **numbers of repetitions** and keep 10 repetitions, for instance.

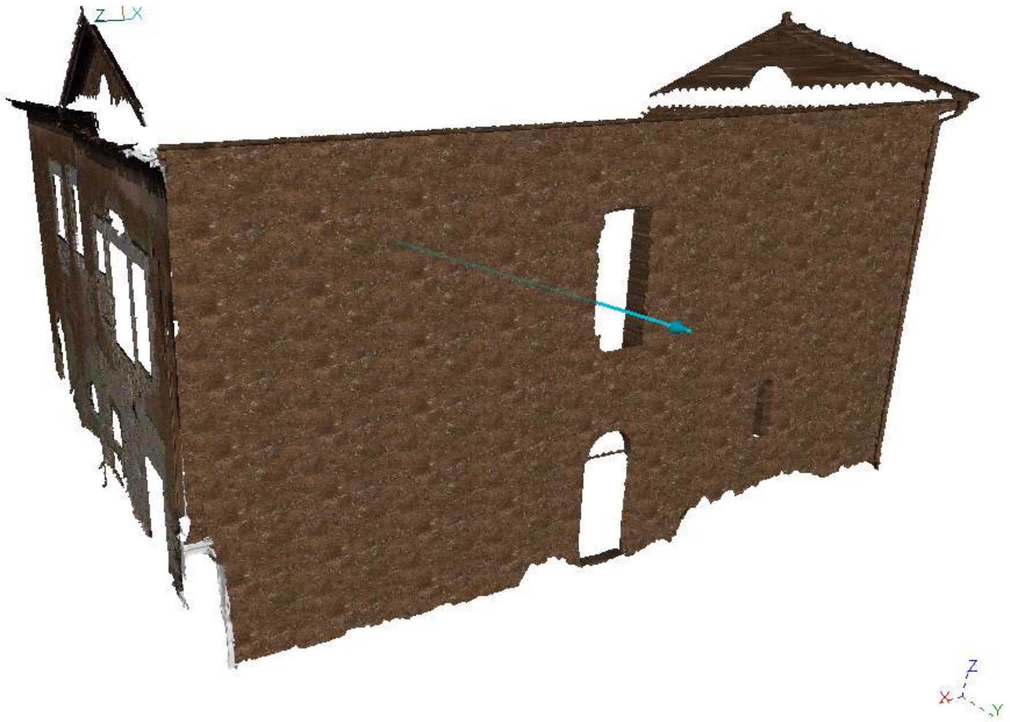


Fig.5 Textured with ortho-image and a material

Similarly to the ortho-image texturing process, the texture material will be applied on all the untextured parts of the mesh. The repetitive texture will produce unsatisfactory results on the third wall. To correct the aspect for this third wall, the applied texture must be removed by using the same technique as described in step **Adjust texture on the side wall** to prepare the next side.

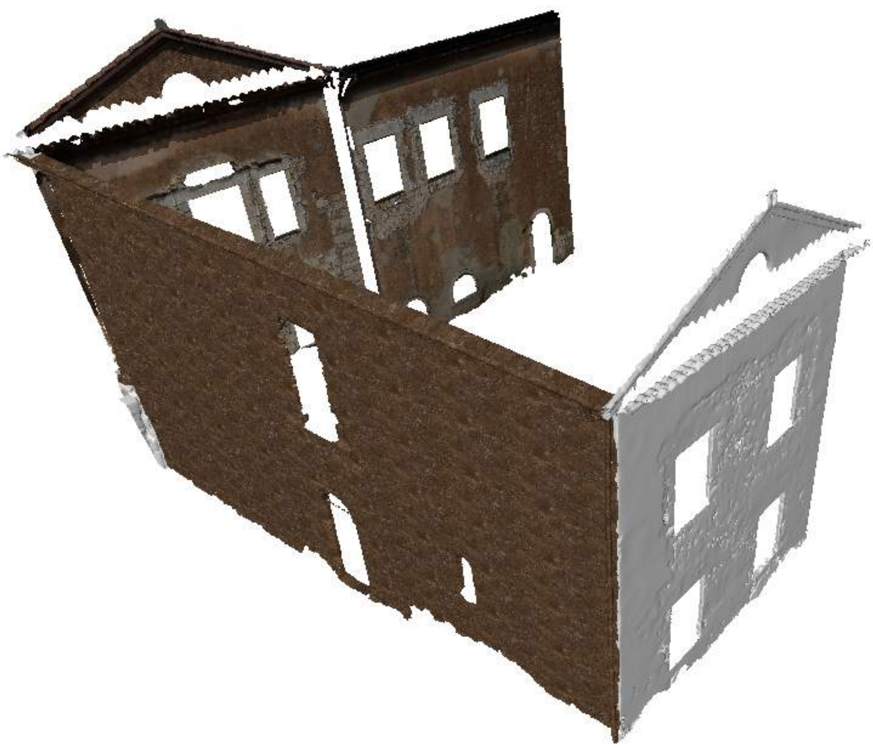


Fig.6 Erase a texture again

Color the mesh with a point cloud

We will now create a texture on the last wall using the colors coming from the original point cloud. To do it, the workflow is to first color the mesh, then convert the colorization to a texture.

Select the mesh and the point cloud and launch Mesh / Take Color from Cloud. Actually, the mesh has a texture but no real color. Consequently, you can choose both options **Previous color, if any**. Choose to use point color **if the distance to vertex is less than** 0.05. Note that the result depends on the size of triangles (refer to Exercise: Apply the color of a point cloud on a mesh).

Finally, select the mesh and launch Image / Texture From Color. Enter 4 pixels per triangle for the image resolution, and select the option **Keep existing texture if any** in order to texture only the last wall without texture.

Now, you can enjoy your photomodel!

Fig.7 Final result

12.2 Ortho-image

- Exercise: create an ortho-image and import it in AutoCAD
- Exercise: send an ortho-image to AutoCAD

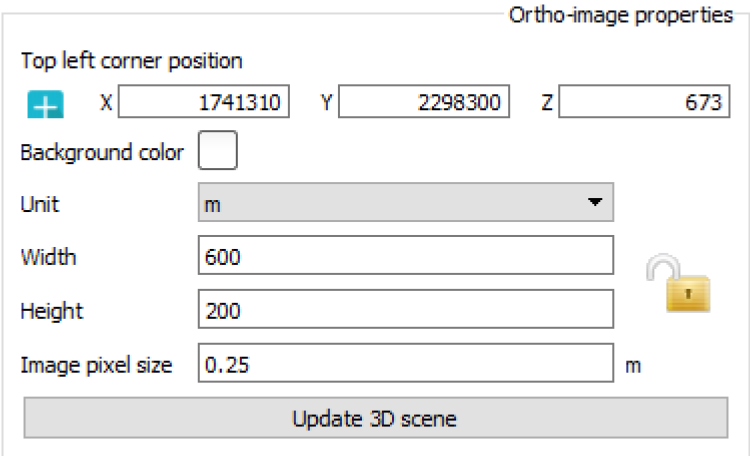
12.2.1 Exercise: create an ortho-image and import it in AutoCAD

Create the ortho-image

Open the file “TextureParam&CameraPath.rsh”.

Show only the mesh **CliffTextured** and launch the command Image / Create ortho-image.

Set the parameters as shown on the figure below:



Ortho-image properties
Check the **Save image and georeferencing information file** option and uncheck the **Send To AutoCAD** option.

3D scene

Click **Preview** to visualize the image and validate with **OK**. In the dialog **Select the destination file...**, enter **orthoimage** as file name and choose **jpeg** as file format.

Two files have been created:

- orthoimage.jpg: the picture
- orthoimage.txt: the georeferencing file

Insert the image in AutoCAD

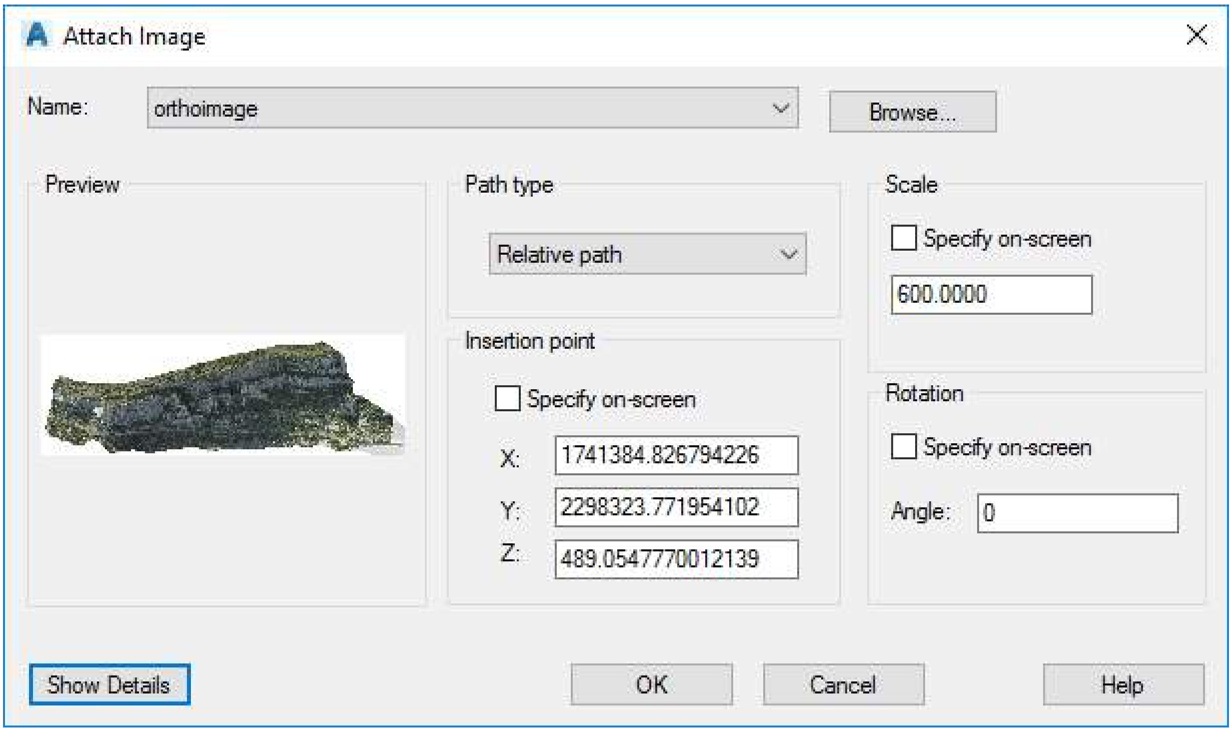
Open orthoimage.txt with a text editor like Notepad and an empty file with AutoCAD.

From AutoCAD, enter **IMAGEATTACH** in the command line prompt and select **orthoimage.jpg**

Tip & Trick

You can also add a **Raster Image Reference...** from the menu **Insert...** or try to find the function from the ribbon.

Uncheck all the options as below. Then copy-paste the insertion point from **orthoimage.txt** (see the **Autocad import** section) to **Insertion point** group. Do the same for the **Scale** and set the **Rotation** to **0**. Validate with **OK.**



Attach Image

Rotate the image in AutoCAD

Now, it is necessary to rotate the image.

Warning

Choose counterclockwise either **Decimal Degrees** or **Radians** as Drawing Units (enter **units** in the command line prompt or launch **Units** from the menu **Format**).

Warning

If more than 1 rotation is needed, remember that a first rotation along an axis will modify the 2 others axis. That is why, this exercise shows you the worst case you can find: 3 rotations.

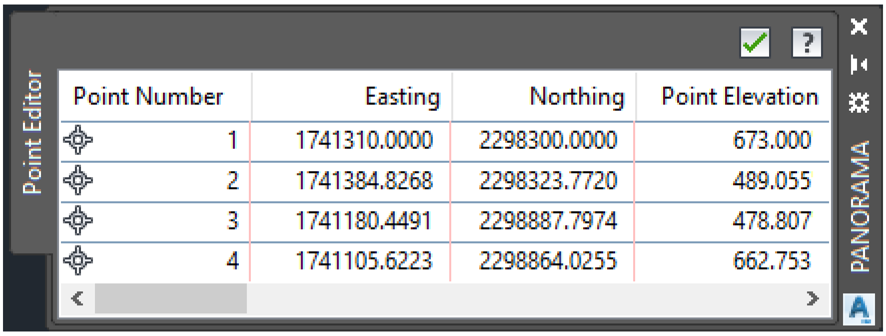
If rotation Z is not 0, enter **ROTATE3D** in the command line, select the image, select the Zaxis direction, select bottom left corner in image object and copy-paste **rotation Z.**
Then if rotation X is not 0, enter **ROTATE3D** in the command line, select the image, use **the bottom left and bottom right points of the image object** to define the rotation axis (X'axis) and copy-paste **rotation X** .
Finally if rotation Y is not 0, enter **ROTATE3D** in the command line, select the image, use **the bottom left and top left points of the image object** to define the rotation axis (Y''axis) and copy-paste **rotation Y.**

Command line prompt

```
Command: _imageattach

Command: ROTATE3D
Current positive angle: ANGDIR=counterclockwise ANGBASE=0
Select objects: 1 found
Specify first point on axis or define axis by
[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: z
Specify a point on the Z axis <0,0,0>:
Specify rotation angle or [Reference]: 107.6246919131824


Command: ROTATE3D
Current positive angle: ANGDIR=counterclockwise ANGBASE=0
Select objects: 1 found
Specify first point on axis or define axis by
[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: Specify second point on axis:
Specify rotation angle or [Reference]: 66.88607460754096


Command: ROTATE3D
Current positive angle: ANGDIR=counterclockwise ANGBASE=0
Select objects: 1 found
Specify first point on axis or define axis by
[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: Specify second point on axis:
Specify rotation angle or [Reference]: 2.49350983527415
```

AutoCAD result

You can check the result with corner pixel coordinates (see the **Image attributes** section in orthoimage.txt).



Point Number	Easting	Northing	Point Elevation
1	1741310.0000	2298300.0000	673.000
2	1741384.8268	2298323.7720	489.055
3	1741180.4491	2298887.7974	478.807
4	1741105.6223	2298864.0255	662.753

Control points

12.2.2 Exercise: send an ortho-image to AutoCAD

Open the file “TextureParam&CameraPath.rsh” and a new empty file with AutoCAD.

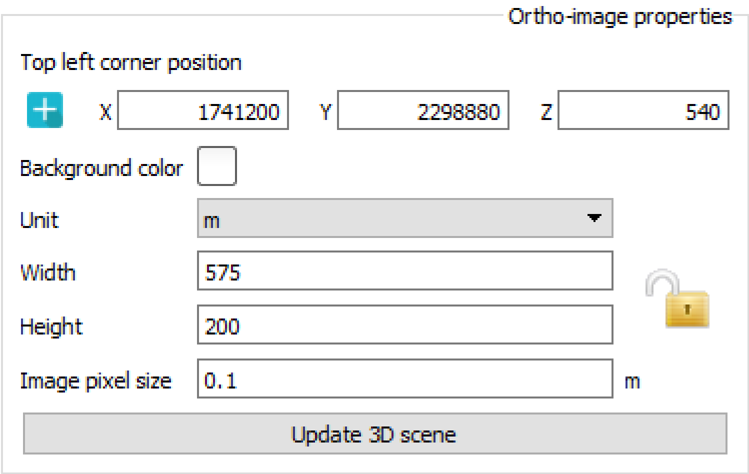
Warning

The 3DReshaper plugin for AutoCAD is needed to do this exercise.

Press **Z** on your keyboard to display a view from the top and do a freehand rotation along Z axis in order to create a landscape orientation image.

3D scene

Launch the command Image / Create Ortho-image. Set the parameters as shown on the figure below (for example):



Ortho-image properties

Uncheck the **Save image and georeferencing information file** option and check the **Send To AutoCAD** option.

Click **Preview** to visualize the image and validate with **OK**. The ortho-image has been inserted in your DWG drawing, in the active layer.

> The ortho-image file is added in C:\Temp folder.

> Check the **Save image and georeferencing information file** option in order to create the ortho-image in a custom folder.

12.3 Virtual visits

In the software, you can create and export a video of a trip between the objects in the 3D scene. You can either draw a polyline representing the path of the camera for the video, or define several camera positions between which the camera's path will be interpolated.

- Exercise: Create a video with a camera path

- Exercise: Create a video with a camera scenario

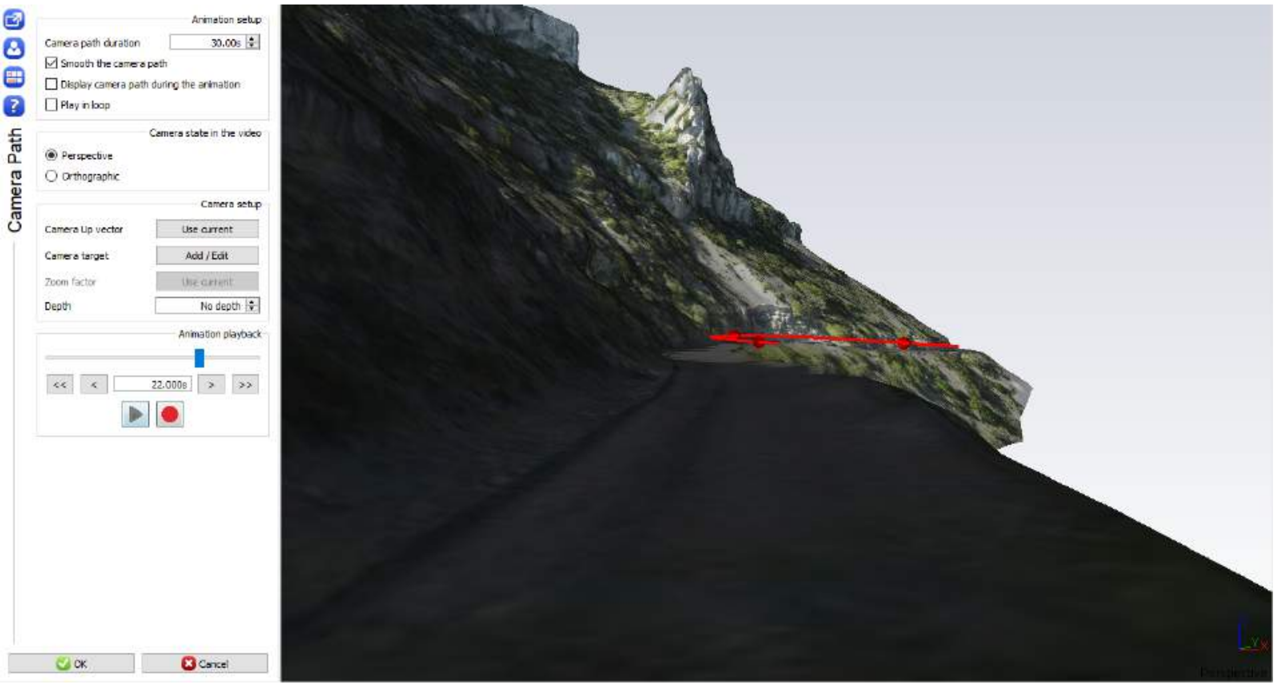
12.3.1 Exercise: Create a video with a camera path

Open the file “TextureParam&CameraPath.rsh”

This file contains a textured mesh of a cliff, also used for the texturing exercises.

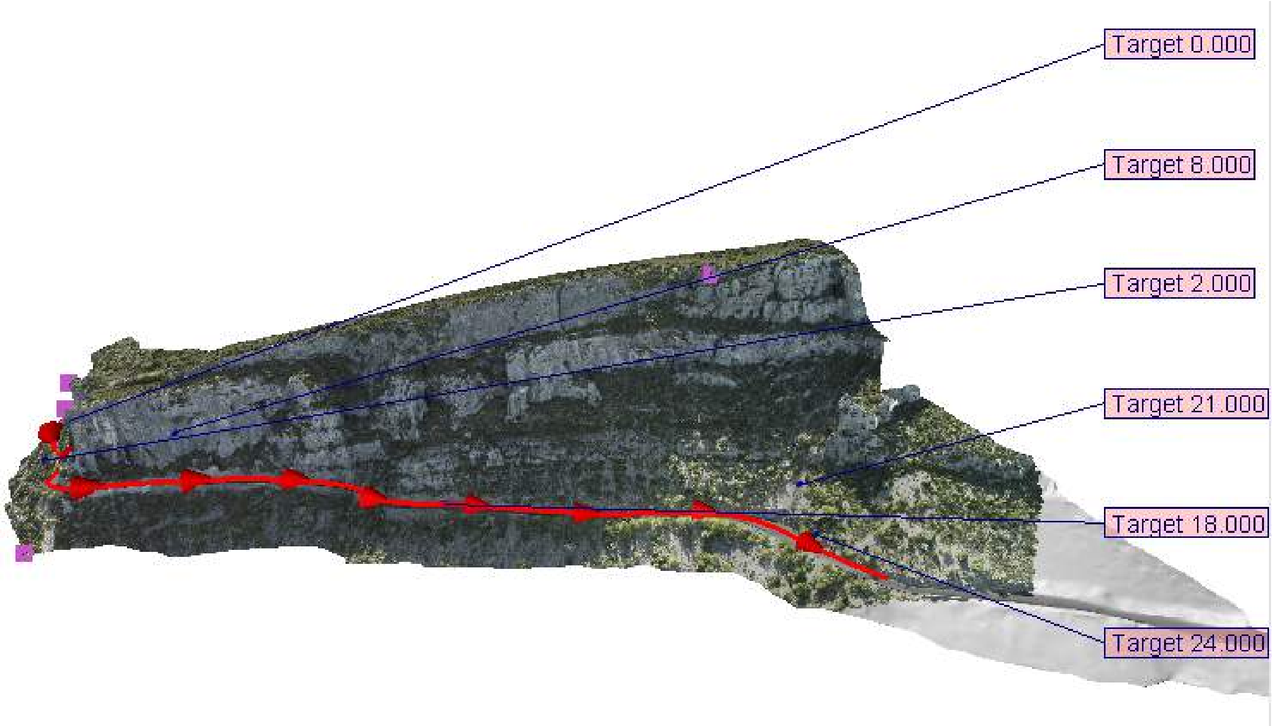
First, it is important to check the direction of the polyline because, by default, the camera orientation will follow the camera path selected. To check this parameter, select the polyline and go to Home \ Colors and aspect. Check if the parameter **number of arrows** is at least 1 for the current selection in order to verify if the arrows will be correctly displayed on the polyline to represent its direction. Click **OK** to exit. Now you can right click on the polyline and check **Reverse** to reverse its direction if needed. Then, use the perspective view, select the polyline **Camera path** and launch the command Image \ Camera path.

First, you can setup the options for the animation. The option **Smooth the camera path** is automatically check to smooth the polyline selected and have a fluid video. You can choose to display or not the camera path during the animation. You can also choose to play the video in loop if the polyline is closed, or back and forth if it is open.
Then, set the up vector of the camera by setting the view in the good orientation and clicking on **Use current**. For example, to set the up vector as the Z axis, click in the scene with your mouse and press the key **X** or the key **Y** to have the Z axis perfectly vertical, and then click **Use current** to set the up vector of the camera.



Virtual visit thanks to a camera path

During the animation, it is possible to add specific targets for the camera, at specific moments. To do that, move the slider when you want the camera to look at a specific point. Then click **Add/Edit** and click the point in the scene. Click again **Add/Edit** and then click on a new point to replace the target. Or click **Add /Edit** and then **Backspace** on keyboard to delete the target point created previously. If you click on only one target, the camera will stay focus on it. You can click on the polyline to orient the camera along the path again. If you want to use the same target than the previous one, click **Add/Edit** and press the key **Enter**.



Set several targets during the animation

To move forward or backward in the animation, use the slider or the buttons with the arrows. One arrow means one second forward or backward the current time. Buttons with two arrows allow you to go to the next or the previous target added.
If the focus is on the 3D scene, you can press the key **Space** to switch between the camera view and a view in which you can see an object representing the camera moving along the path (free fly mode).
You can preview the video with the button

. To record it, press the button

. It will be recorded as an AVI file. Then you can choose the codec to use for the compression.

We recommend you to install the free Xvid codec (http://www.xvid.org/Downloads.15.0.html) to compress efficiently the videos.

See the Help files of the camera path command for more details about keyboard shortcuts.

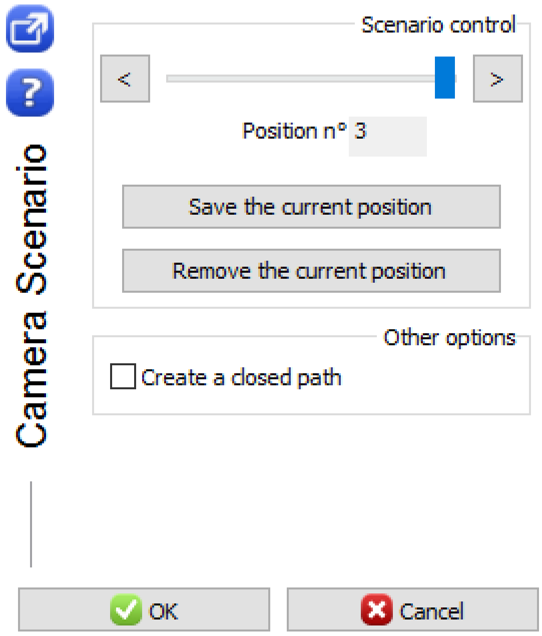
12.3.2 Exercise: Create a video with a camera scenario

A virtual visit can also be created by defining several views. Then a path will be interpolated automatically from these views.

Open the file “TextureParam&CameraPath.rsh”

This file contains a textured mesh of a cliff, also used for the texturing exercises and the previous exercise.

Go to Image \ Camera scenario. Set the 3D scene in the desired view and click on **Save the current position**. Proceed likewise for the next positions. A polyline representing the camera path is automatically drawn in the 3D scene.
You can go from one position to another by using the slider and the arrows in the upper part of the dialog box. Make a **Zoom all** in the scene to preview the path created. It is possible to remove a saved position by displaying it in the scene (use arrows or slider) and click **Remove the current position**.
If you would like to turn around an object, you could check the option **Create a closed path**.



Set several positions for a virtual visit

When you click **OK**, the dialog box corresponding to Image \ Camera path opens. The positions saved previously are automatically interpreted as targets. You can edit them as explained in the previous exercise. Click on the main arrow to preview the video and on the red circle to export it, as explained in the previous exercise.

13 CAD

- Generalities about Reverse Engineering
    - Introduction
    - Rules to make a good polyline network
- Generate patch on a mesh
    - Computing intersection
    - Computing NURBS curves
    - Computing BSpline surfaces
- Improve Surfaces
    - Improve continuity between surface
    - Make holes / restriction on surfaces
    - Making one surface only from patches
    - Exporting your model
    - Solving surface display artifact issues

13.1 Generalities about Reverse Engineering

- Introduction
- Rules to make a good polyline network

13.1.1 Introduction

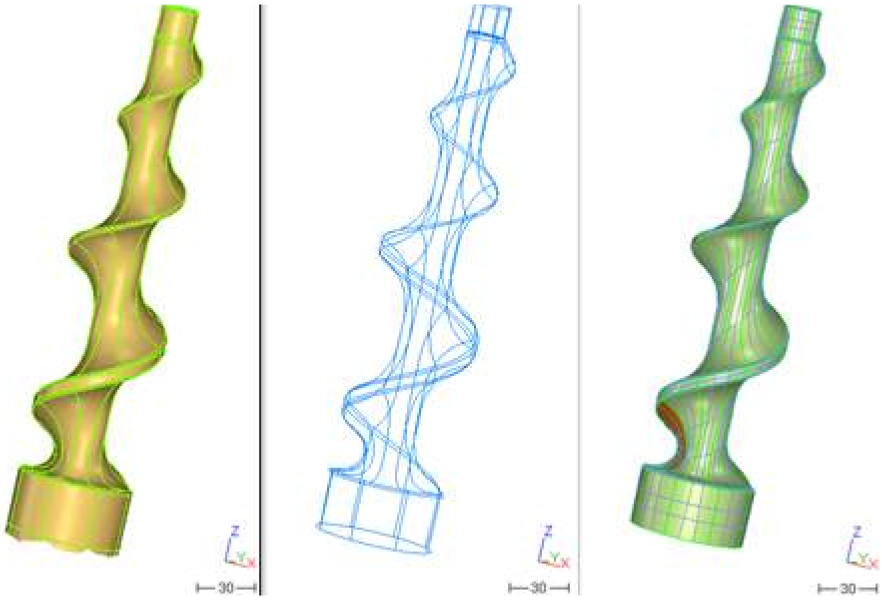
The polygonal mesh modeling created by the software generates models, made up with hundreds or thousands of non-continuous triangles. These 3D meshed models are ready for rapid prototyping, tool path generation, simulation, analysis, etc.

However, a “continuous” model is sometimes required by CAD-CAM software. This process of making a CAD model is also called “reverse-engineering” because you generate a continuous model, also called “exact model” from scattered data sets: mesh and point cloud.

This module allows you to make CAD Surface reconstruction starting from a mesh. CAD Surface means NURBS and BSpline surfaces that are fitted on your original mesh. Finally, these surfaces can be exported into IGES or STEP files and/or used to process inspections.

The process to create surfaces from a 3D mesh is divided in 3 parts:

- First, you have to create a polylines network in order to delimit the different zones having similar curvature properties on your mesh: fillet, planar zone. These lines must lie “on” the mesh.
- Then, this network of lines is used to create NURBS/BSpline curves using an automatic tolerance which can be modified after for a specific curve or for all curves.
- Finally, we create NURBS/BSpline surfaces using previous BSpline curves. These surfaces are fitted on the mesh. The patches are displayed with different colors, which evaluate the quality of the result.



The three steps in order to create surfaces

13.1.2 Rules to make a good polyline network

Our “Reverse-engineering” process is based on a polyline’s network that you must achieve first; so, before having your CAD objects (curves and surfaces), you must create delimitation with polylines on the mesh. To make a good delimitation you must follow these rules:

- Make borders on zones having same curvature characteristics: lines along a small radius, line along sharp edges, etc.
- Create polylines that intersect so that the software can easily determine the accurate intersection.
- Make contours with 4 sides (wherever possible).
- The lines that are created must lie “on” the mesh; otherwise some surface reconstruction may fail.

Make smooth polylines along curvature discontinuity

The goal of the surface reconstruction is to divide the complete surface of the model into elementary surfaces called “patch” or “face”. The most interesting property of the NURBS / BSpline mathematical definition is that the surface is continuous. “Continuous” means that the shape changes smoothly from a point to another point of the same face.

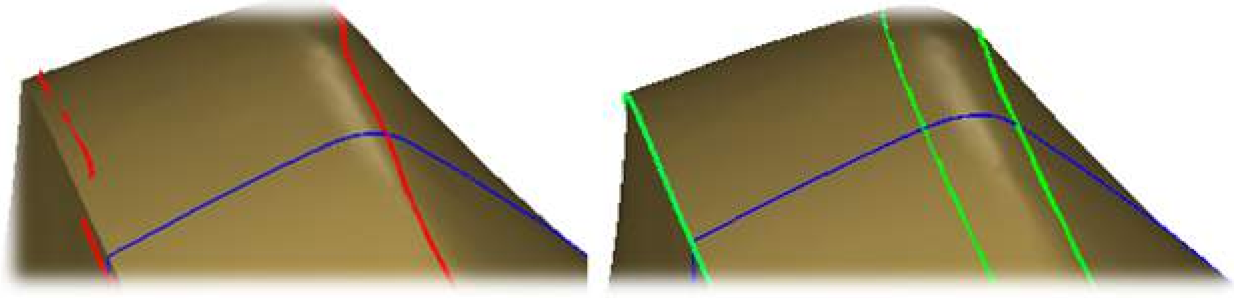
Some discontinuities may exist in a surface but they are always located on the border between two patches. This never occurs inside one patch. You may have two types of discontinuity:

- Tangency discontinuity: typically this occurs when you have a sharp edge on your model.
- Curvature discontinuity: typically this occurs when you have a fillet on your model.

When you create your polylines network you must follow the discontinuities on the surface as shown on the figure below.

When you have a fillet, it is important to consider that there is one curvature discontinuity on each side. This means that you should have 2 curves: one on each side on the fillet like on the right picture and not only one on the top of the fillet, as shown in the picture on the left. These lines can be extracted with Feature Line
.

=



Make polylines intersecting each other

You must have intersecting lines in order to create a real network. From a network of intersecting lines, The software will automatically calculate the intersections and trim irrelevant parts. Every time a valid polyline contour is detected, The software will automatically transform polyline pieces into NURBS curves and fill a surface patch inside the contour.

Make contours with 4 borders

The mathematical definition of a NURBS surface has exactly 4 borders. Then, it is better to make as much as possible rectangular contours with 4 borders when designing patches.

When the software analyzes the borders, several situations may occur:

- If you have 4 sides; this is the ideal situation and this is the reason why the resulting patch will be displayed in green color.
- If you have less than 4 sides; the software will create a “degenerated face”, which means that the mathematical definition will keep 4 borders but some border(s) will have a null length. The resulting patch will also be displayed in green color.
- If you have more than 4 borders; Reshaper will analyze whether some borders are made of several continuous pieces of polylines.
    - If the software can successfully merge continuous borders and can find 4 sides at the end of the process; a 4 side face will be created. However, the resulting patch will be displayed in orange color because a bad accuracy may occur on the side coming from the merge of different pieces.
    - If no merge is possible; the software will create a bigger patch with 4 sides and limit the valid zone of the face on the contour. A face like this is called “trimmed face” or “restricted patch”. This situation is the worst case. This is why the resulting face is displayed in red color. When you get such a face, you can access to the base face using the command Explode into CAD menu.

13.2 Generate patch on a mesh

The main command to create surface patches is CAD \ Generate patch. This command is very powerful because it drives you directly from a set of polylines to the set of patch in different steps which are explained further.

The philosophy of this command is that at each step you can decide:

- To continue automatically to the next step. This is interesting if you want to make rapid surfacing because the execution of this command takes only a few minutes.
- To stop execution at a certain step to make manual control and (or) modifications. This is interesting if you want to have an optimized result.

You can launch the command with:

- A list of polylines + a mesh, or a Set of polyline and a mesh. Then the step 1 is displayed.
- A list of BSpline and a mesh. Then the step 2 is launched. The surface computation starts directly.

Open the file ReverseEng.rsh. This file contains a mesh **Screw** and some polylines created with the following commands:

- Freehand Sections
- Projected Polyline
- Extract all Holes & Borders
- Radial Sections
- Feature Line

- Computing intersection
- Computing NURBS curves
- Computing BSpline surfaces

13.2.1 Computing intersection

Select the mesh and all the polylines and then go to CAD \ Generate Patch. It will compute first all intersections between lines. An information message displays the number of segments detected and the module sets a random color for each of them. Each segment is a part of the initial lines.

After that, you have 2 choices:

- Go to the **next step** creation. We will take this option to **create** BSpline **curves**

- You may also exit the command to manually edit polylines. If you exit to make some corrections (move line extremity, delete lines …) on the network, you will be able further to re-enter into the "Patch Creation" command. For this, you just have to select all polylines and the mesh, and then restart the process.

> It may happen that you have a warning message telling you that you have some potential dangerous lines that can introduce troubles when you make BSpline surfaces. Our advice is to check (and correct if needed) these lines before running the next step. You can see these polylines into the object explorer (they appear in first position in the list). You may need to **Zoom on** these objects thanks to the contextual menu.

The lines that may alter the quality of the result are detected and identified in the tree so that they can be easily corrected

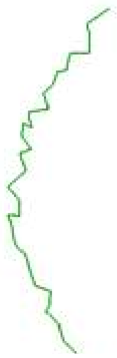
13.2.2 Computing NURBS curves

When your network of polylines is clean, you can start the **next step** in order to compute NURBS curves. In this process, lines of the network are approximated by BSpline curves, which are a type of NURBS, with an automatic tolerance.

The tolerance is a very important parameter because it has an influence on:

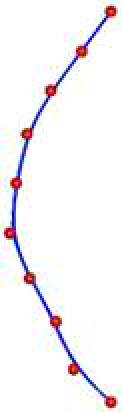
- The smoothness of the curve.
- The number of control points, which means the complexity of the curve. The computation time depends on the number of control points.
- The distance between the curve and the measurement, represented by the mesh.

A low tolerance needs a lot of control points to create the curve. If we apply a tolerance of 0.0, the resulting curve will have a large number of control points and may follow the noise of the mesh: in this case, at the end, CAD surfaces will not be relevant for you and not useful because they would not be smooth enough.

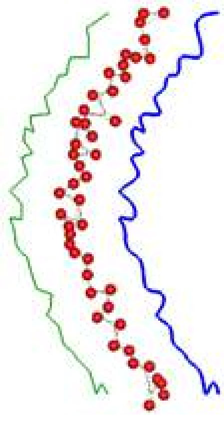


Example of polyline to approximate

The default solution proposed has 5 control points



With 10 control points, the resulting curve makes some undulations



Making an approximation with a null error provide a very bad quality

By default the software tries to find the best compromise and you can see the average deviation error for all curves and the maximum error for all curves. When the process is done, you can manually modify tolerance for all curves or just for some selected curves. To modify tolerance for only one curve, you just have to select it (Deviation values changes in the dialog box) and adjust tolerance with the slider.

The function manages also the tangency constraint when 2 polylines have similar directions. You can "Modify Tangent Constraints". By convention, tangent constraints are represented in green while non-tangent constraints are red. You can click on one tangency constraint or on two non-tangency constraints to respectively lock or unlock the constraint.

At this step, you can either exit command (to edit manually the curves with the command Construct \ BSpline \ Draw) or go to the **next step**, which is the final step: BSpline surfaces creation.

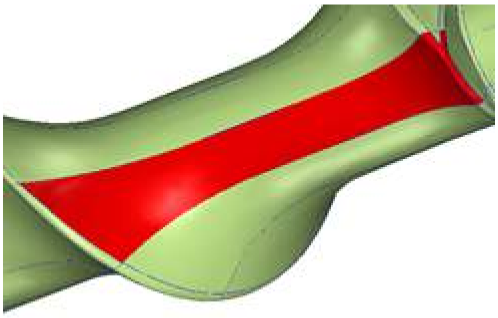
13.2.3 Computing BSpline surfaces

When you have adjusted correctly the tolerance of the BSpline curves, you are ready to create the final surfaces by clicking on the **next step button**. This step is fully automatic. The curve network will become patches borders. These surfaces will also be fitted on the mesh; you can see in the dialog box the standard and maximum error deviation between the mesh and surfaces.

Note that after validation, some artifact issues may appear. This can be easily solved by entering a 5µ deflection to the surface representation. This whole process is detailed in the paragraph Solving surface display artifact issues.

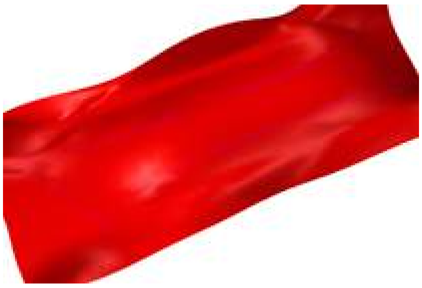
At the end of the process you can have 3 types of surface regarding the surface quality:

- Green BSpline surfaces: These surfaces are made with 3 or 4 borders (BSplines curves). With this kind of surface, you have G0 (G-zero) constraint between surfaces (they share the same curve border).
- Red BSpline surfaces: These surfaces have more than 4 border curves that cannot be merge to have only 4 curves (tangent vector at extremities are different). In this case, we make a greater surface (which has 4 borders), we fit it on the mesh and we make restriction on this surface in order to have the final surface. The problem of this type of surface is that the continuity is not G0 with neighbor surfaces. The restriction is made with the projection of the BSpline curve on the great surface.



A restricted surface is generated every time a contour with more than 4 edges is found.

Restricted surface with a contour of more than 4 edges



The restriction has been removed with the command CAD \ UnTrim Surface(s). We see the natural patch with the restriction contour

Depending on the result and surface quality you want, you may have to modify the initial line network or BSpline curves. All the intermediate polylines and curves coming from the previous steps can be found inside the folders **Contour Group** and **CAD group** of your object explorer.

13.3 Improve Surfaces

- Improve continuity between surface
- Make holes / restriction on surfaces
- Making one surface only from patches
- Exporting your model
- Solving surface display artifact issues

13.3.1 Improve continuity between surface

It is possible to apply tangency continuity (G1 constraint) between neighbor patches sharing a common edge. This command will align, as much as possible, tangency on the two patches common border.

To apply tangency constraint, select two neighbor surfaces and go to CAD \ Tangency Constraint. When the execution starts, in the dialog box, you can see values corresponding to the current angle between surfaces (standard deviation angle and maximum angle in degree). In the 3D scene, you also see normal vector for each patch along the common edge.

In the dialog box, you can decide to move only one surface or both. Select the option “Both surfaces” and apply the constraint with the **Preview** button, new values of the angle between the patches are displayed on the right, inside the Dialog Box. In terms of angle value, you will never obtain exactly zero but a certain error will remain. However, if the resulting error is “small” the surfaces can be considered as tangent. The threshold value to consider two surfaces as “tangent” is in relation with the surface quality that you want to output:

- If you obtain an angle less than 3 degrees, this may be acceptable in most cases.
- If you obtain an angle less than 1 degree, this can be considered as a good quality.

- If you obtain an angle less than 0.5 degree, this can be considered as a very good quality with a severe criterion.

You can also enter in the command CAD \ Tangency Constraint without making any selection. In this case, the function will work like a paintbrush. Every time you pass over a common border between 2 patches with the left button pressed, the tangency continuity will be improved.

13.3.2 Make holes / restriction on surfaces

Sometimes it is interesting to make hole directly on a BSpline patch: this is useful if you want a smaller number of patches or if you want to avoid tangency problem between surfaces.

To do this you have to create a polyline network of lines without taking care of the hole. The best is to have a large rectangle around the hole. The resulting NURBS patch will completely cover the hole.

After that, you have to:

- Extract the hole with the command Construct \ Polyline \ Click Holes & Borders to get a polyline around the hole.
- Select the contour polyline and launch the command Construct \ BSpline \ From Polyline(s) to transform the contour into a NURBS curve.
- With the patch covering the hole and the closed curve representing the hole you can use the command CAD \ Hole/Restriction you can make a restricted patch. Do not forget to select the **Hole** option inside the dialog box before clicking the **Preview** button.

13.3.3 Making one surface only from patches

If you have followed this tutorial, you will obtain at the end a set of NURBS patches. However, in most situations, you want to get only one surface:

- This is easier to select and to handle.
- This is absolutely necessary if you want to make a comparison between the surface and something else because the command Measure \ Compare / Inspect needs:
  - One and only one reference object (and not a set of patches) as reference entity.
  - One and only one object to compare (and not a set of patches).

To do this:

- select all the patches that you want to group together in a unique surface, and
- launch the command CAD \ Create compound.

13.3.4 Exporting your model

When you have reconstructed your CAD curves and surfaces, you may want to use them with other CAD-CAM software.

You have the export function, which allows you to export your CAD objects into IGES or STEP files. Select objects that must be exported and go to the Export / Export Selected CAD object(s) - IGES/STEP

13.3.5 Solving surface display artifact issues

Sometimes, you may see that some patches do not have a good aspect because the reflection of the light inside the patch is not smooth. This is a side effect due to the technique used to represent a NURBS surface.

A NURBS surface cannot be displayed “as is”. It requires some transformation so that your graphic board can make the representation. This transformation is called “discretization” and consists in sampling the continuous surface with “discrete” points.

In this process, the surface is “simplified” with a certain error called “deflection”. By default, Reshaper takes a deflection of 0.05 but in some cases this value is not low enough to get a good representation.

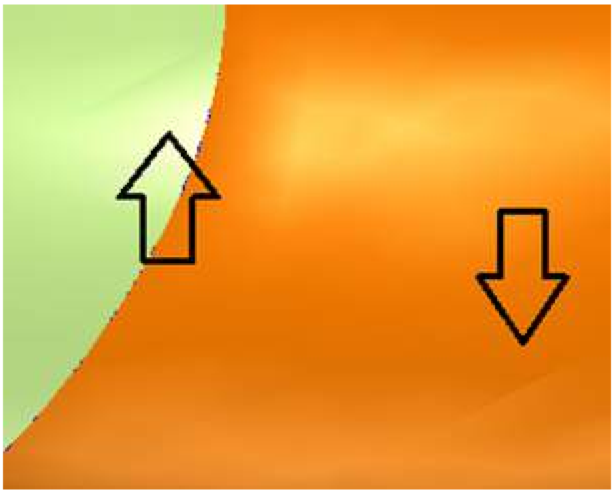
To change this deflection parameter, you have to select the patch(es) or surface(s) and launch the command Edit / Colors and Aspects. You will find a tab called “CAD” that contains the deflection of your model and you can change this value. You can also decide to change the default parameter so that the future patches are created with this new value.

If this deviation is smaller, the representation will be better. However, you should avoid giving a value too small because:

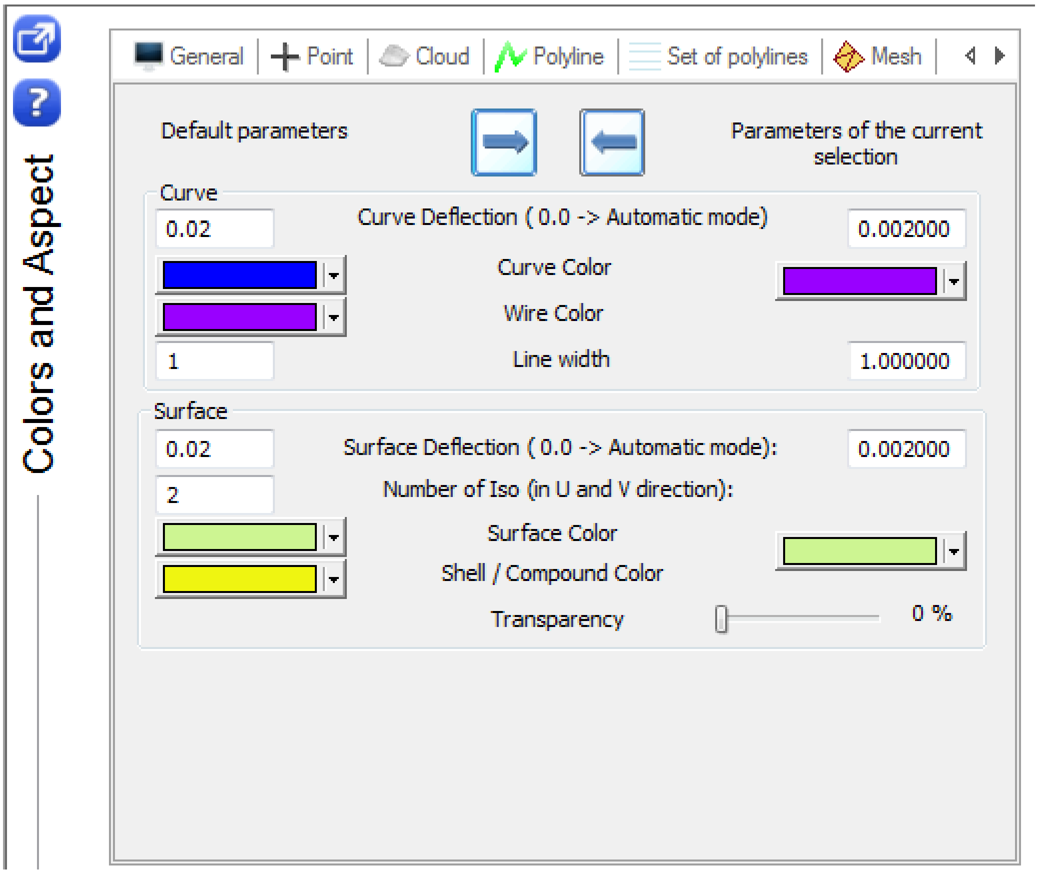
- The number of “discrete” points will increase.
- More RAM memory will be needed
- Response time will be longer.

Artifacts that may appear during the surface display. On left: deflection=1; In the middle: deflection=0.1; On right: deflection=0.01

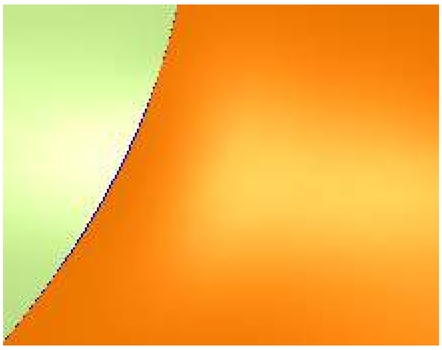
In the case of this exercise, you can enter 0.005 (5µ). However, this value must be adapted to the object size. In other words if you reshape a ship, a value of 5µ is not relevant!



Before display artifact correction



Reducing the deflection to correct the display artifacts



After display artifact correction