- when it has to be **right**

BEGINNER'S GUIDE

Help 2022.1

1 Legal notice

The goal of this beginner's guide is to learn how to start using Cyclone 3DR

This manual is provided for informational use only, and is subject to change without notice. *Leica Cyclone 3DR* 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 Cyclone 3DR, called the **software** in the following.

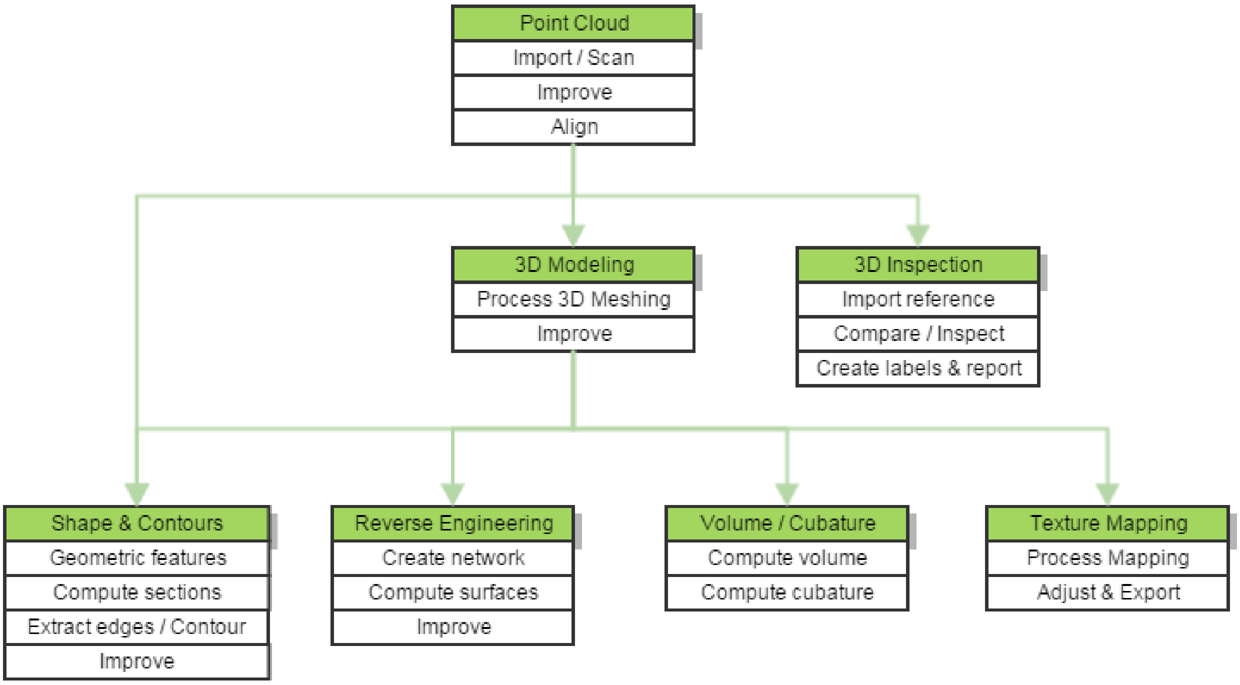
All sample files used in this guide can be downloaded in **Technical Documents** section from Cyclone 3DR - Downloads page.

3 Content

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

4.3 *Exercise: Browsing a project*

- Loading a 3DR file
- Changing the view
    - Rotating, panning and zooming
    - Predefined views
    - Keyboard shortcuts
    - Viewsets
    - Orthographic/Perspective
    - Limit plane
- Selecting objects
    - In the tree
    - In the 3D Scene
- Editing an object

4.3.1 Loading a 3DR file

Several options are available in order to open a 3dr file:

- Double-click on the file in your Windows explorer.
- Launch the software and run the command Open.
- Launch the software and then drag the .3dr file from your Windows explorer to the software.

For this exercise, open the **EnterPoints.3dr** file.

Drag and drop as well as double-click is also possible for file formats which are known by the software.

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.

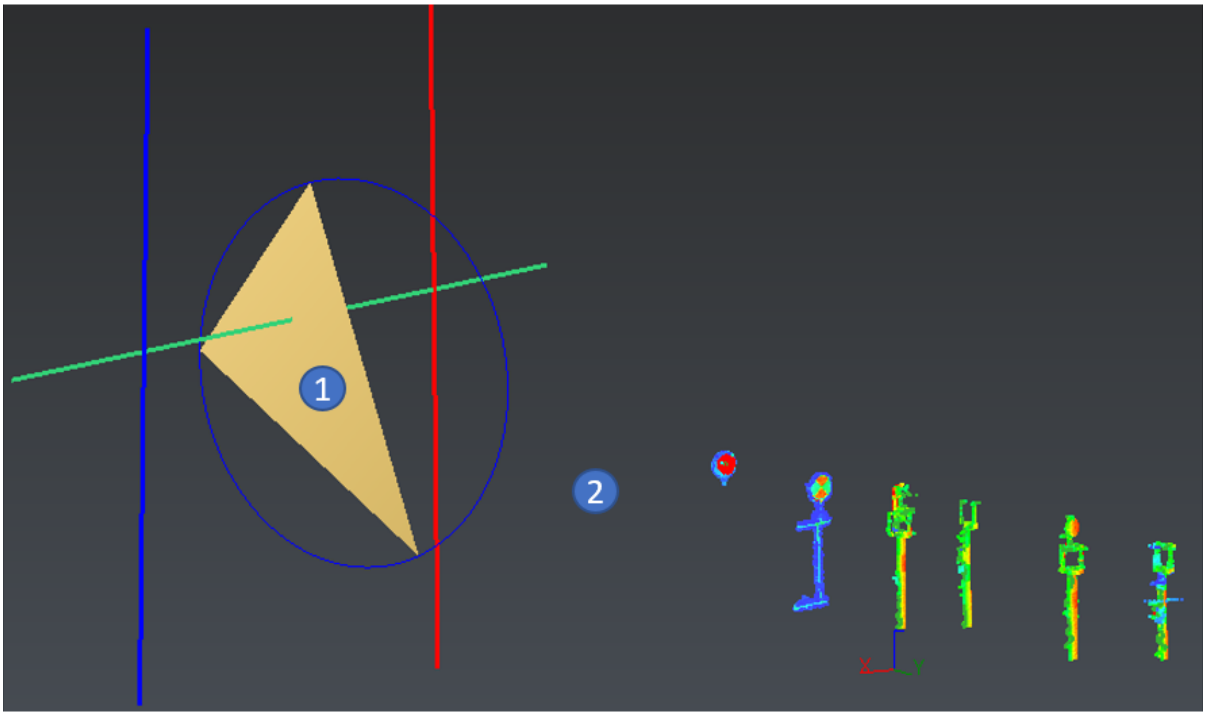
Rotating, panning and zooming

The mouse will allow you to manipulate the view. Press **O** to switch to orbit ortho camera mode.

To **rotate the view**, **left 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).

Note the Z axis is always parallel to the screen.



1 The mouse position will change the rotation conditions
Panning is also done using the mouse, by pressing **right button**.

Zooming in and out is possible by **scrolling the mouse.**

→ Scroll with your mouse in the 3D scene.

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

Predefined views

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

→ Run the Predefined Views (Front) command.

→ Run the Zoom All command.

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

→ Click on Split View Vertically

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

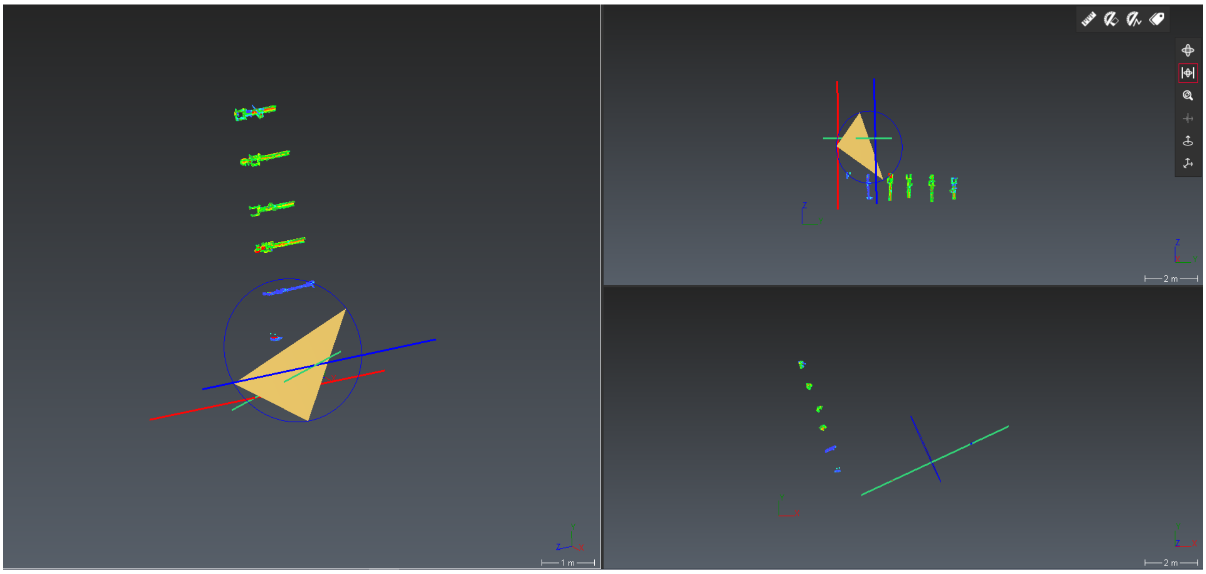
→ Click on Split View Horizontally.

→ Click on the top right view and then press the **X** key in order to obtain a Right 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 Keep Only One View.



2 Split views

Keyboard shortcuts

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

- **X**: changes the view to YZ view (also called Right view)
- **Y**: changes the view to XZ view (also called Back 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 Right to Left, and of course, the same mechanism is setup to change the view to Front view with **SHIFT+Y** and to Bottom view with **SHIFT+Z**. Refer to the dedicated Shortcut keys for more details about all available shortcuts.

Viewsets

The View Set command 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 magnifying glass.

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.

Orthographic/Perspective

The view can be switched between orthographic and perspective by using the tool bar on the right or shortcuts:

- **R**: orbit perspective.
- **O**: orbit orthographic.
- **P**: panorama (perspective).
- **L**: fly (perspective).

- **T**: top orthographic.

3 The tool bar contains buttons allowing to switch from orthographic to perspective

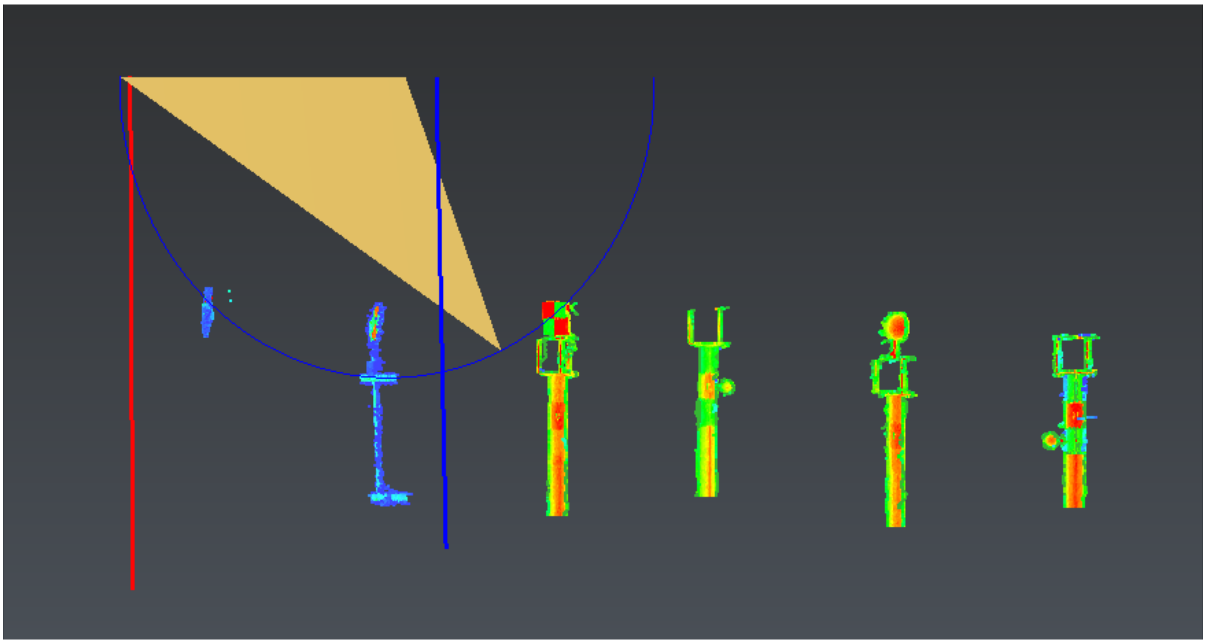
Perspective view can be very useful for looking inside objects. An alternative to perspective view is activating a clipping plane.
More information are available in the dedicated page: Perspective and orthographic view.

Limit plane

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

- Press **X** and launch the command Limit Plane (view top) without selecting any objects.
- Click a point on the triangle

A horizontal limit plane has been created.



4 Horizontal clipping plane

In the tree, you can hide (or display) and switch off (or on) the limit plane.

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

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.

In the 3D Scene

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

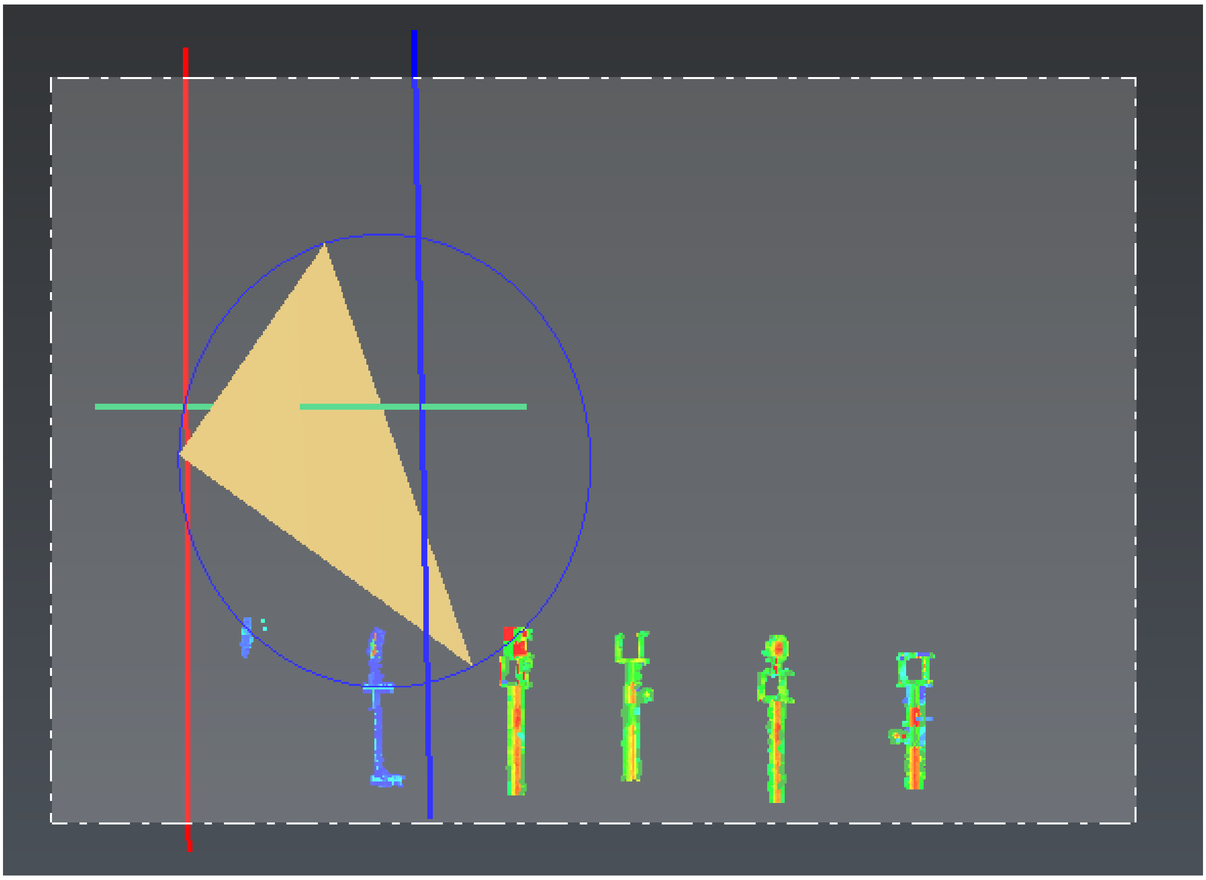
- by using the mouse **LEFT click** on an object.
- by using a rectangle selection (moving the mouse from one point to another with **SHIFT** and **LEFT click** of the mouse pressed).

Refer to the dedicated page Navigation and selection for the difference between rectangles created from left to right or right to left.

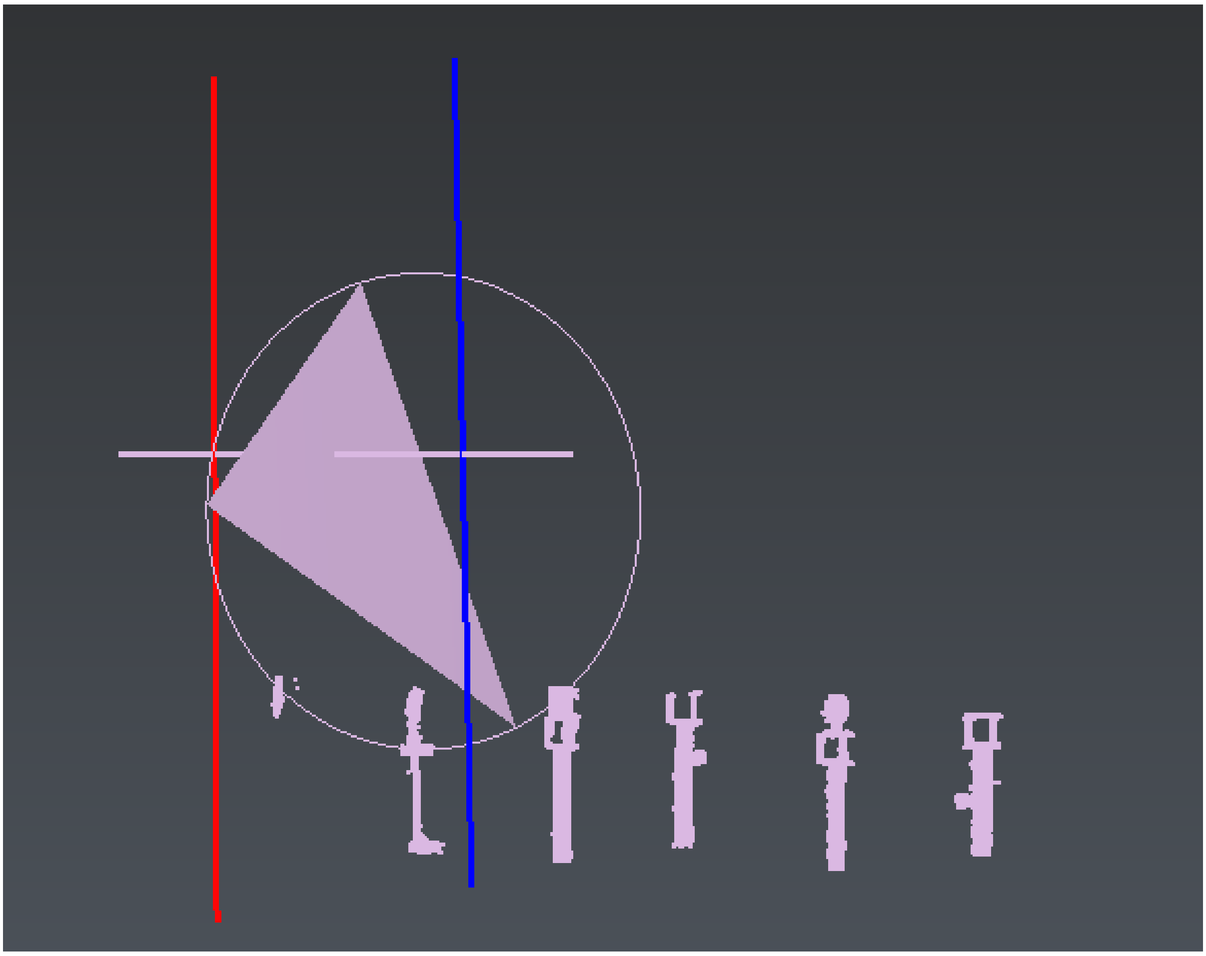
→ Press the **X** key and then on the **A** key to display all visible objects in front view.

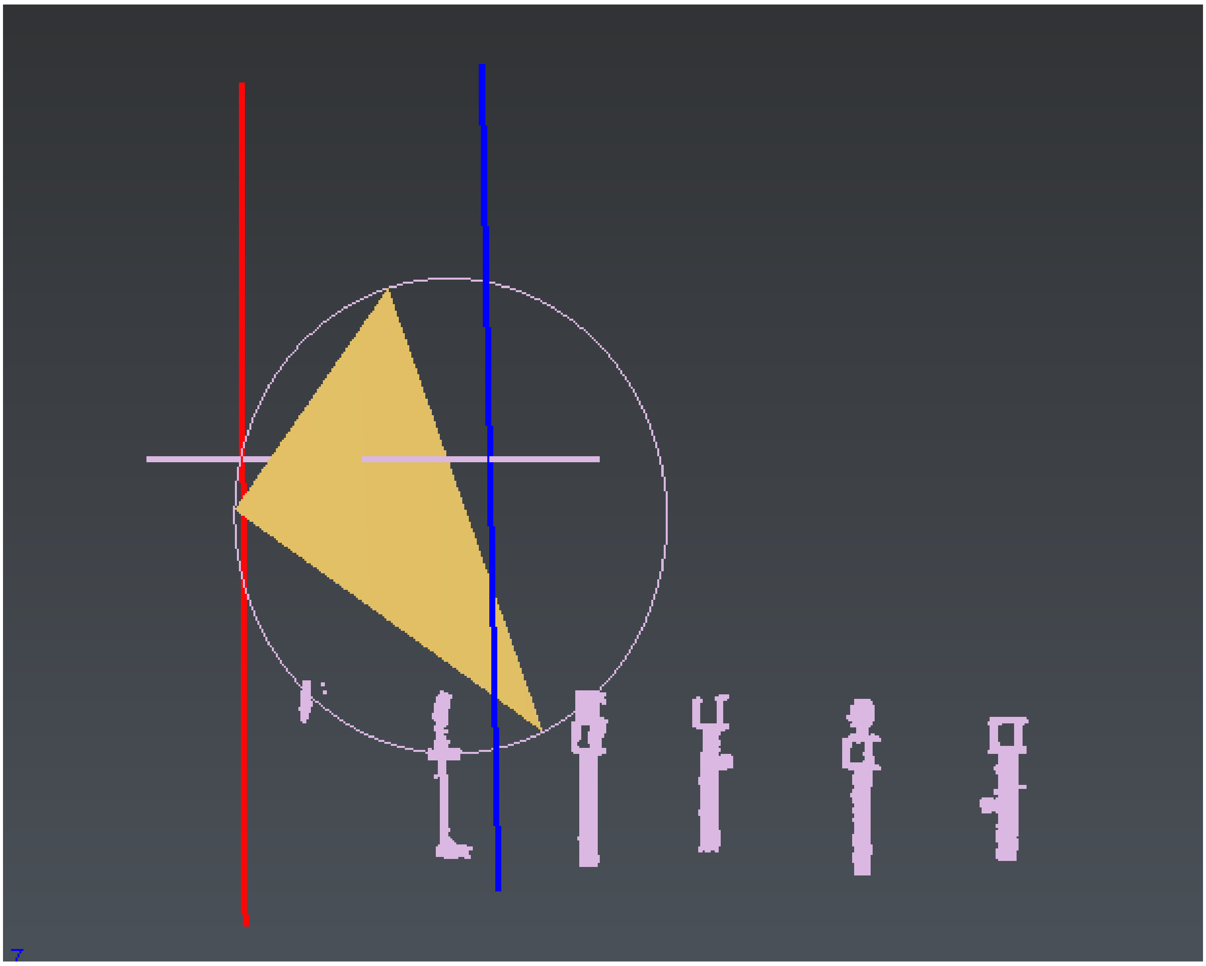
→ Draw a rectangle like in the 1st image below from left to right. The result of the selection should be similar to the 2nd image below

→ Press **CTRL** and **LEFT click** 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



5 Rectangle selection in the 3D scene





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 bulb 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 **Rename**
- In the tree, left click twice (slowly, to avoid double-click) on the element you want to rename
- Select the element you want to rename and press **F2**

→ 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. 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 Undo and 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 goes back to the "Geometric 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 Object representations for more details). Meshes, for example, can be displayed in: Smooth, Flat, Wire, Smooth+Wire, Flat+Wire, Textured, Real Color or 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 disk

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

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

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

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, geometries (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's properties using the command Properties in the contextual menu, or simply by keeping the mouse up to an object in the tree view.

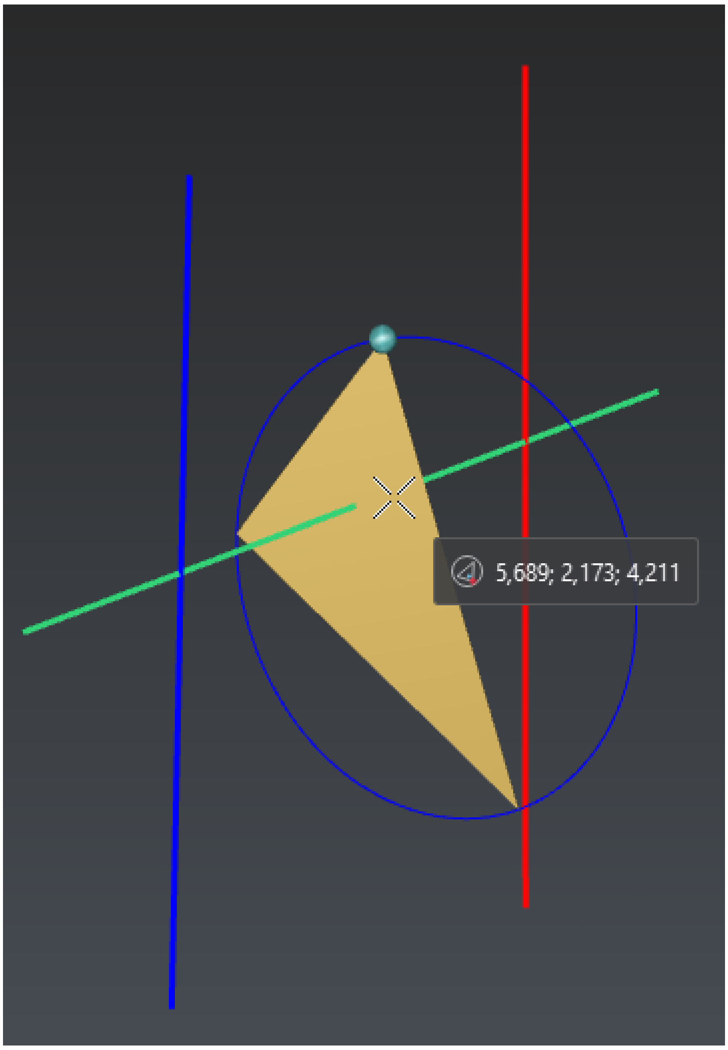
> When using the contextual menu, you can display properties of several objects at once if several objects are selected.

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

In the software, each time you have to click a point, a toolbar will appear. This toolbar 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). For more information, see Define points.

> Open the file **EnterPoints.3dr** in order to practice. Make sure all the objects from the tree explorer are visible and open the command Point.

Therefore, when the software is waiting for a point, you will see, next to the cursor, 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 this following picture the current point is (5.689, -2.173, 4.211) and is located on one vertex of the triangle.



6 When the software is waiting for an input point, 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 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 polyline; 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

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 no 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**, 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 no 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 these two lines, and you will not have a warning message.

4.4.6 XYZ

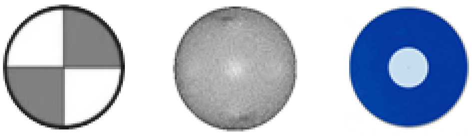
This option allows entering a point without projection on geometry. You can enter manually (or with copy/paste) an XYZ coordinate. You must enter in the field one, two or three values separated by a character that is neither a number nor a character nor a dot:

- 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 "DX3". The software will understand X=4.1; Y= 1.1; Z= 3.3 because the previous value was 1.1 and DX3 means "add 3 to X"; the Y and Z coordinates will keep the previous values.

However, you can also click on points directly in the 3D scene, even if there is no object behind the mouse. The point will be created in a plane parallel to the screen plane (impossible to control the depth).

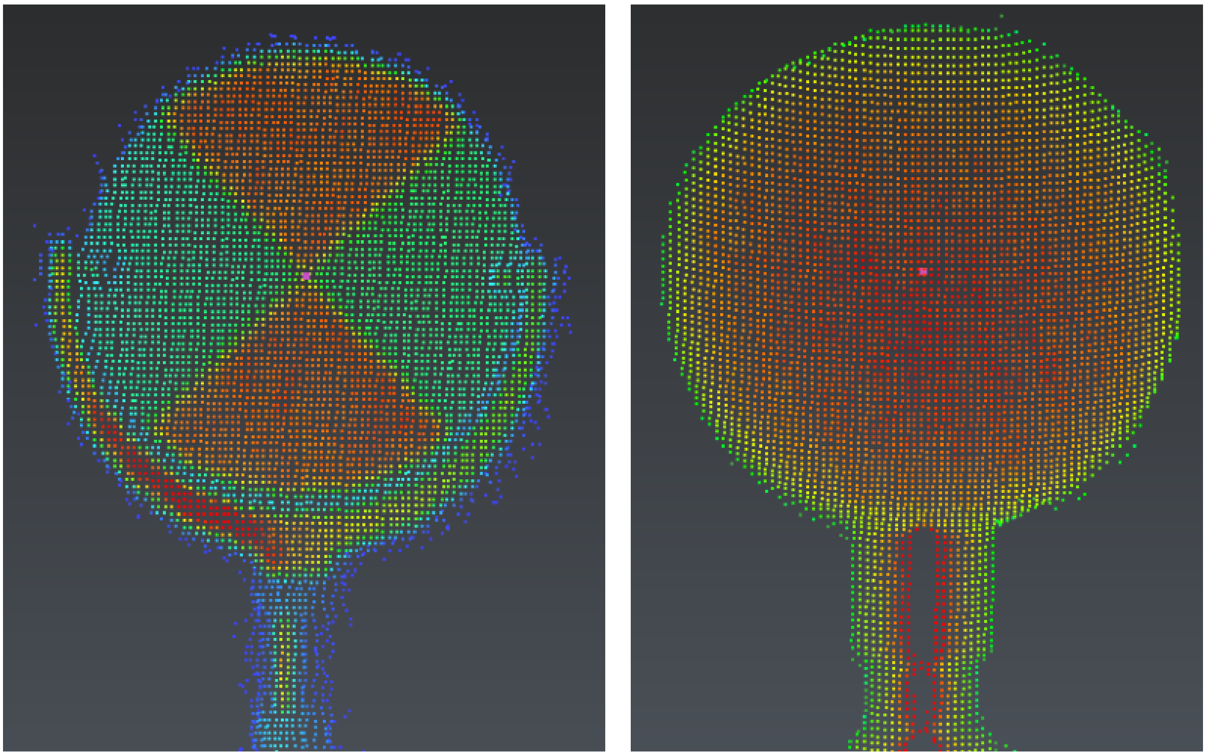
4.4.7 Surveying Target

⚠ This option only works on point clouds having intensity values.



7 Compatible targets

If your point cloud contains some surveying targets (black and white, spherical or blue and white) like one of the pictures above, you can click on the center directly. 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).



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

4.4.8 Highest Point

> ⚠ This option only works on point clouds.

With this option, you can click on the highest point of the cloud above a seed point. Try to click a point on a target: the created point will be at the highest point above the clicked point.

4.4.9 Lowest Point

> This option only works on point clouds.

With this option, you can click on the lowest point of the cloud below a seed point. Try to click a point on a target: the created point will be at the lowest point below the clicked point.

4.4.10 Ground Point

> This option only works on point clouds.

With this option, you can click on a point above or below the seed point at the ground level.

4.4.11 White Line

> This option only works on point clouds having intensity values.

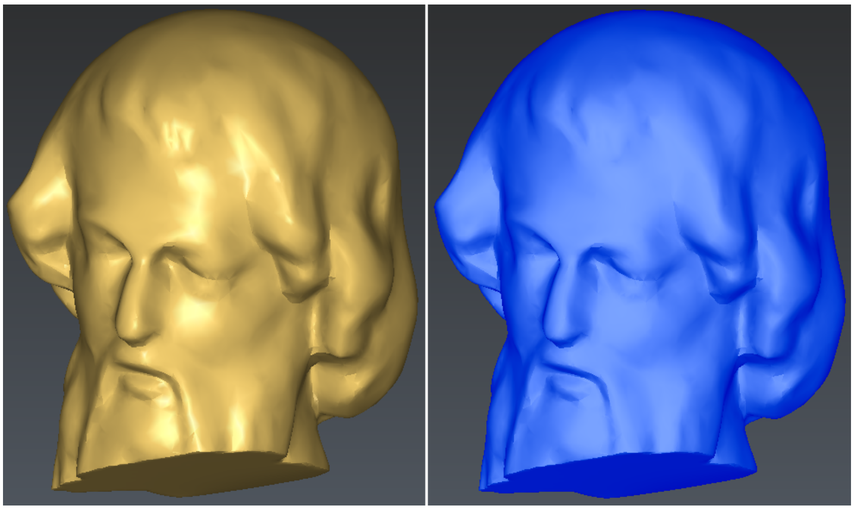
With this option, you can click a point at the axis of white lines thanks to a seed point.

4.5 Exercise: Understand meshes orientation

> Open the file **FillHoles.3dr**.

In the software, the default color for meshes is gold. Select “4-Closed Mesh” and show only the 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).



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

Dollhouse view

Launch Dollhouse to hide back-facing triangles, i.e. triangles displayed in blue in the example above.

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 ImportCloud.

5.1 Import a point cloud

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

Import 6 clouds

Import the files "**ImportCloud-1.asc"** to "**ImportCloud-6.asc"**

1. Open the menu **File \ Import**
2. Choose Add files and 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**.
3. Click **Import**.

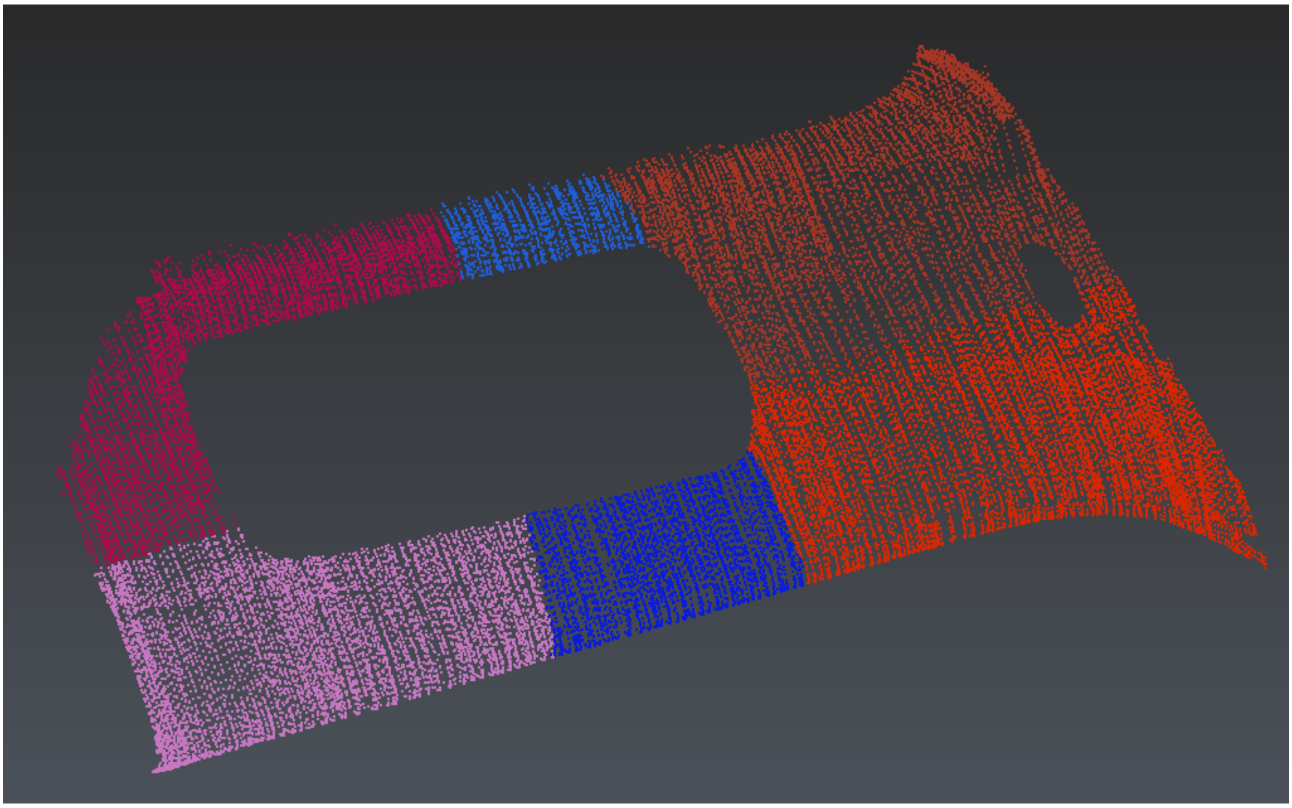
Six independent clouds are created in the Cloud Group.

Merge the 6 clouds

They can be merged into one unique cloud.

1. Select the 6 clouds and go to Merge Clouds.
2. Choose whether to keep the colors of the clouds or not.
3. Validate the result with **OK**.

You obtain one cloud in the tree, replacing the 6 selected clouds.



10 Merged cloud keeping the initial color of each cloud

Group

You can select all clouds, right click and use the command **Group** from the contextual menu.

5.1.2 Exercise: Convert a cloud from a unit to another

Open the file **BestFitOnRef.3dr**.

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 Resize. Define the center (0, 0, 0). 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 **OK**. Have a second look at the properties; the size is now exactly the same.

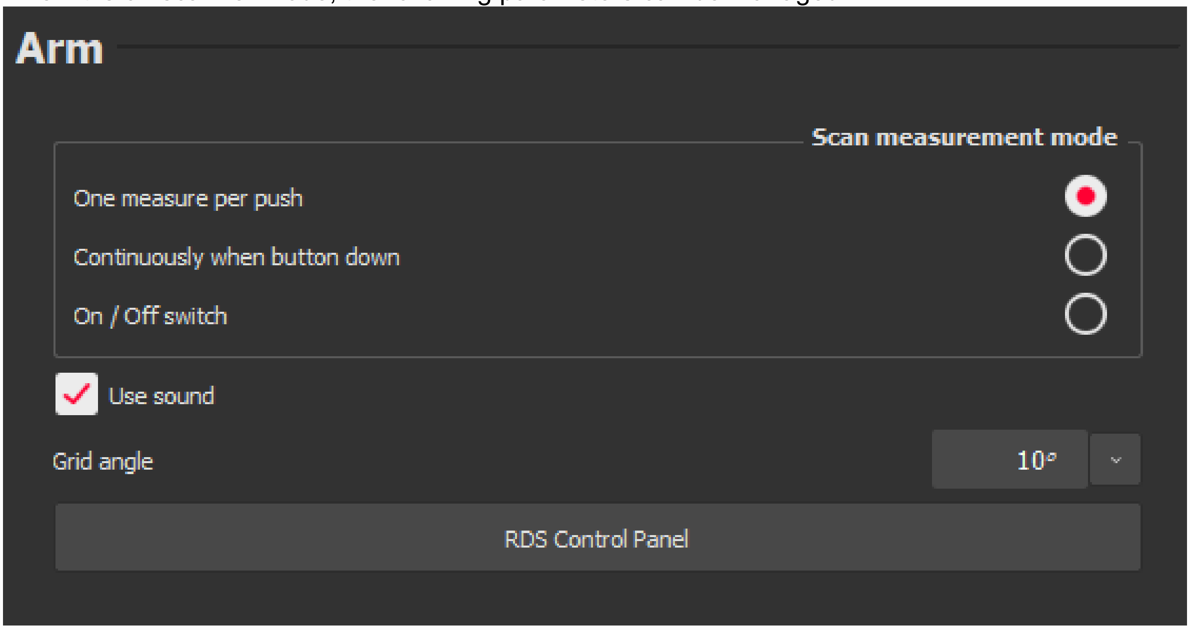
Before resize	After resize
Aligned Dam: CLOUD 500 000 points Max dimension: 30.7162 m	Aligned Dam: CLOUD 500 000 points Max dimension: 30.7162 m
Aligned Dam in ft: CLOUD 500 000 points Max dimension: 100.775 m	Aligned Dam in ft: CLOUD 500 000 points Max dimension: 30.7162 m

5.2 Scan point clouds

If the software is directly connected to a measuring arm or a handheld scanner managed by the RDS, you can go to Measure through RDS.

Arm settings

Launch the settings page Arm Settings in order to adjust the arm settings before scanning points. The software automatically detects if the measurement device is in probe mode or scanner mode. When it is on scanner mode, the following parameters can be managed.



Main parameters are:

- **Measurement mode**:
  - **One measure per push** is useful when you measure with a probe.
  - **Continuously when button down** or **On/Off switch** are more suited to measure with a scanner.

Scan point clouds

First, set the arm in scanner mode and launch the command Measure through RDS.

In the scene, a white line corresponding to the laser stripe seen by the embedded camera is displayed. So, to be able to acquire points, you must see the line (you have at the same time the preview of the current 3D points in the scene). In any situation, you need to take care of the focus between the laser and the camera (try to have the point laser on the laser line). Otherwise, the points will not be computed by the arm.
Once you have some points displayed in the scene (*i.e.* the focus is correct), you can launch an acquisition. Then, move the scanner on all the parts you want to scan, as a painter would do to paint the part. You see at the same time the part displayed in 3D in the scene (with a huge amount of points). Actually, you obtain a kind of reconstruction of the real part in the software.
Use the function **Auto view** to have a look at the region where you want to be precise, *i.e.* where you need many points.
Once the scan is complete, you can exit the command.

5.3 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
- Exercise: Separate walls and floors
- Exercise: Classify a cloud manually

5.3.1 Exercise: Remove or separate a part of the cloud

Open the file

Open the file **CleanWithObject.3dr**.

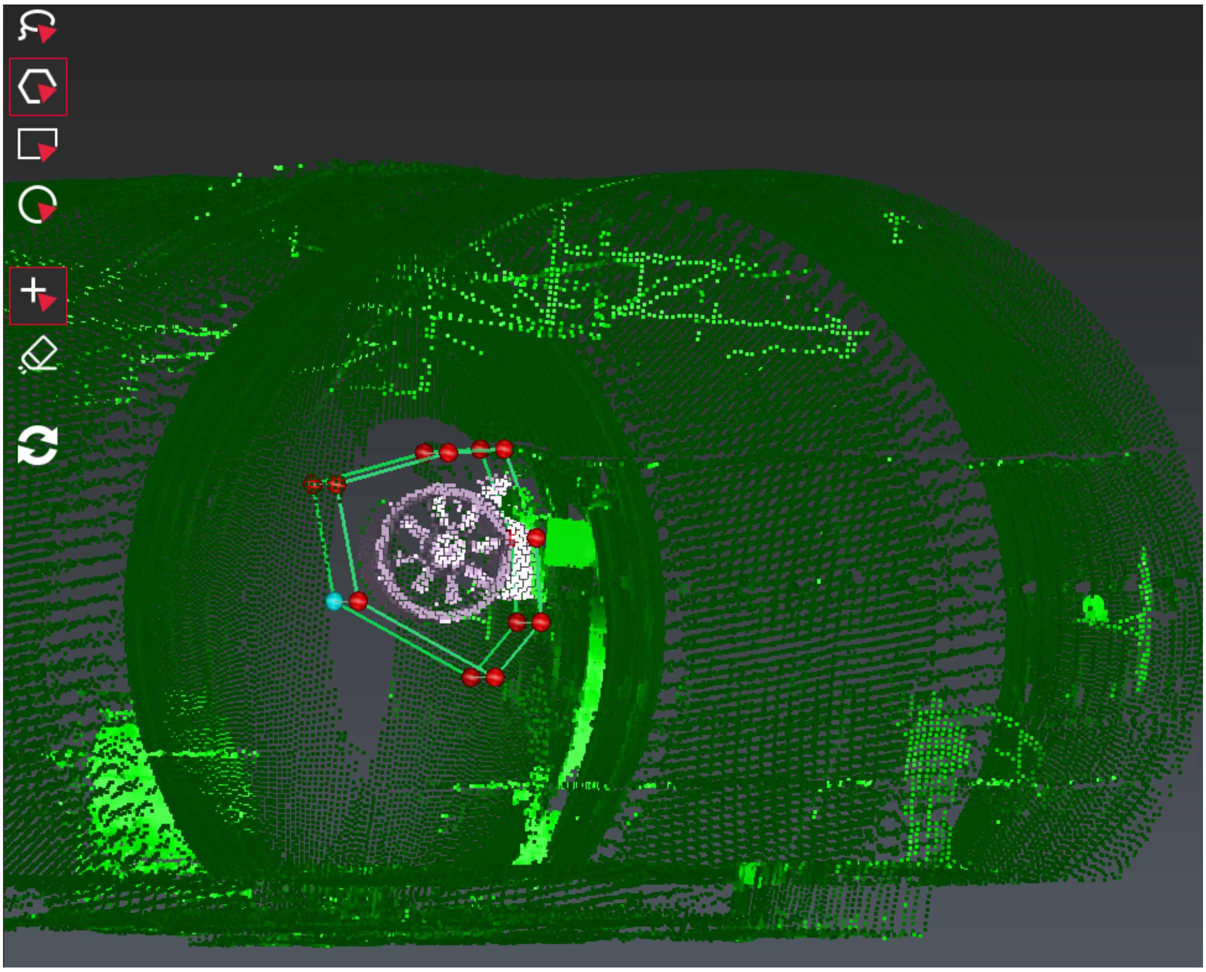
The file contains the point cloud of a tunnel. To make an inspection of the tunnel, we need to keep only the points from the vault. We can use three different tools to split a cloud into several parts and separate the vault from the rest of the points.

Draw a polygon

Select the point cloud **Tunnel** and go to 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 the selected points. Repeat the action on all the other big areas and validate with **OK.**



11 Remove a part of the cloud by drawing a polygon

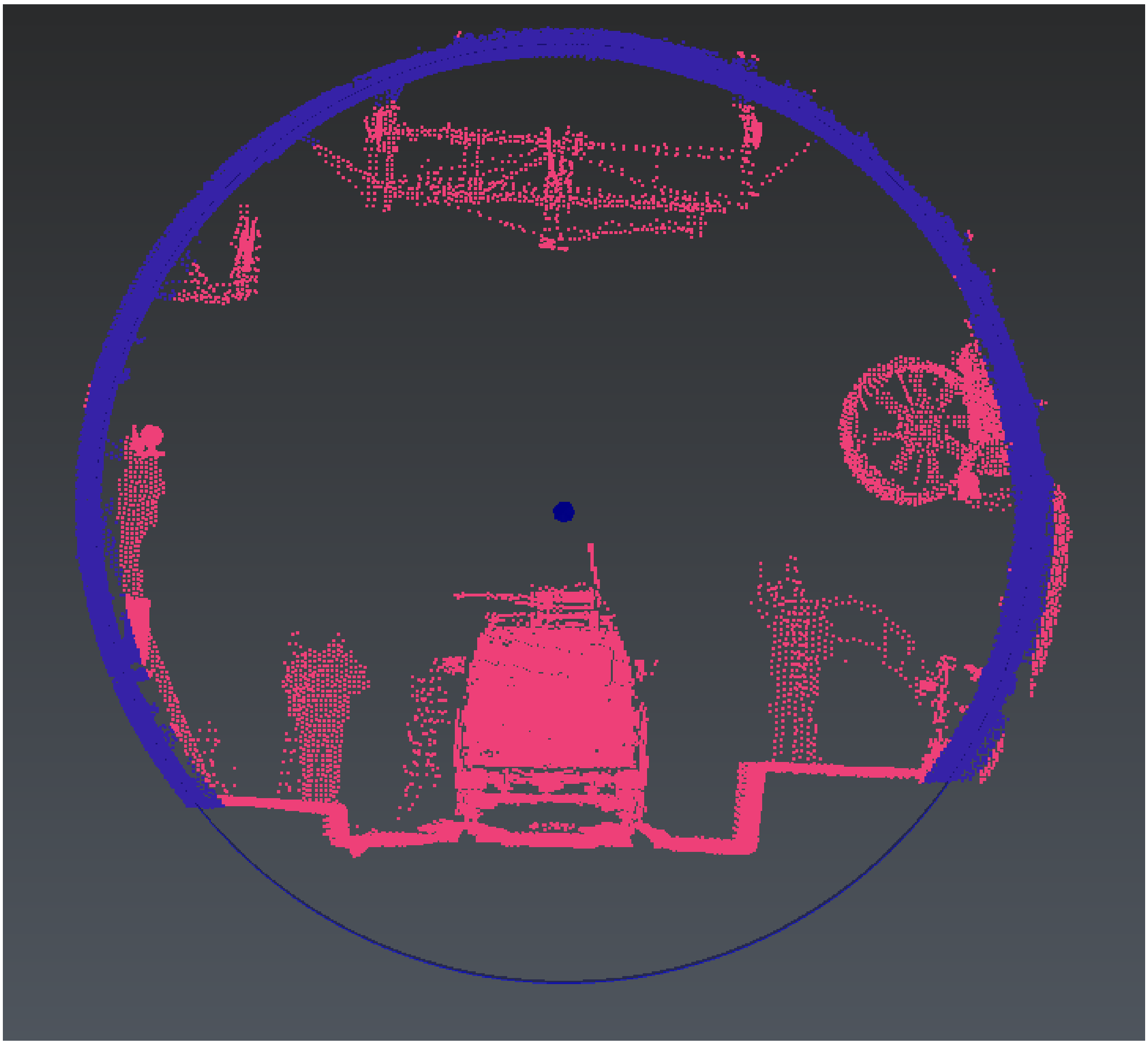
Use an object

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 Separate according Distance.

Set the **Distance threshold** to 0.2m. 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 uncheck the option **Points far from the object** to directly delete the points farther than the given distance, or check both options to divide the cloud in two sub clouds.

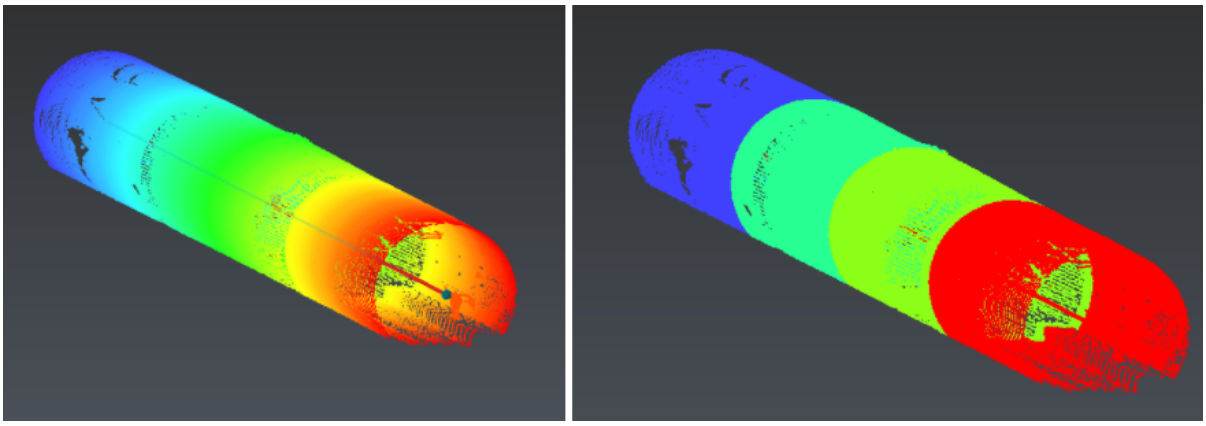


12 Separate a cloud with an object

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 color 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 Along Direction. Choose the **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, in **Presets**, choose the **Regular Steps** preset and set the **Number of steps** to 4 and Click **OK**. Click **OK** to validate the color map. Click **OK** again to validate the colored cloud.



13 Color a point cloud along a direction
Now we can explode the point cloud according to the colors we have just set. Select the point cloud and go to Inspection Steps. The four sub clouds are added into the Measure Group and you can work on each one separately.

5.3.2 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

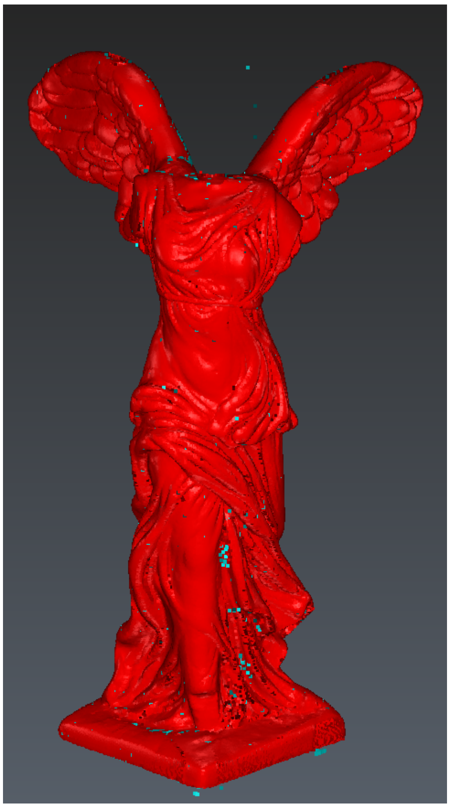
Open the file **Victory.3dr**

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 Noise.

If the goal is to delete sparse points inside the cloud, don't check the **Noisy points** option. 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 **intensity** to 80 in order to remove points which are in fact measurement errors (blue points in the picture). Information appear 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.



14 Remove noisy points in a cloud

Automatic Segmentation: Distance

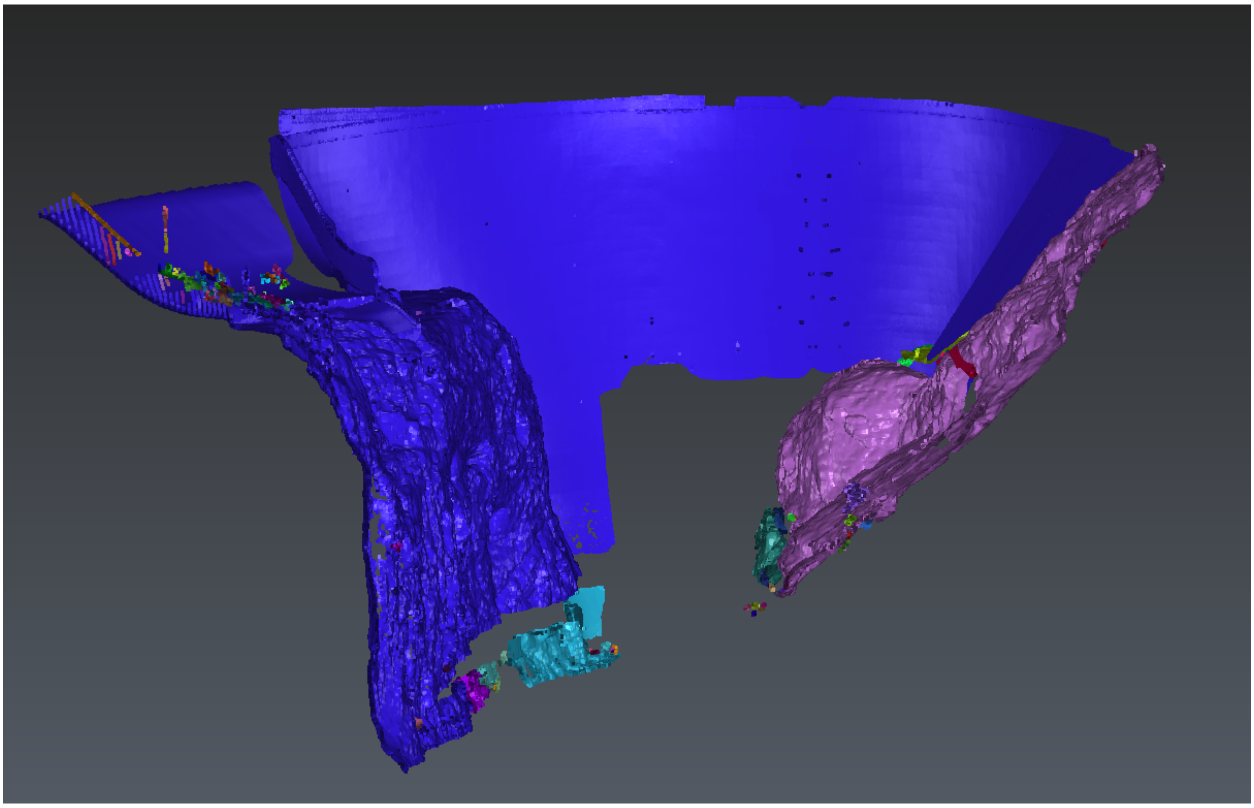
Open the file **CleanPointCloud.3dr**

This file contains the point cloud of a dam having noisy and undesired parts. Select the cloud named “DamRock” and go to 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 the **Restore default value** button and preview the result.

Then, you can change **Segmentation distance** value to fit your needs. Set the parameter to 0.4 m. You can delete automatically the small clouds with the option **Filter small sub-clouds** of less than 10 points. Click on **Preview** to preview the results and **OK** to validate them.

Check the displayed results: Clouds have been exploded into 898 sub-clouds. Then, 789 sub-clouds have been deleted.



15 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.

5.3.3 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: Reduce
- By keeping best points evenly spaced: Resample

Load the file **CleanPointCloud.3dr**

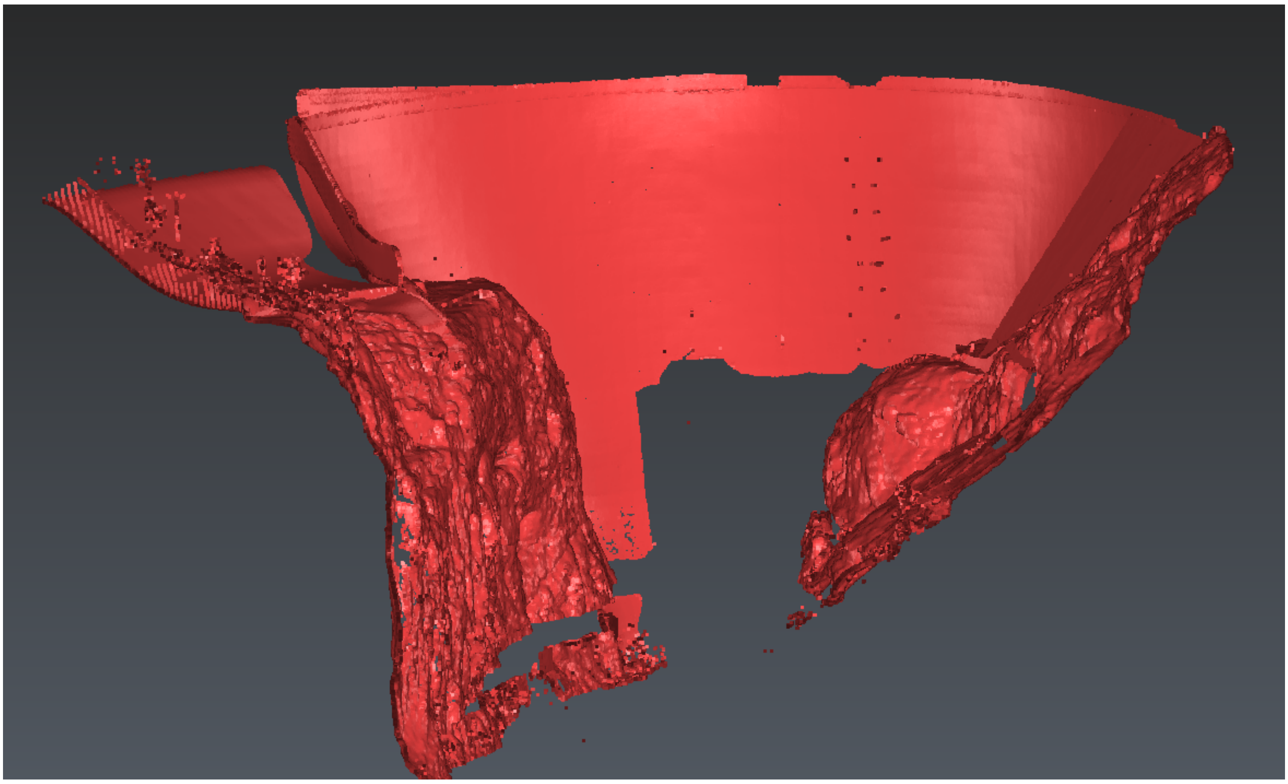
Reduce (Keep a certain number of points)

Select the cloud "DamRock" and go to Reduce.

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.

1541447 points will be kept; 60.0% of 2569079



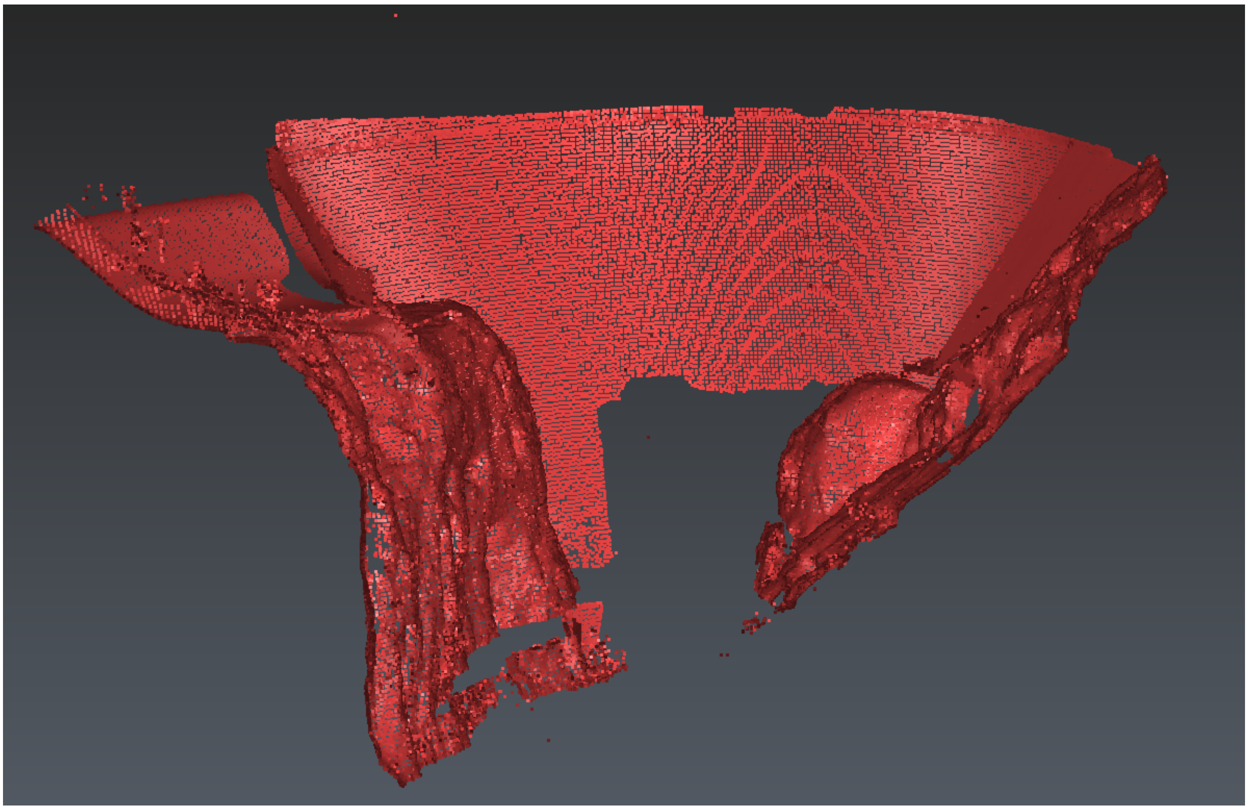
16 Reduce a point cloud keeping a certain number of points

Resample (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 **Max distance.**

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

91931 points will be kept; 3.6% of 2569079



17 Reduce a point cloud keeping best points

5.3.4 Exercise: Separate walls and floors

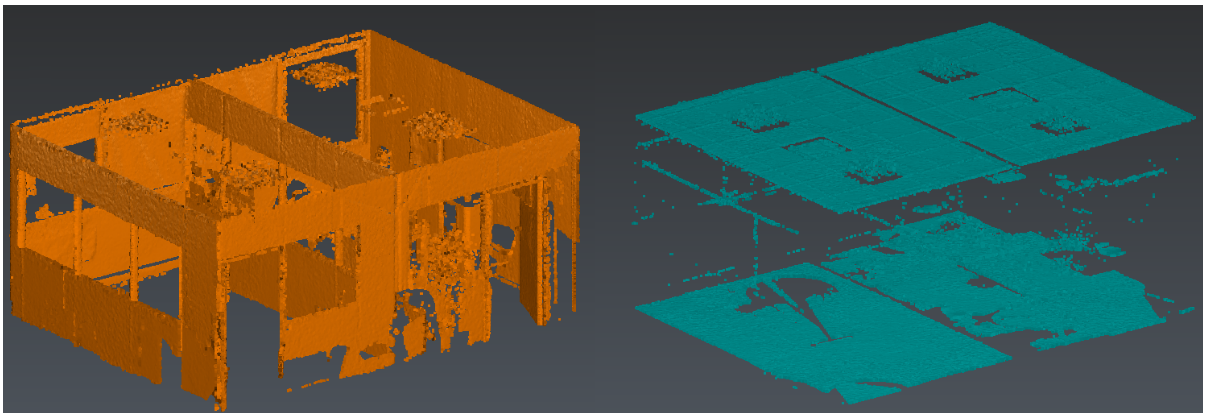
Open the file

Open the file **ClashAnalysis.3dr**.

The file contains the point cloud of an office which has been scanned with a Leica BLK360. For instance, to make an inspection either of the walls or the floors, we need to keep only the corresponding points. This can be achieved fully manually or can be prepared thanks to algorithms.

Split the cloud

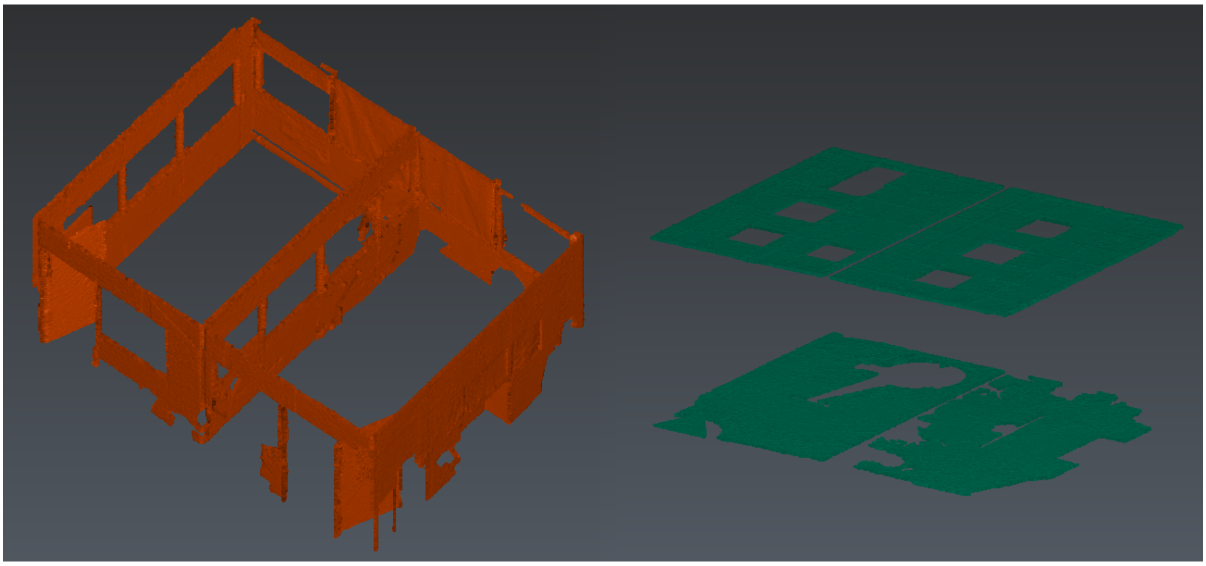
Display and select only the cloud, then launch the command Walls and Floors. The angular tolerances can be adjusted to eliminate more or less points. Let both tolerances on 5°. You may also set a higher local normal smoothing to get a smoother result.



18 Walls and Floors segmentation

Explode by distance

Then, select the result that corresponds to the walls (or to the floors/ceilings) and launch Distance with a **Segmentation distance** of 0.02m and **Filter small sub-clouds** with less than 10000 points.



19 Distance segmentation
Optionally, complete manually the cleaning with Clean / Separate.

5.3.5 Exercise: Classify a cloud manually

Open the file

Open the file **Fountain.3dr**.

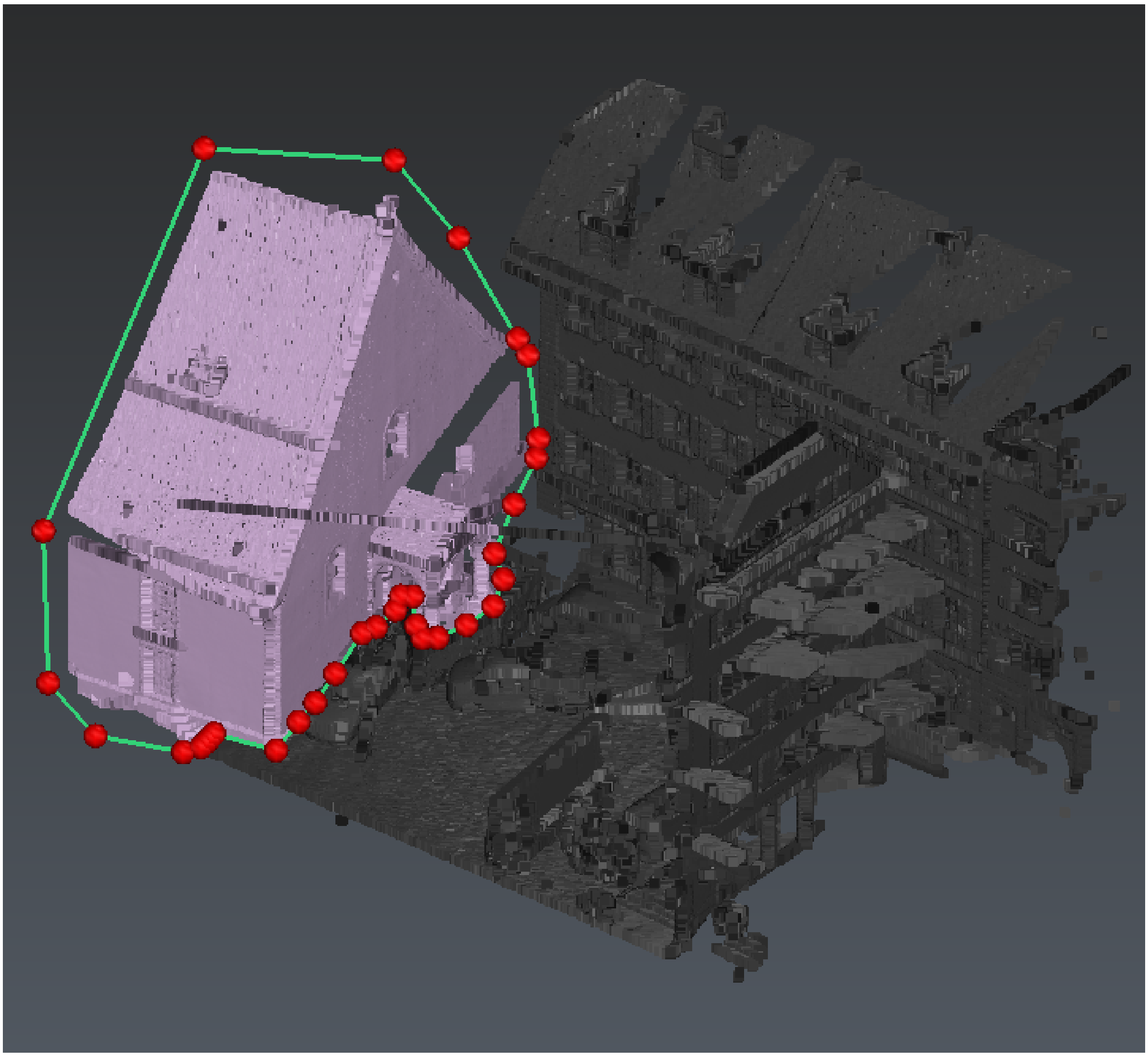
The file contains the point cloud of a part of a city, with building frontages, vehicles and people. We will use manual classification tools to classify the points.

20 The scan of a city

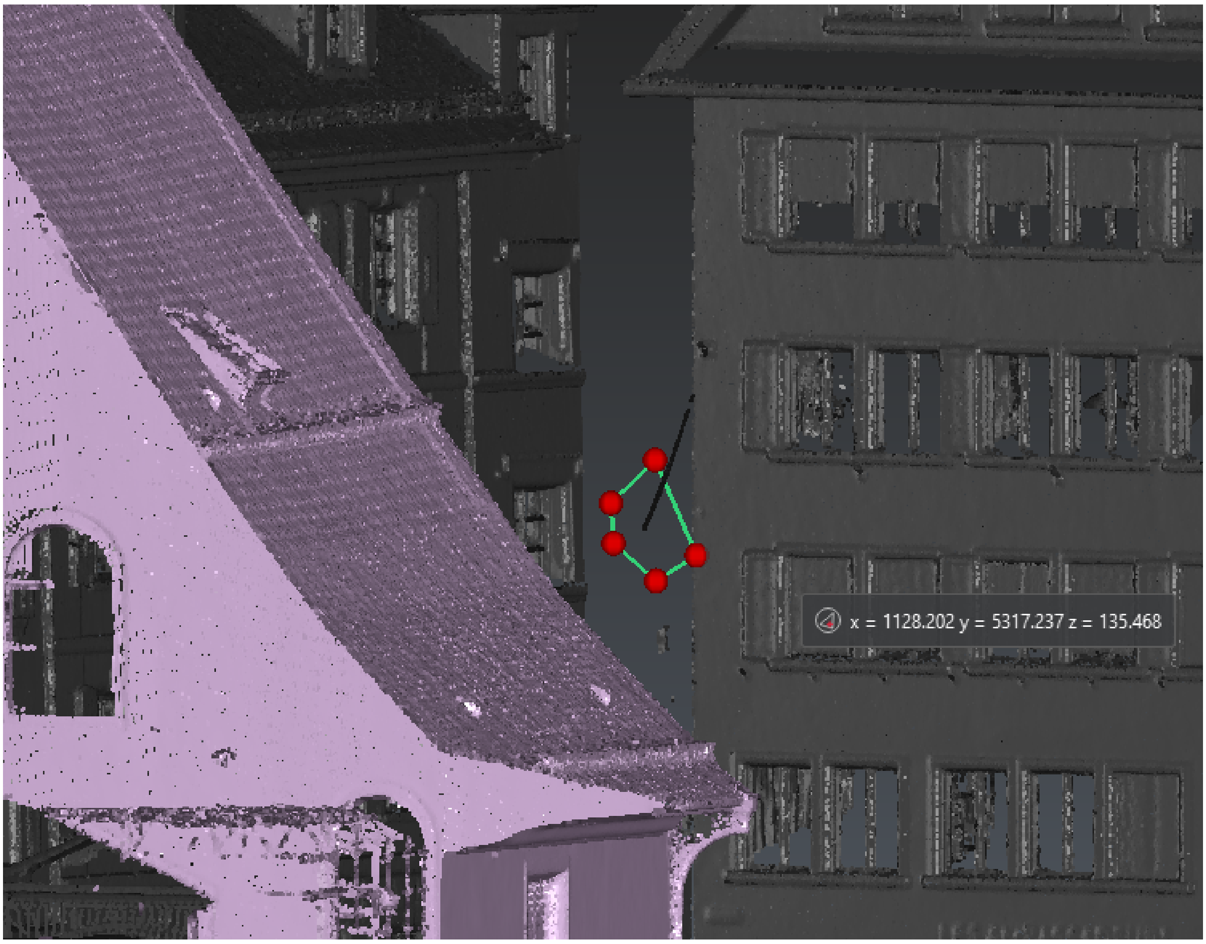
The toolset resembles the one from Clean / Separate. The principle is close: instead of splitting cloud(s) in parts, it associates each point with a class. This cloud has no classification data, so the **Classification** representation is not available yet.

Select the cloud and go to Manual Classification.

All points are classified as **Created, never classified** for now, and are displayed in dark gray. Start by selecting the main part of the church.



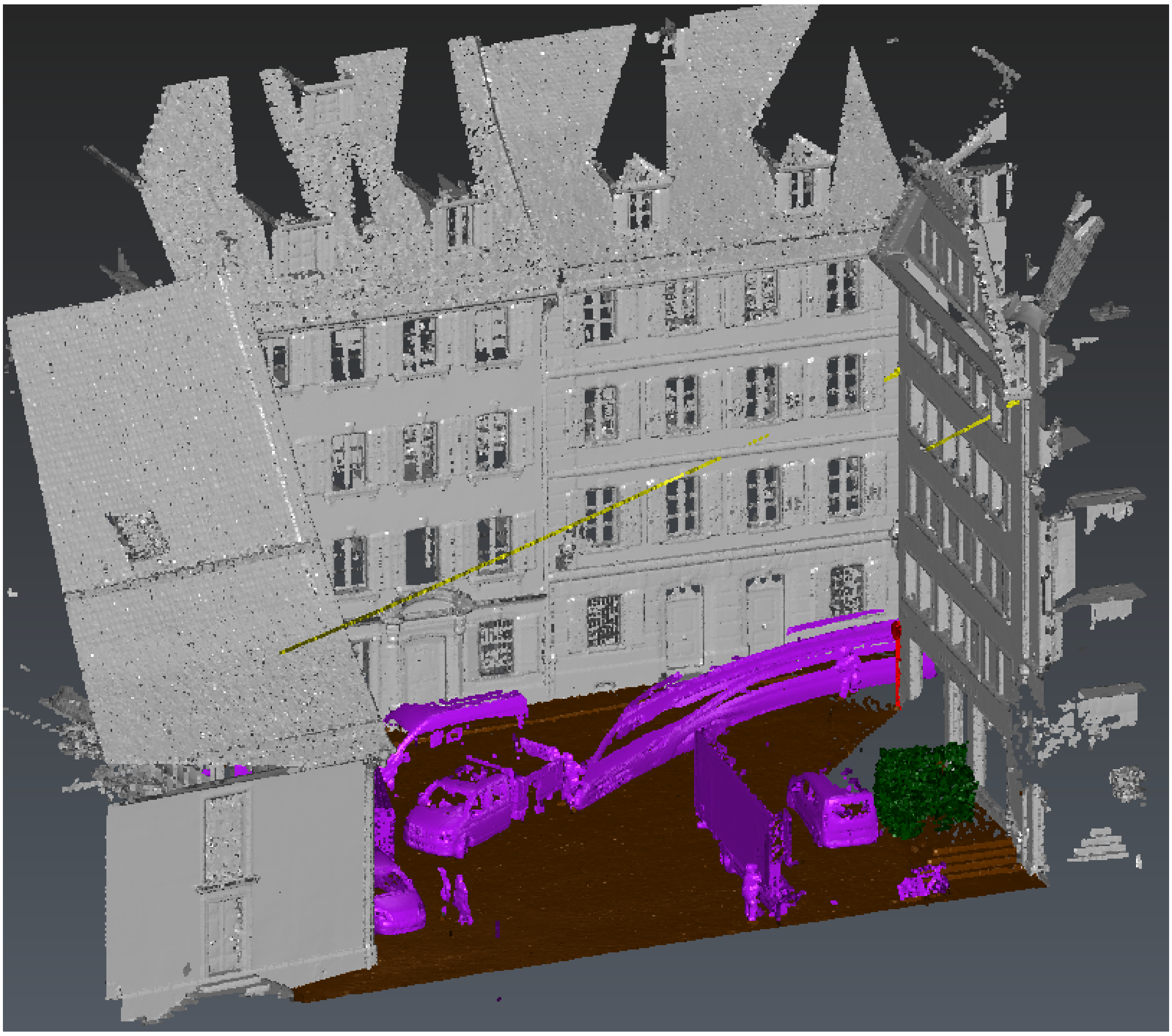
21 Select points to classify them
Just like in Clean / Separate, you can change the depth of the polygonal selection to take only a part of the selection. Then, you will probably have to refine the selection. For instance, you can remove the wire from the selection.



22 Remove the wire from the selection
Click **Add in Class** and select **6 - Building** from the list. Selected points are now colored in light grey.

You can hide the classes already used for a better visibility of **Created, never classified** points. You can also lock them to avoid inappropriate selection.

- Select the other frontages, assert the active class is **Building** and click **Add in Active Class**.
- Some points are probably still floating in the air. You should identify two wires. Add them to the class 14 (**Wire - Conductor**).
- Next to the right building, there is a tree to classify in 5 (**High Vegetation**).
  To help in the classification, you can temporarily switch the representation of the **Created, Never classified points** to **Real Color** by clicking on the disk icon.
- Select the road sign, also next to the right building. There is no default class for it: let's make one.
  - Click **Edit Classes**,
  - Double-click on one of the custom ones, for example 76,
  - Edit its name **Road Sign**, color and mark it as favorite.
  - Now click **Add in Class**: your custom class is on top of the list, select it.
- Select the ground and add it to 2 (**Ground**).
- Cars, bicycle, people and the fence (next to the church entrance) can be added to 65 (**Scanning Artifacts**).



23 Result
Now, you have classified your cloud, you can use Explode By Class to make each class become a separate cloud, making it easier to mesh buildings or the ground separately. Or simply to avoid meshing scanning artifacts.

Later, exploded clouds can be merged without loss of class information.

6 Alignment - Registration

6.1 Align clouds together

If you have not done so beforehand, 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 contain 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)
- Exercise: Align clouds according to specific points (from geometric features)

6.1.1 Exercise: Best fit between clouds with overlapped area

Open the file **BestFitClouds.3dr** and run Elevation View twice.

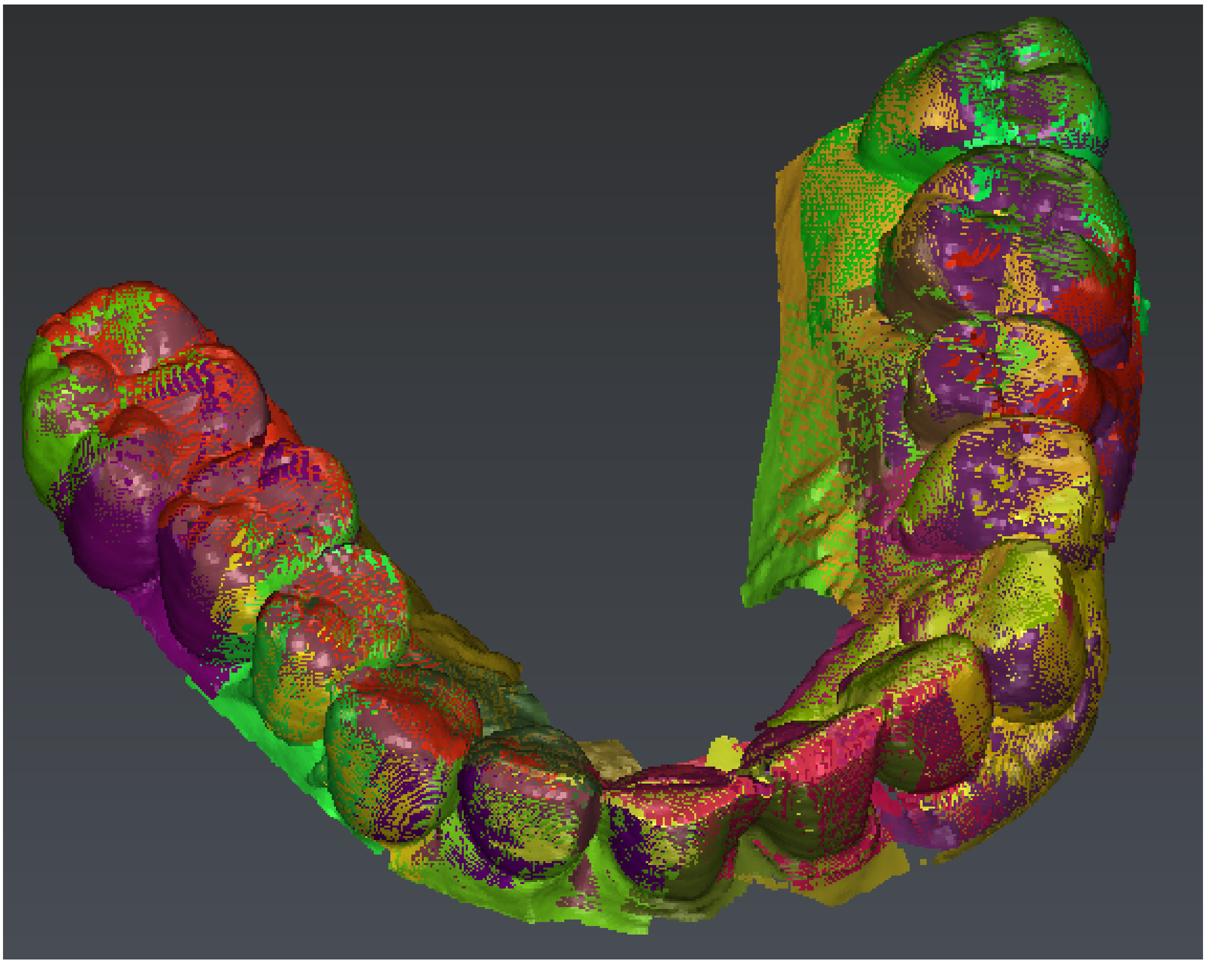
This file contains 32 clouds with overlapped areas. If you zoom in, you can see that all the clouds are not correctly aligned.

Select all of them and then go to 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 bottom (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.



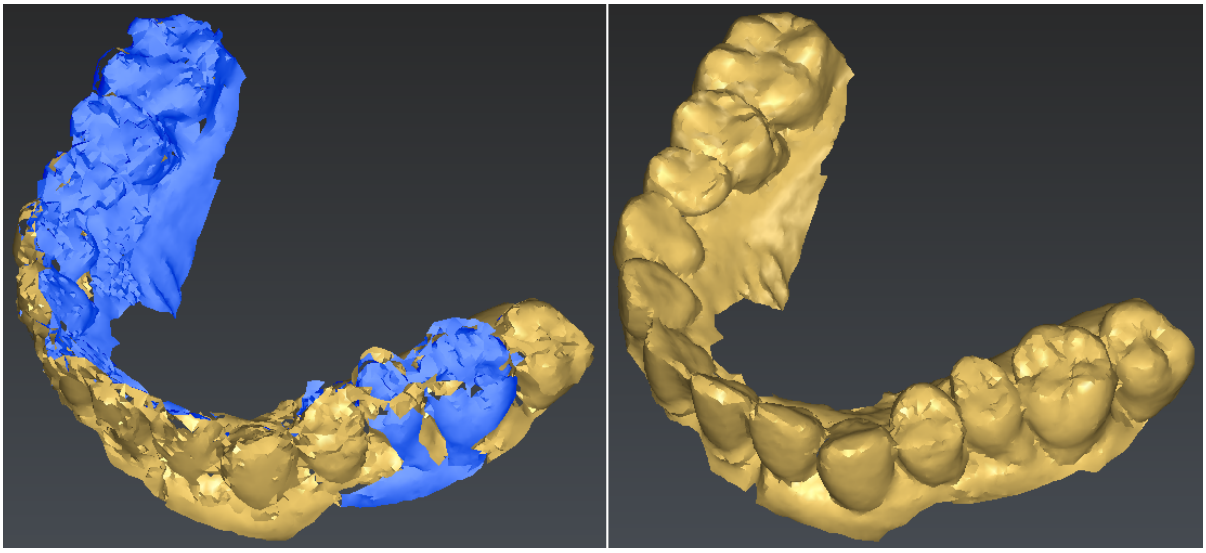
24 Result of best fit

> **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.



25 Comparison between two meshes with/without best fit alignment

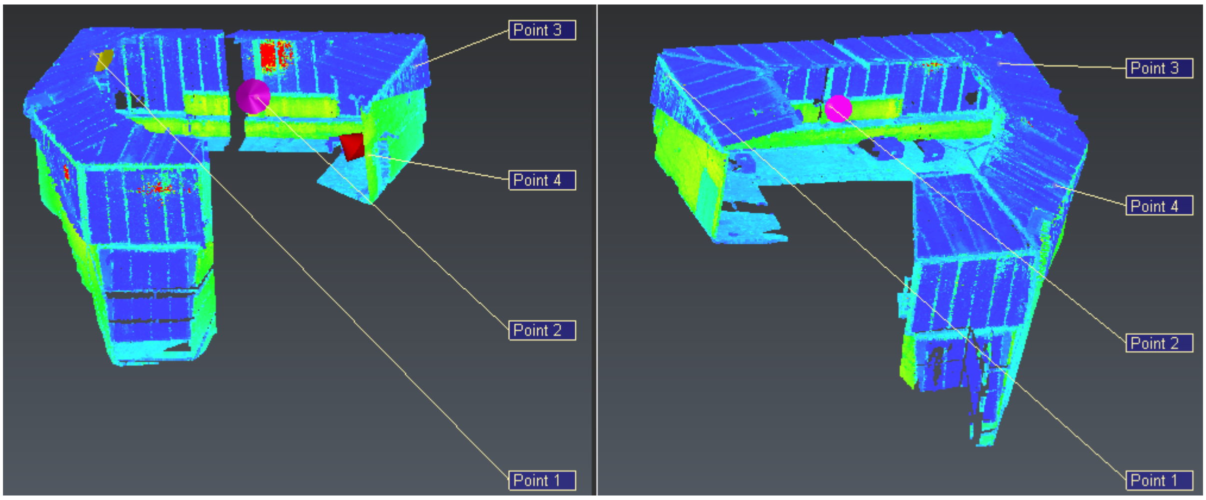
6.1.2 Exercise: Align clouds according to specific points (surveying target centers)

Open the file **AlignTargets.3dr**

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 N Points Registration.

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 bottom toolbar, select only the option **Surveying target** and set a **checkered pattern target** with a **Diameter** of 0.15.

In the **Define constraints** section**,** select the option **Constraint rotation axis** and choose Z as the Z axis is aligned with the vertical in both clouds. Uncheck the others options to allow translations along the three axes.

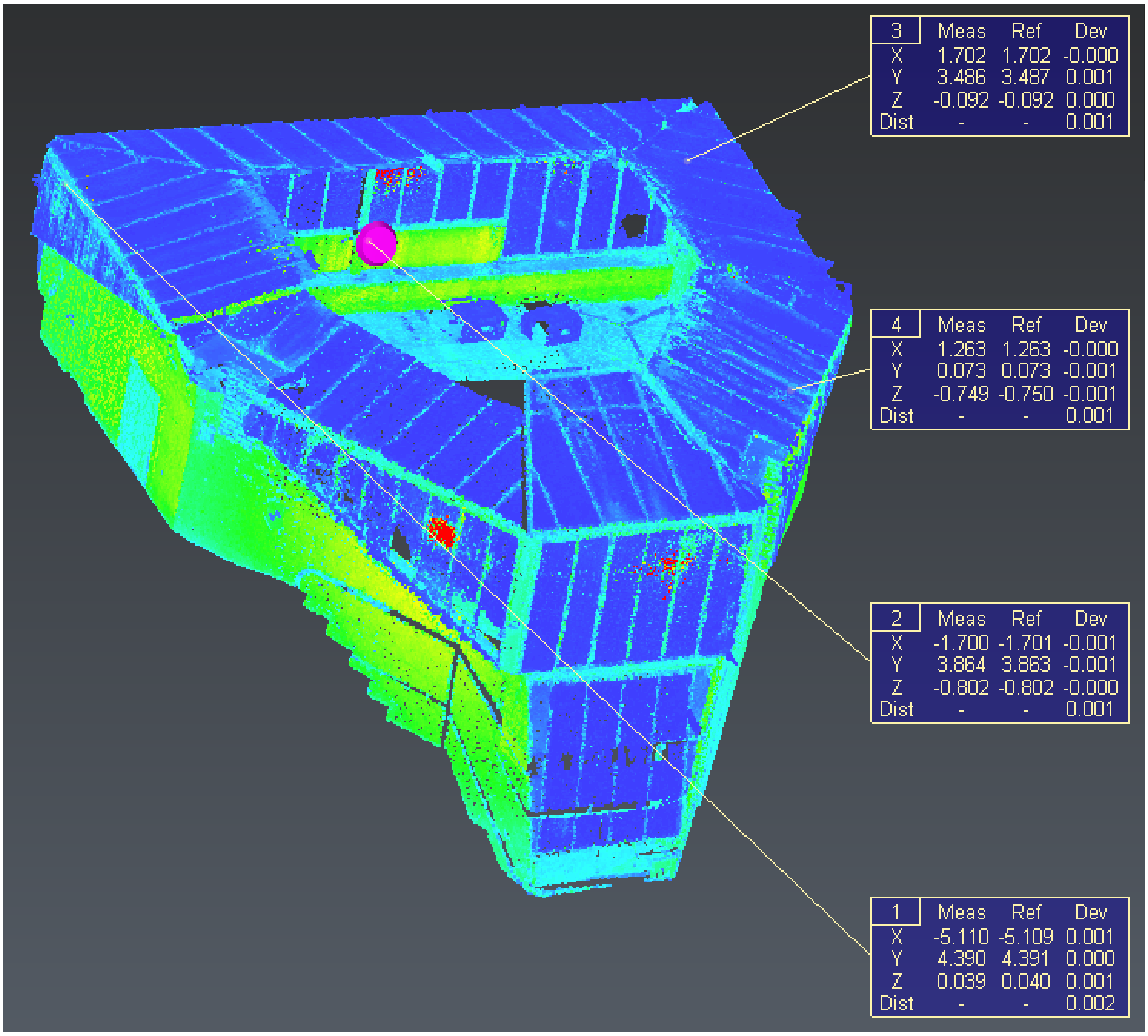


26 Use target centers to align point clouds

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 Bestfit** as we want to do an alignment according to targets only.

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



27 Labels to show deviations during the alignment according targets (less than 2mm 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 point 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 done with only 3 couples of points.

Click **OK** to validate. Both clouds are now aligned: you can use the command Merge 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 **Inspection** representation then go to Edit Colors to take only one color level.

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

6.1.3 Exercise: Align clouds according to specific points (from geometric features)

Open the file **AlignRefPoints.3dr**.

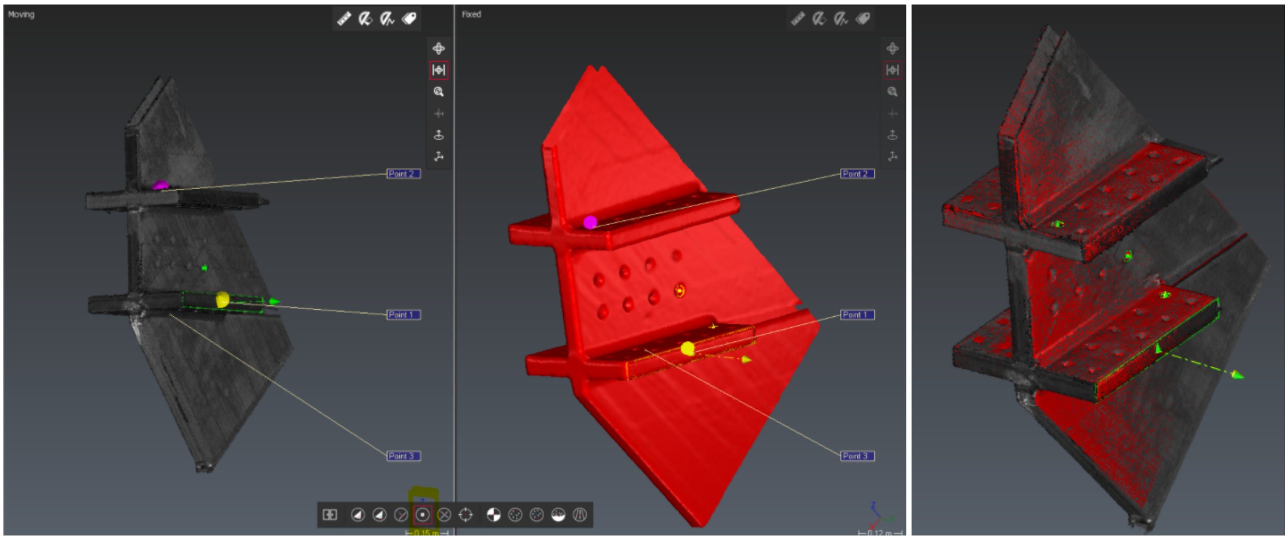
This file contains two clouds of the same object, for instance scanned at different times. There are also 2 groups of geometric characteristics respectively extracted from each cloud.

The clouds need to be aligned together by the leap frog technique. To do this, we will use the command N Points Registration, using the center of each geometrical shape as reference (it could also be points measured beforehand).

First, select the movable objects: for example the **Cloud1** and all objects in **Geometrics on Cloud 1** group. Then, launch the command N Points Registration.

In this command, you can click on the corresponding points of each element: the moving one and the fixed one. At least 3 couples of points need to be clicked. For each couple, click a point in the left view and its corresponding point in the right view, as in the following picture.

Select the circles, and/or the plane, **Middle/Center** point in the selection toolbar.



28 Points definition in Best align N points

Then, click on **Preview**. You will obtain the following alignment:

The distance between the same points scanned varies from 0.3 mm and 0.9 mm. This value is quite good for the measuring conditions and of course strongly related to the precision of your measurement device.

You can use as many points as you want. You will only have a more constrained system.

If you are satisfied with the error between points, validate the alignment with **OK**.

Be careful, the moving objects are moved in the 3D space. So, your initial data is put in the trash. If you want to keep it, make a copy before doing the alignment.

Same exercise with a CAD model

You can also make the alignment directly with the similar compound CAD model, located in the CAD group. Here, there is no need to extract geometrical features on the CAD model. You can directly select the middle / center points just by hovering over the CAD model, near the corresponding geometric area.
You can also refer to the Exercise: Align clouds according to specific points (from probing)

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 Z axis orthogonal to the wall.

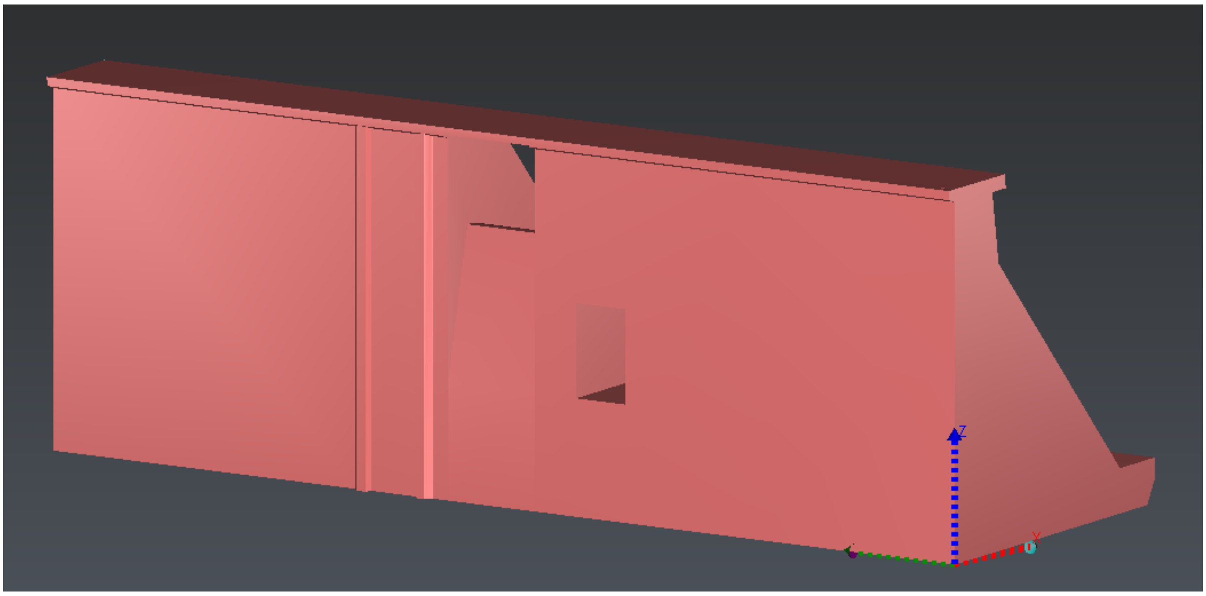
- Exercise: Move to the Coordinate System (CS) of a 3D model

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

Open the file **BestFitOnRef.3dr**.

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 Geometric Registration.



29 Move to the coordinate system of the dam

You must follow 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 Geometric Registration, the selected objects will moved to the new position. The 3D coordinates of the object are updated. This command differs from Define UCS 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.

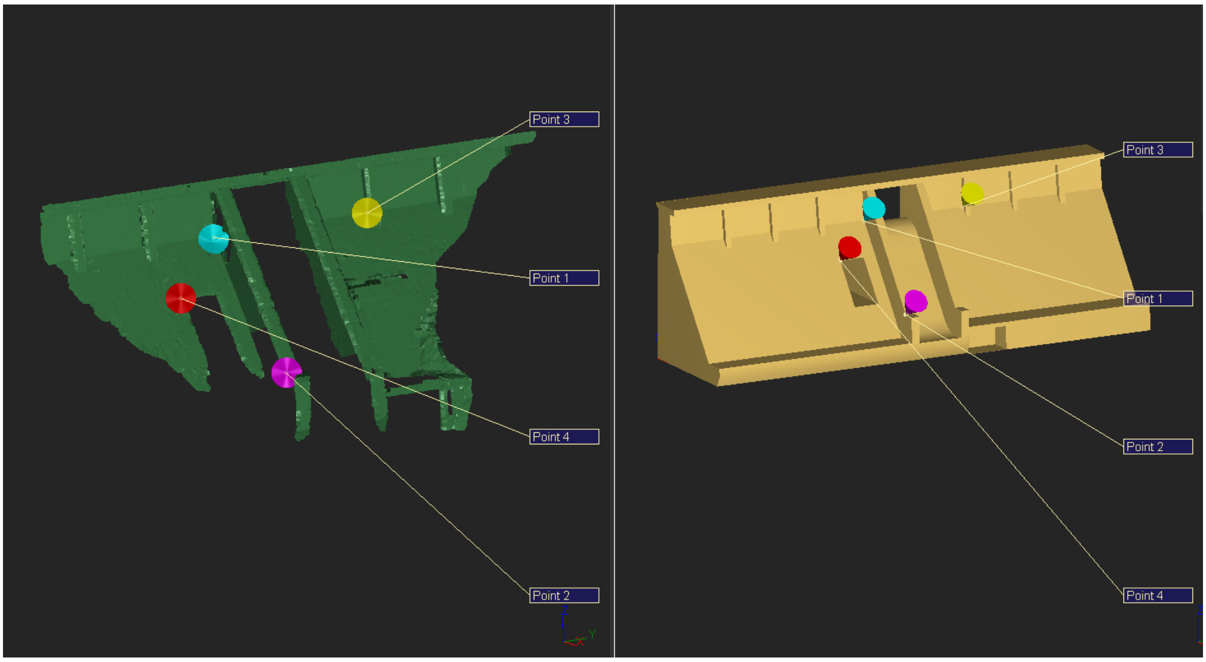
- Exercise: Align a point cloud on a reference model according to the shape (Best Fit)
- Exercise: Align clouds according to specific points (from probing)

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

Open the file **BestFitOnRef.3dr**.

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 N Points Registration.



30 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.

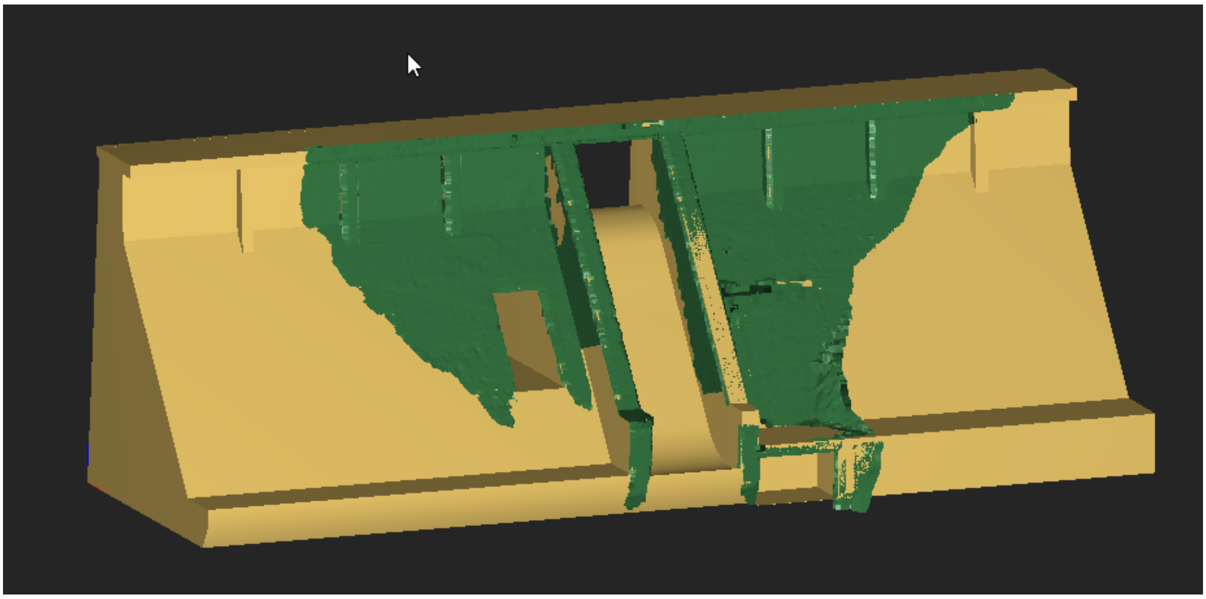
Select the option **Compute new alignment** in the list. Then, enter couples of points in order to do a rough alignment by combining a point on the cloud and a point on the mesh. To realize this step, select the option **Vertex End**. First, click on a point on the cloud and then the corresponding point 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, check **Apply Bestfit** and click **Preview**. The software has now computed a best fit (using the rough manual alignment that you can check thanks to one label per couple of points that is also created in order to see deviations). Next to the dialog box a small report is displayed in order to summarize all the transformations.

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 **Reset** the computation and disable some pairs.

> **Note**
>
> You can define some constraints during this alignment. For example, if the Z axis is correct on the cloud, select the option **Constraints rotation axis** and then choose the option **Z**. 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.



31 Cloud and mesh aligned with the best fit method

6.3.2 Exercise: Align clouds according to specific points (from probing)

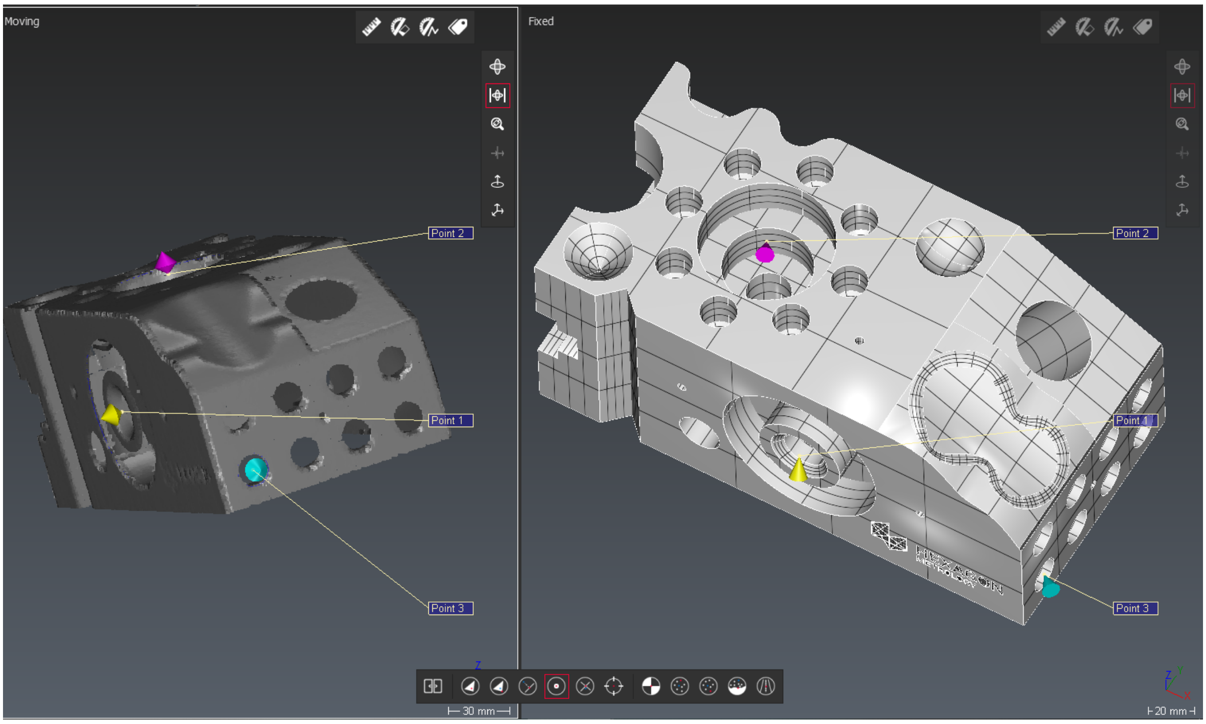
Open the file **AlignOnRef.3dr.**

In this file you can find a point cloud **Original CloudToAlign**, 3 circles (**Circle 1**, **Circle 2**, **Circle 3**) and the CAD object **H009357_revA**. The 3 circles have been measured with a probe to have a better precision for circles than what you can get for the point cloud scanned with a laser.

The goal of this exercise is to align the point cloud on the reference thanks to the 3 circle's centers.

Show the cloud, the 3 circles and the reference, then select the circles and the cloud and go to N Points Registration. The screen will be divided in two parts, on the left side you have the cloud and the 3 circles (will move), on the right side the reference object (will not move).

Set the option **Middle / Center** and select the center of the 3 measured circles (left side). Do the same but directly hover over the CAD model the corresponding point to the measure, on the right side as follow:



32 Point picking for alignment
Click on **Preview**. The cloud and the 3 circles are now aligned. You can see that the distance is around or less than 0.1mm.

As probing is more precise than scanning. Where necessary it is always recommended to align the part with points coming from probing rather than using a best fit on a point cloud (using matching points). It is the best way to have a very precise alignment.

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: Scan to Mesh
  - Exercise: Create a 3D mesh of the Samothrace Victory
  - 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

7.1 Mesh creation

This section shows the difference between meshing commands.

- Exercise: Scan to Mesh
- Exercise: Create a 3D mesh of the Samothrace Victory
- Exercise: Extrude a profile

7.1.1 Exercise: Scan to Mesh

The goal of this exercise is to create a textured mesh from a cloud in one step with predefined settings.

The Scan-to-Mesh feature has been designed mainly for Leica Geosystems scanners but can be obviously used for any kind of point clouds.

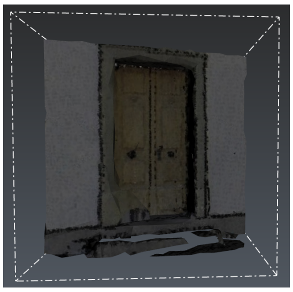
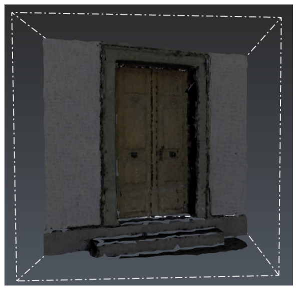
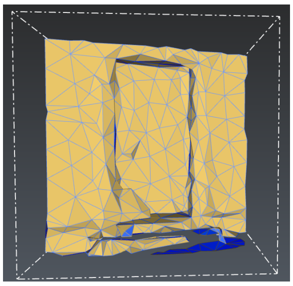
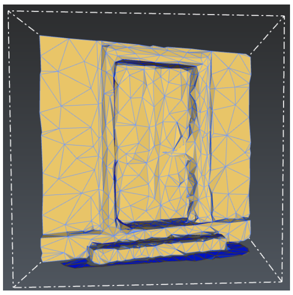
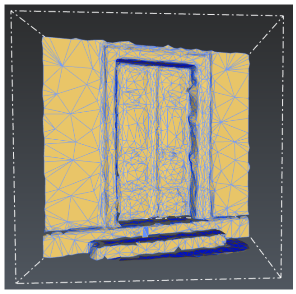
The main interest of this feature is to straightforwardly give realistic models; especially for indoor applications like buildings or plants.

Open the file **Fountain.3dr.**

Select the cloud and launch Scan to Mesh.

Activate the **Evaluation mode** and click on the church side door. A small temporary mesh is created where you clicked, allowing you to get a preview without computing the whole mesh.

This sample is regenerated after each setting change. If necessary, press **DEL** to remove the current evaluation box and sample another part of the scan.

	Low	**Medium**	**High**
With texture			
Without texture **(Flat + Wire representation mode)**			

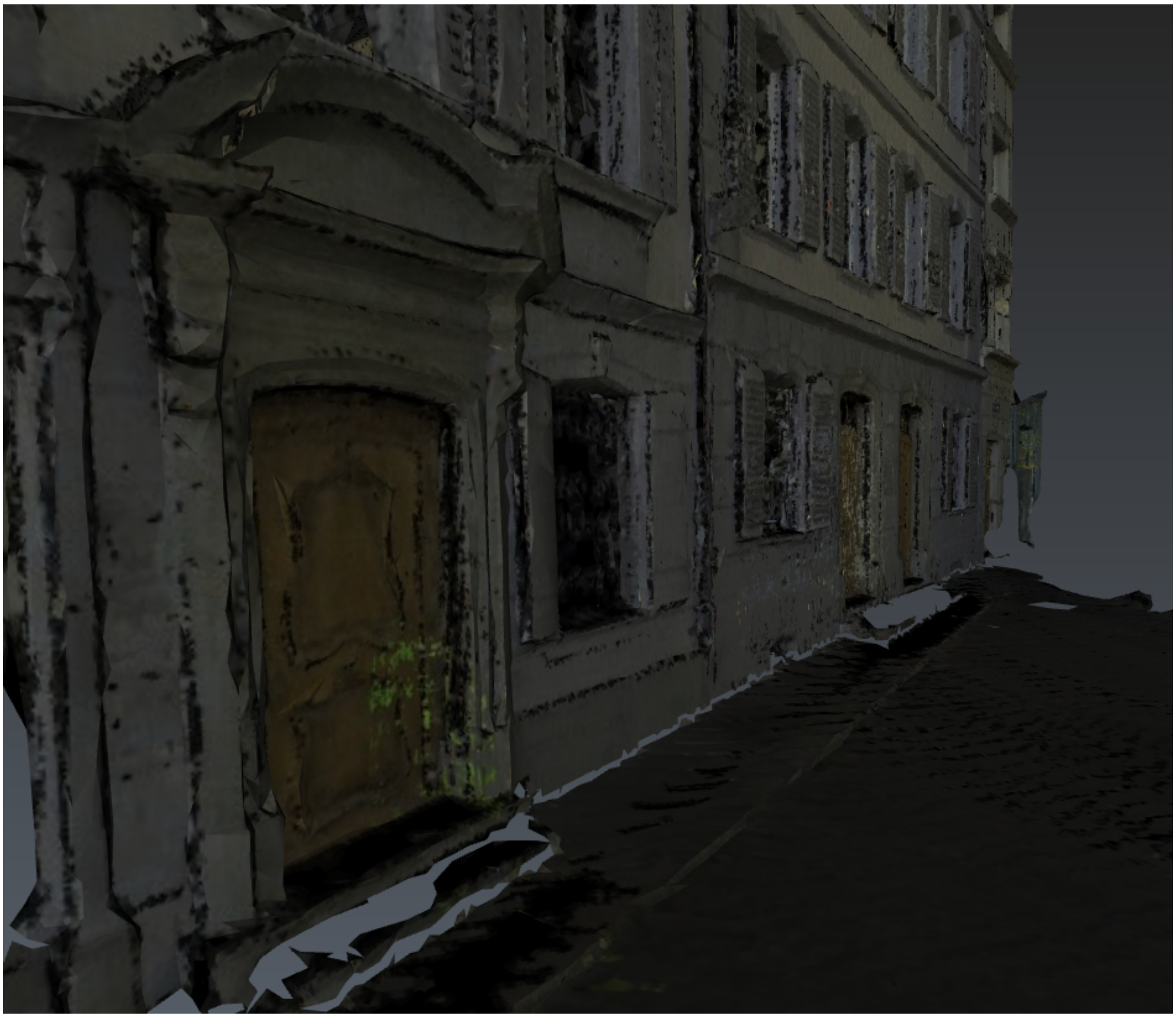
The evaluation mode is suitable to tune the mesh according to your needs and the computation time to spend.

Here, you can use the presets, or define your own custom mode.

Keep the option "Ignore scanning directions" unchecked.

By default, a texture is created using the point cloud real color.

When your settings are fine, deactivate the **Evaluation mode** and click **Preview**: the cloud will be meshed and textured.



33 medium result
The medium level seems here to be the most suitable level of details. The high level is often more optimized with a denser point cloud.

> **Notes**
>
> To enable the meshing of large datasets, the computation will be split in overlapping boxes: you can edit the box size in advanced settings.
> Moving objects are filtered by the same mechanism as Moving Objects. The scanning directions option may impact the result of the Moving objects filter (if this information is available in the cloud).

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

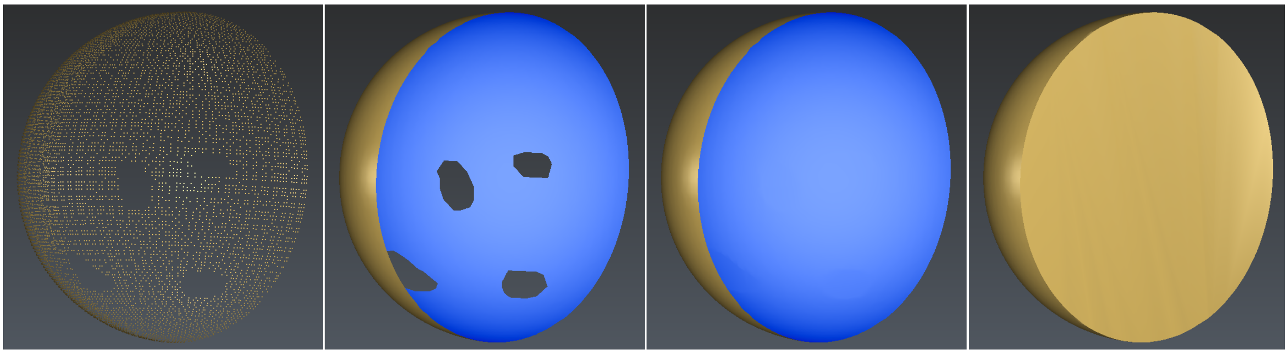
> Open the file **Victory.3dr**.

This file contains two point clouds: one **Victory + noise** with some measurement noise, and one **Victory** already filtered with the command 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).



34 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.

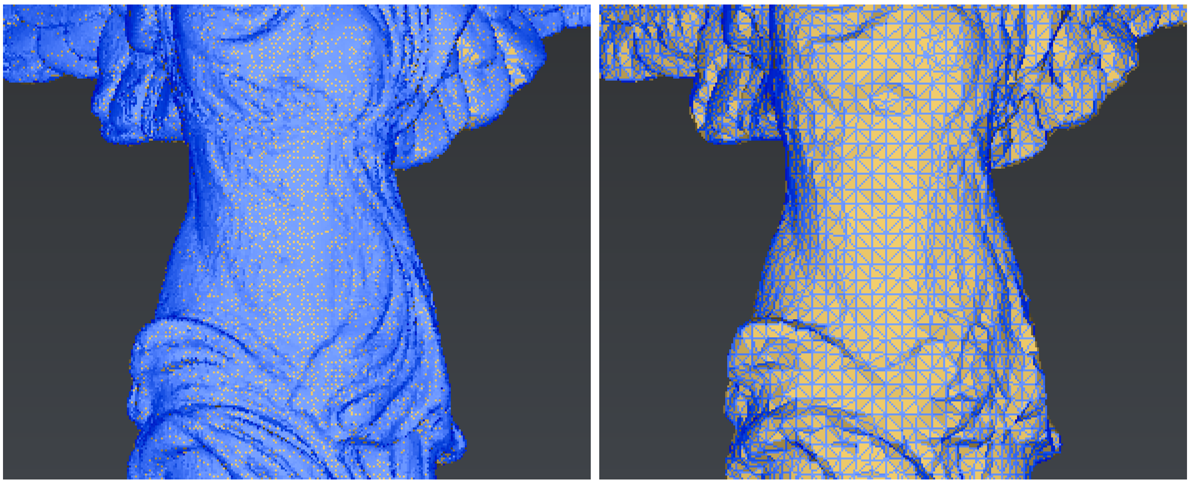
Regular Sampling

Select the cloud **Victory** and launch the command 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 triangles shapes. You will notice that they are quite regular and equilateral. The value set for the **Average distance between points** roughly determines the size of the grid projected on the cloud. In this case, the value is approximately 0.6 mm and corresponds to the average distance between vertices.

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



35 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, or
- 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 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.

There are three **Refining methods**:

- **Refine from Cloud Interpolation**; if your point cloud contains a lot of points and/or noisy points (measurement errors), it is strongly advised to interpolate new points.
- **Refine from Cloud**; will give you better results if the point cloud contains only precise points and if you want to preserve sharp edges.
- **Refine without Cloud**: this method does not use the point cloud, it is not useful in this workflow.

For this exercise, only the first two methods will be used and explained.

Select the option **Refine from Cloud Interpolation** 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 mm**, it means that the maximum distance between the mesh and a "perfectly smooth" surface will be less than 0.05 mm. There are 2 other parameters in order to control the refinement:

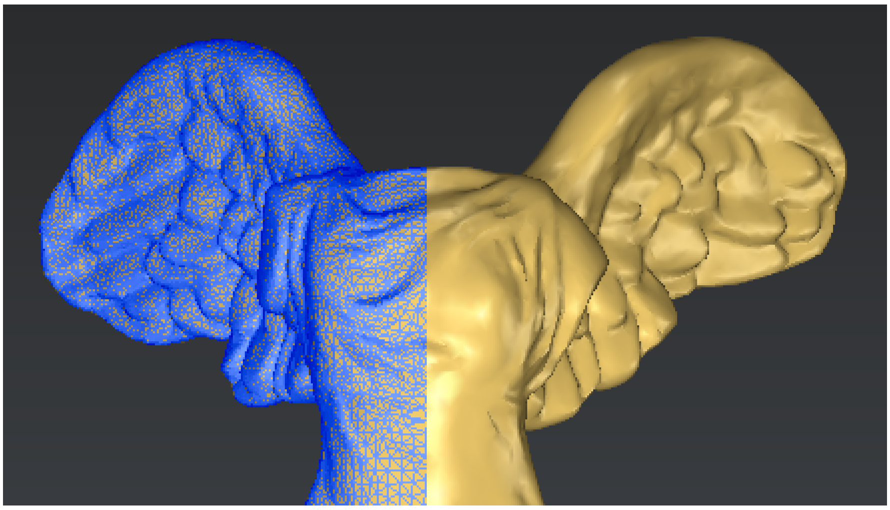
- **Maximum number of triangles**; in order to avoid having too many triangles in the mesh. Enter 1 million.
- **Minimum triangle size**; in order to avoid very small triangles in the mesh. Enter 0.5 mm, this value should be bigger than the deviation error.

Set the other parameters as following:

- **Distance**; to reject the points located too far away from the polyhedron you can enter **1 mm**.
- **Local reorganization**; to give a better mesh of sharp angles and small fillets you can select it.

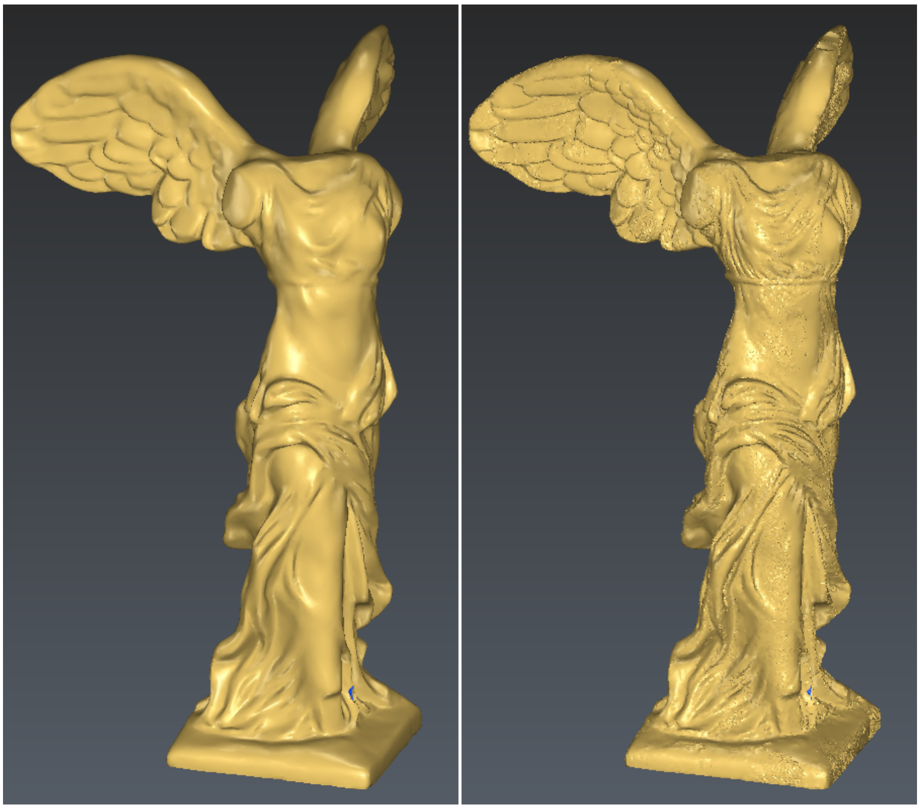
Select **No free border modification**.

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



36 Refine by interpolating new points
Now change the **Refining method** and use the **Refine Mesh from Cloud** option and **Keep only best points** as there is still some noise in the cloud. Enter 0.05 mm for the **Deviation Error** in order to compare with the previous method. Set the **Distance** to **1 mm** and check **Local reorganization**. Check **No free border modification** and click on **Preview** to refine the mesh.

As you can see in the next pictures, 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 **Refine from Interpolation** because it has been reduced during the computation of the new points.



37 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 **Refine from Cloud**.

7.1.3 Exercise: Extrude a profile

Open the file **CrossSections.3dr**.

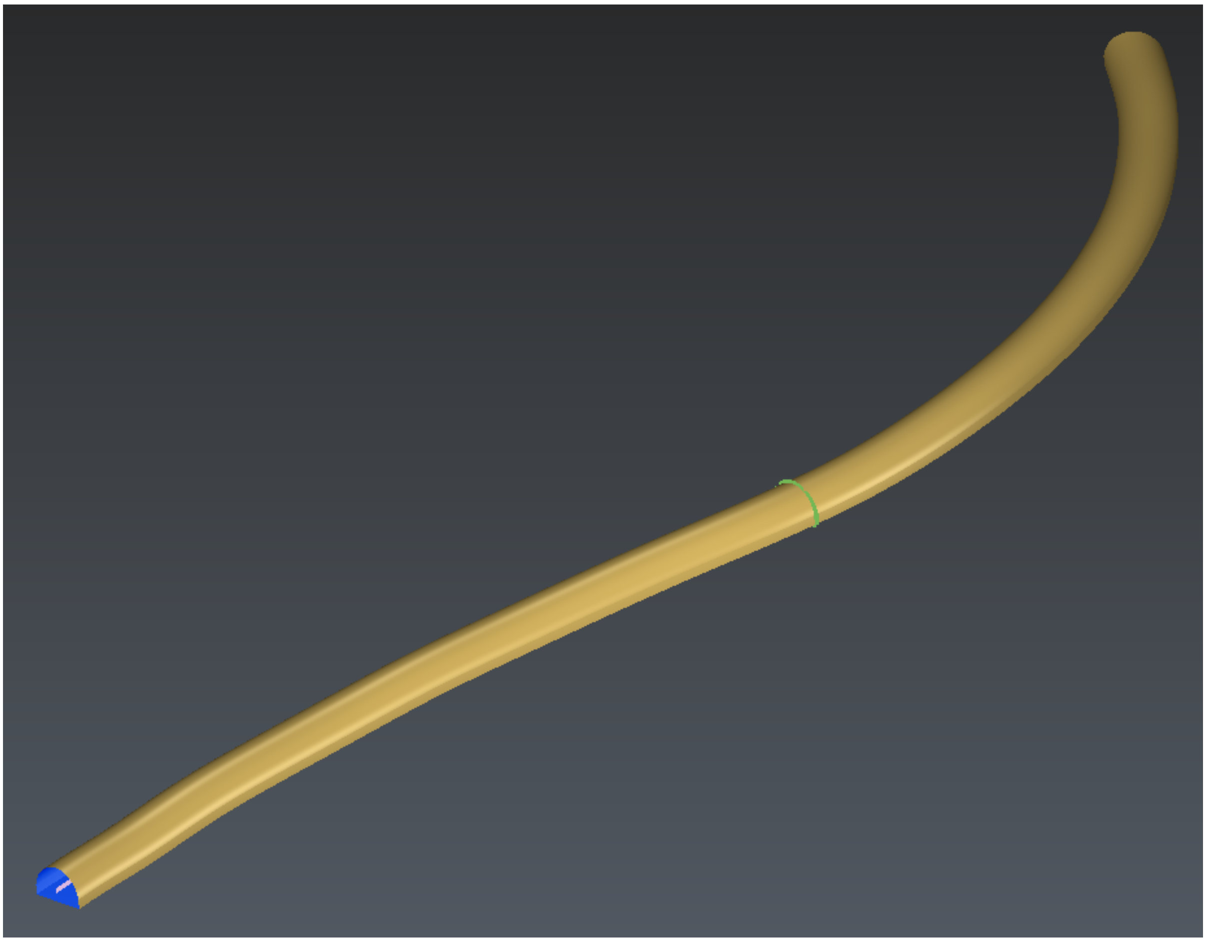
This file contains a theoretical section as well as the neutral axis of a tunnel.

Show only **Neutral axis** and **Theoretical section**.

Select the **Theoretical section** and go to Profile along a Path. 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 **Close extremities** 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).



38 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

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

Open the file **Victory.3dr**.

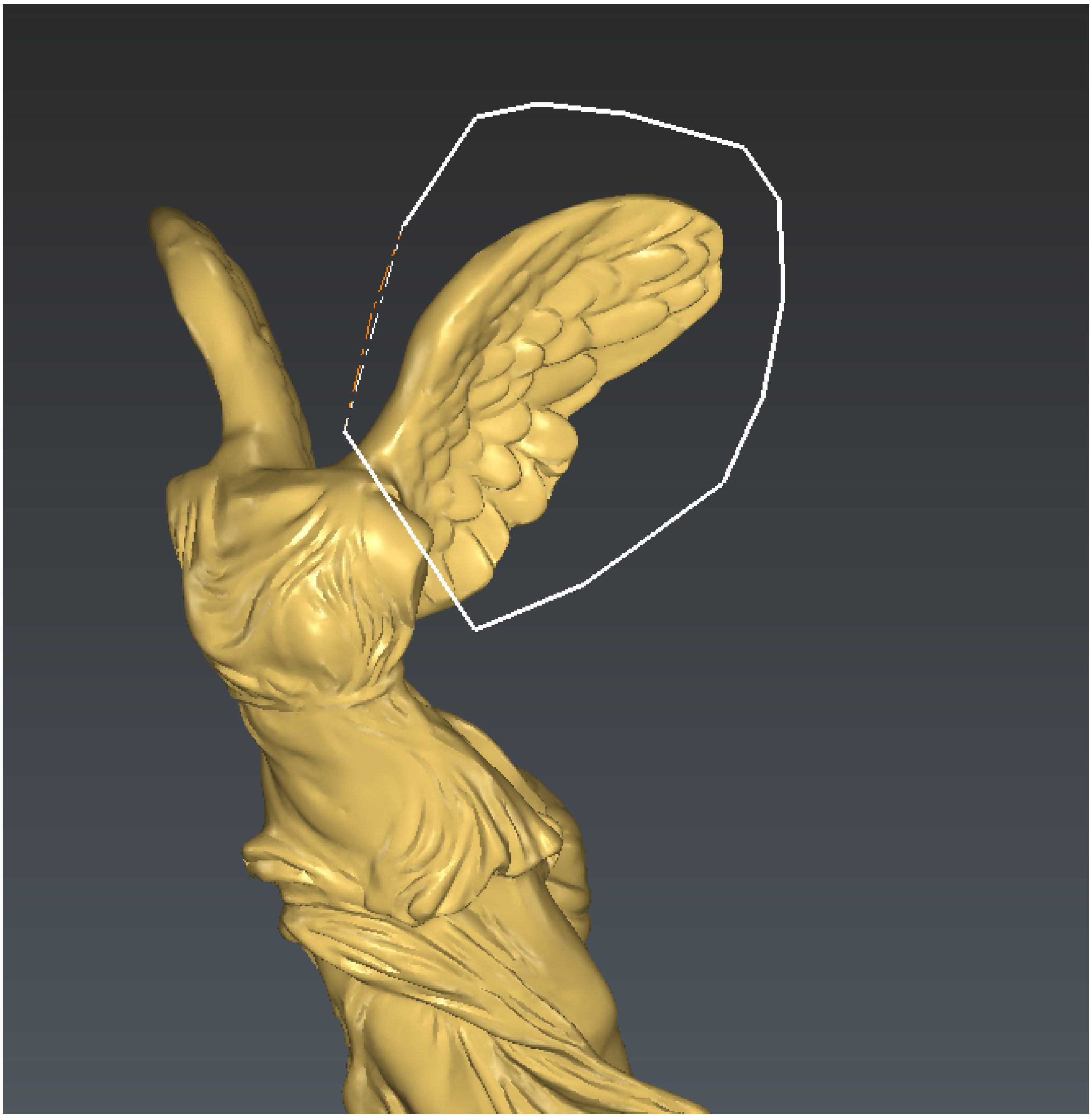
Then show only the mesh **Victory (complete)**. Throughout this exercise, we will see how to cut a mesh and how to work independently on different parts of the mesh (by reducing a part and, on the contrary, refining another one).

Cut / remove triangles

Select the mesh **Victory (complete)** and go to the command Clean / Separate Mesh. 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.

Select **Polyline tool**, and then select the options **Select Through**.

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).



39 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 select mode **Remove from selection** (shortcut "-") to remove triangles from the current selection before validating.

Once the two wings are selected, click **Split** and then **OK** to validate. If you want to delete triangles, click on **Delete**.

Note that all pieces of meshes are grouped into a single mesh by default.

Refine mesh

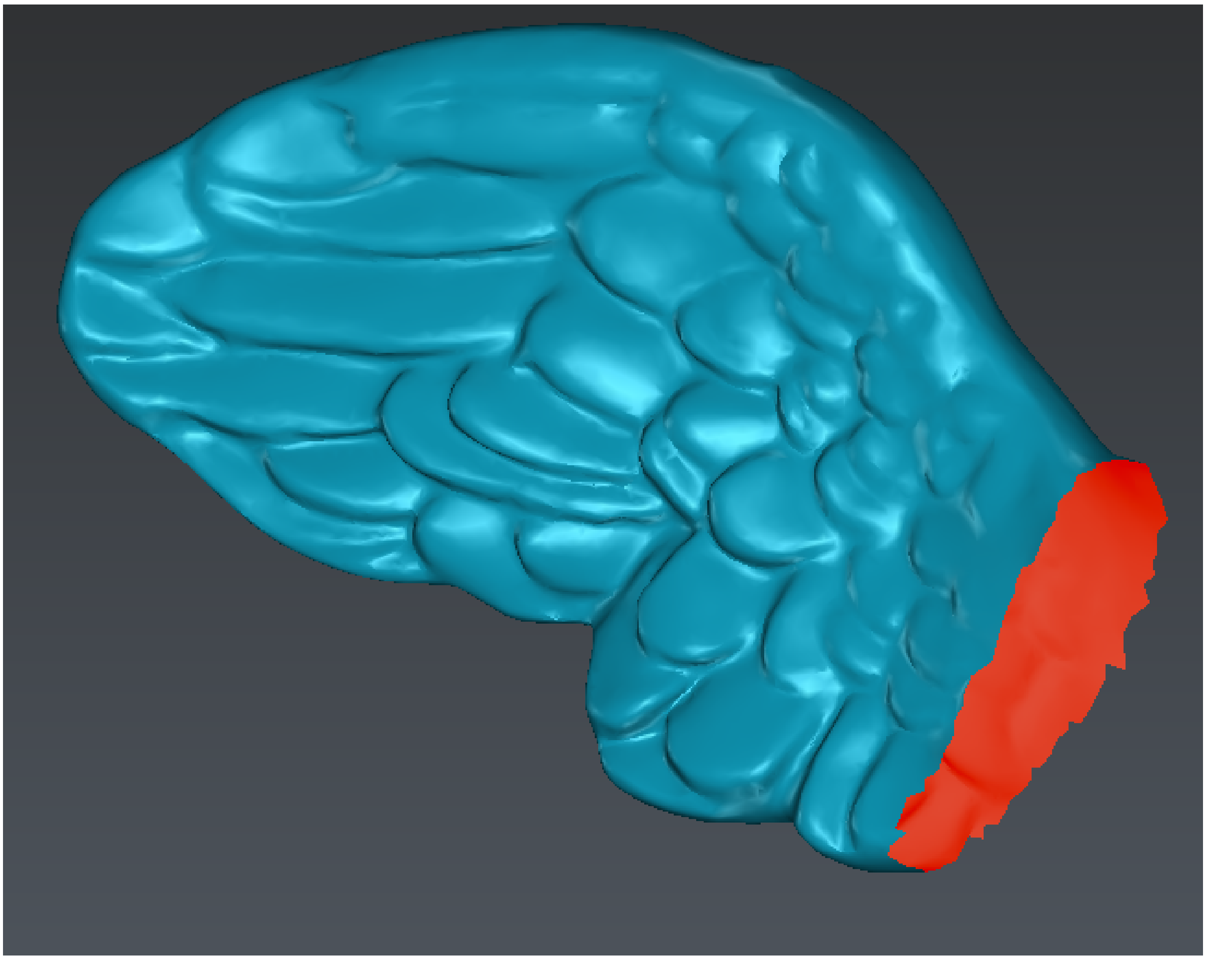
We saw in the 3D mesh creation chapter that we can do a mesh in two steps: create a rough mesh and then refine it in order to add details to the mesh. The command Refine Mesh from Cloud Interpolation corresponds to this second step.

Select the cloud **Victory** and the mesh **Wing 1 (to refine)**, and then go to Refine Mesh from Cloud Interpolation. 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 **Distance**
- **No free border modification**, so that we can very easily merge all the parts at the end



40 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 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 56.02%, it means than we divided the mesh size approximately by 2.

7.2.2 Exercise: Merge meshes with common borders

Open the file **Victory.3dr**.

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 Merge Common Borders, select the option **Do not modify borders** and click **OK**. You will have only one mesh without any holes.

41 The merged mesh

7.2.3 Exercise: Merge meshes with different borders

Open the file **MergeMeshes.3dr**.

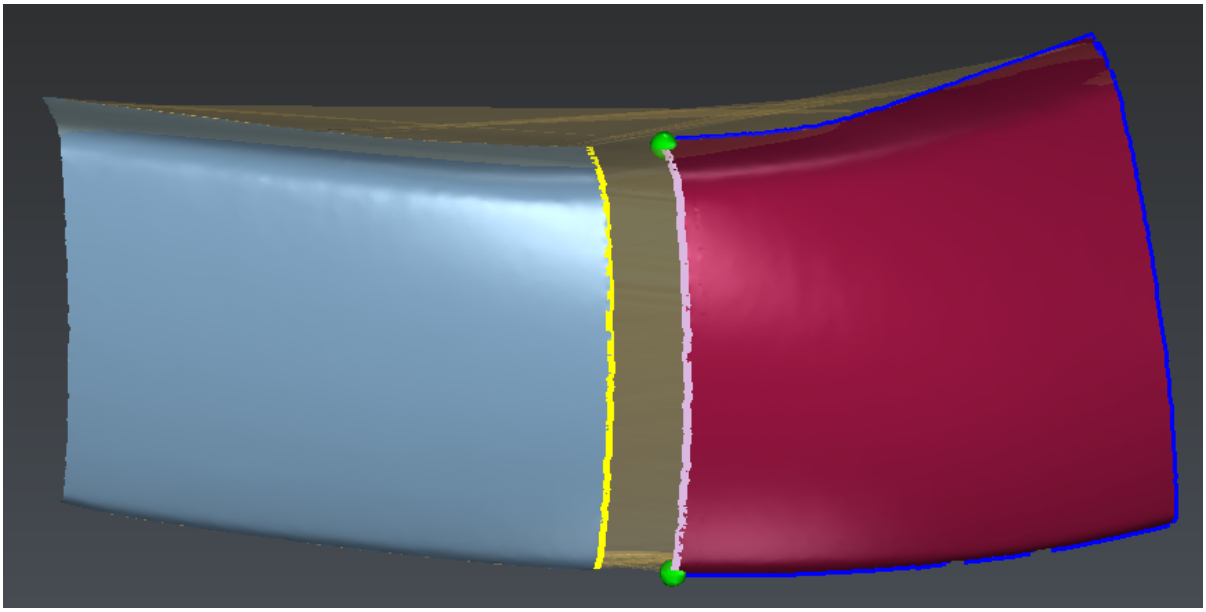
This file contains 3 meshes, and the goal of the exercise is to merge the meshes together.

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 Join 2 Contours.

Then click a point on each mesh border. Choose to continue with the entire contour. The result is not the expected one because the complete borders have been joined, but we want the junction to be only on the middle part.

Click the button **Restrict**, and then click on 2 points on the first contour to join 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.



42 Define the portion to join

Then you can change tangency criteria. The best choice in this case is tangent **to the surface**. Select the option **Triangle Reorganization** in order to improve the result. Do not forget to select the option **Sew** 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 Clean / Separate Mesh), 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 Constraint Mesh. Uncheck **2D computation** and check **Cut in parts.** 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 Merge Common Borders in order to create only one mesh.
- Select both meshes and go to 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 Group Mesh. The associated command Ungroup 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.

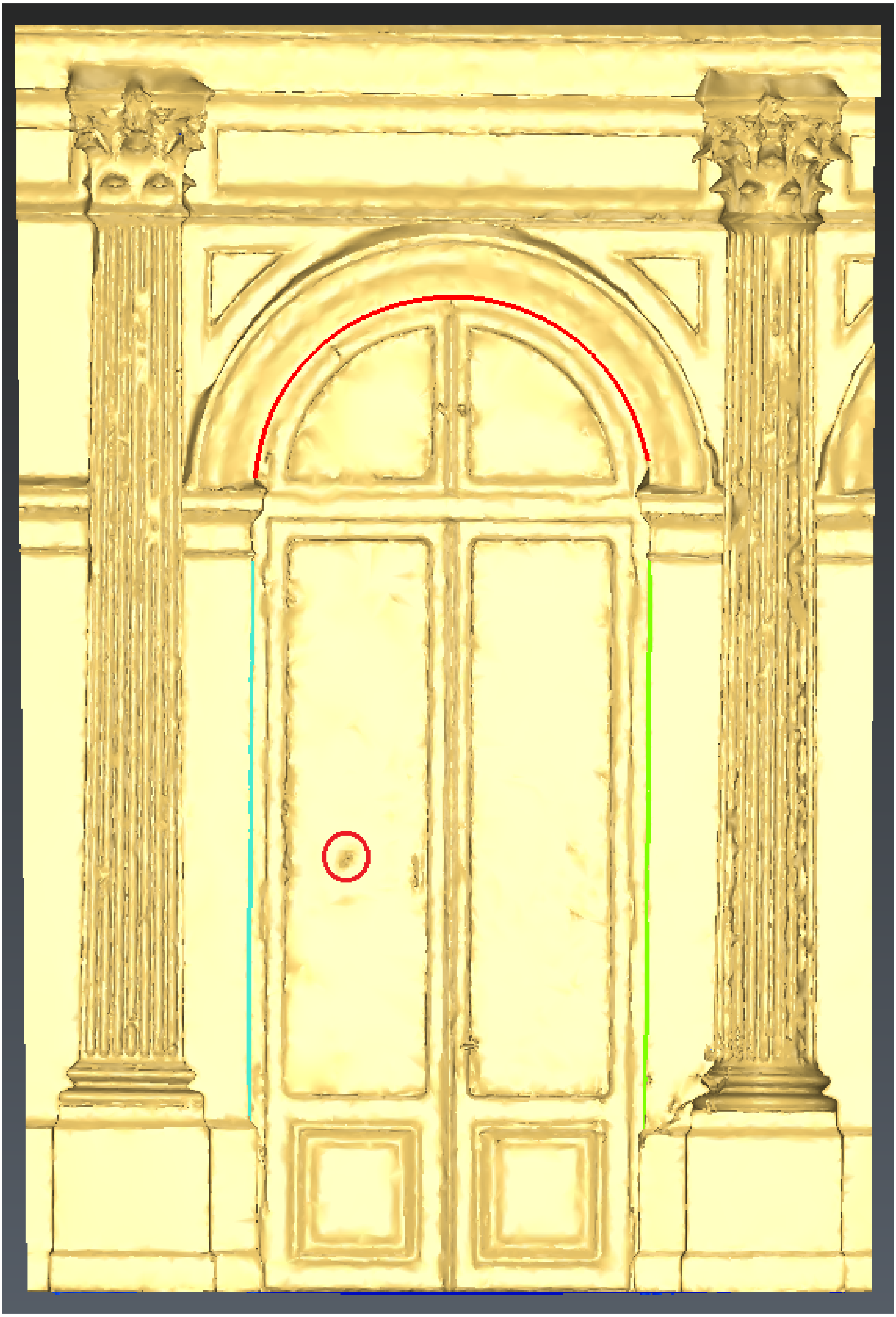
7.2.4 Exercise: Improve global aspect and edges

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.3dr**.

Local smoothing a smoothed fictive line

Select the mesh **Facade 1** and go to Replace a Part. 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.



43 Replace a part of a mesh
Click **OK** to validate the correction.

Global smoothing

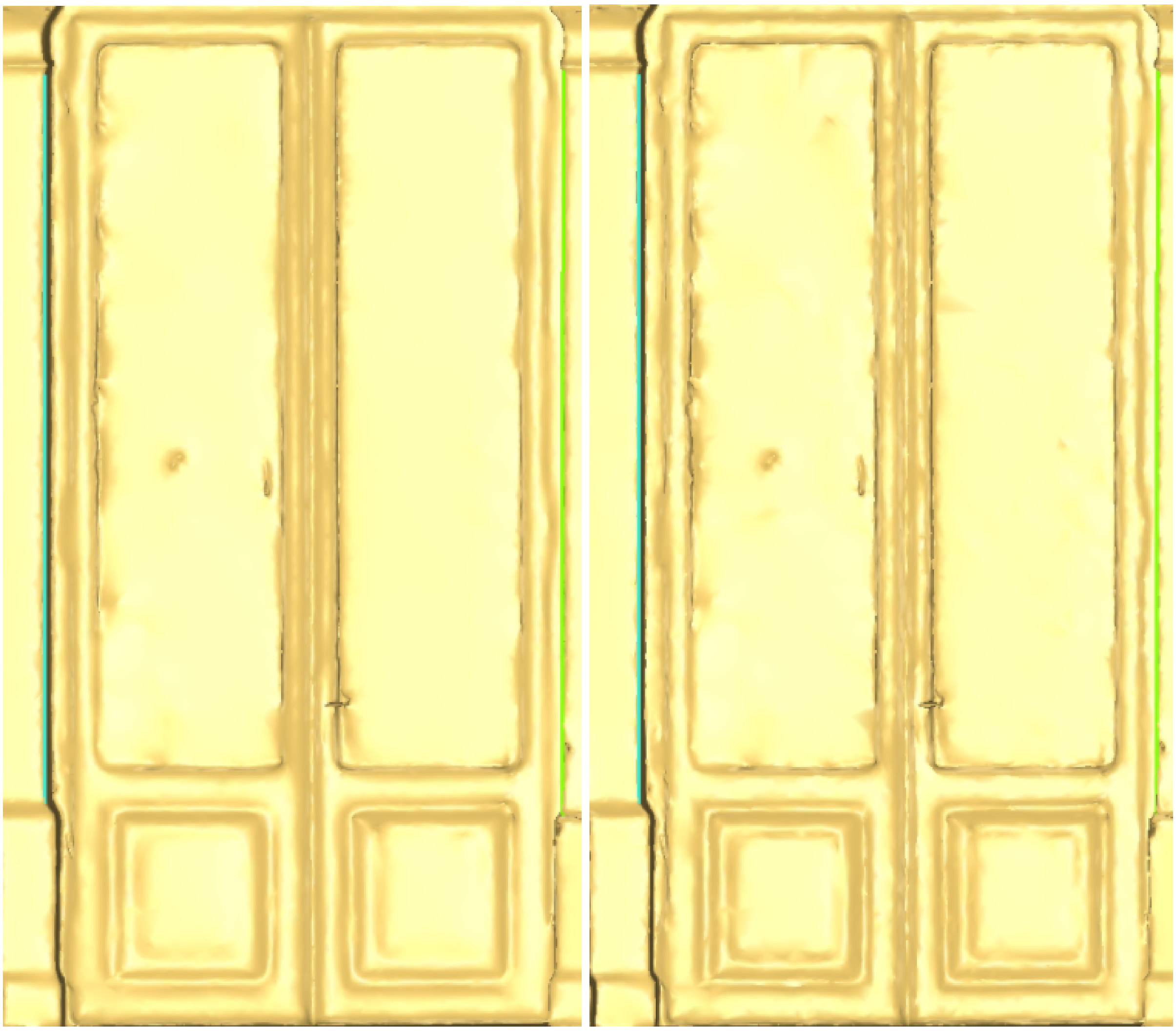
Once all aberrant zones are removed, we can apply a global smoothing in order to make the mesh less faceted, spiky and noisy. Select the mesh and go to Global Smoothing. There are two methods for smoothing:

- **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 **Intensity** to **Medium**, in the middle of the slider. Then click **Preview**. You can see that the global shape looks smoother but all sharp edges have been rounded.

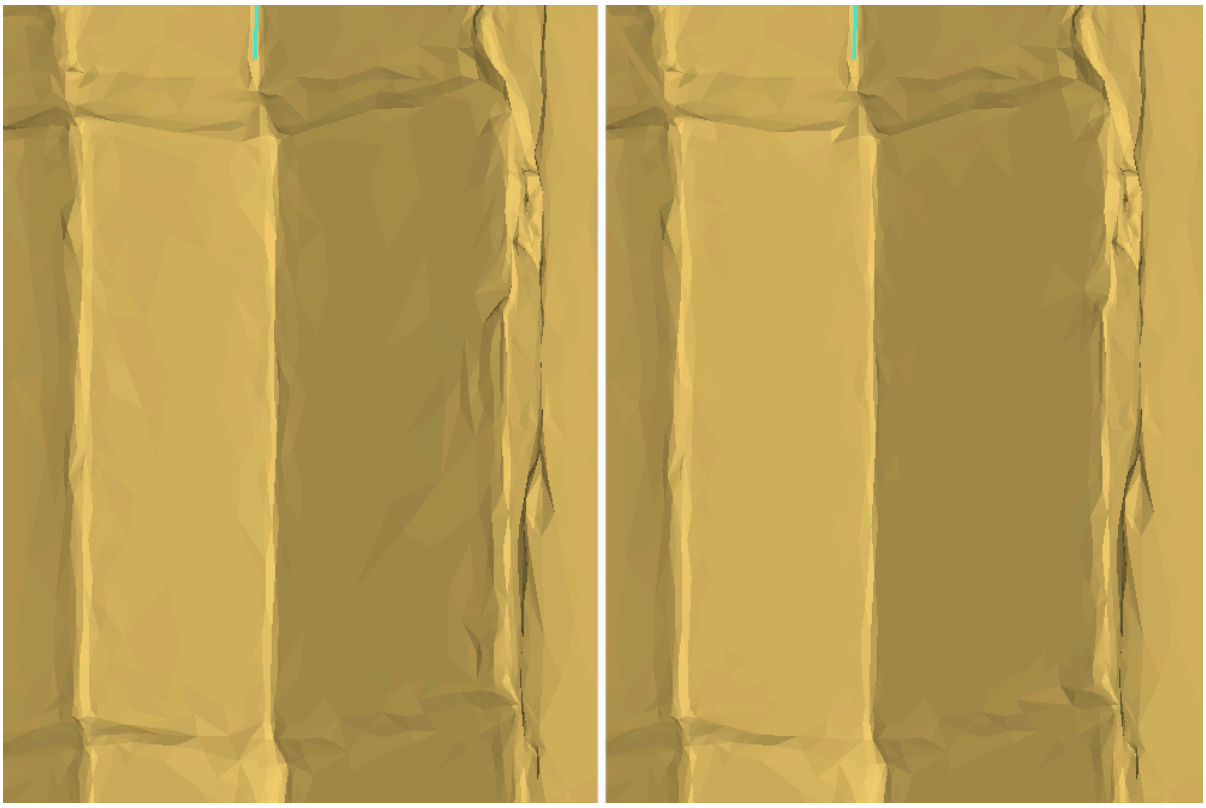
Now select the option **Control Deviation** and set the **Deviation** to 5 millimeters and the **Smoothing intensity** to **High**. Then click **Preview**. With this option, we control the deformation; it means that the maximum distance between the two meshes (before / after) will be 5 millimeters. The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



44 without deviation control (on the left); with deviation control (on the right)

Local Smoothing

You may need to smooth only specific parts of your model. For this, you can choose to work on small parts of the model. Select the mesh and go to Local Smoothing. In this command, the cursor works as a brush that smoothes what is under it. You can change the **Brush size** and **Smooth Intensity** according to your need.



45 Before Local Smoothing (on the left); After Local Smoothing (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, 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 Single Breaking Line,
- **Edge Corner 1** obtained with the command Region Grow Plane and Intersection,
- **Edge Corner 2** obtained with the command Region Grow Plane and Intersection.

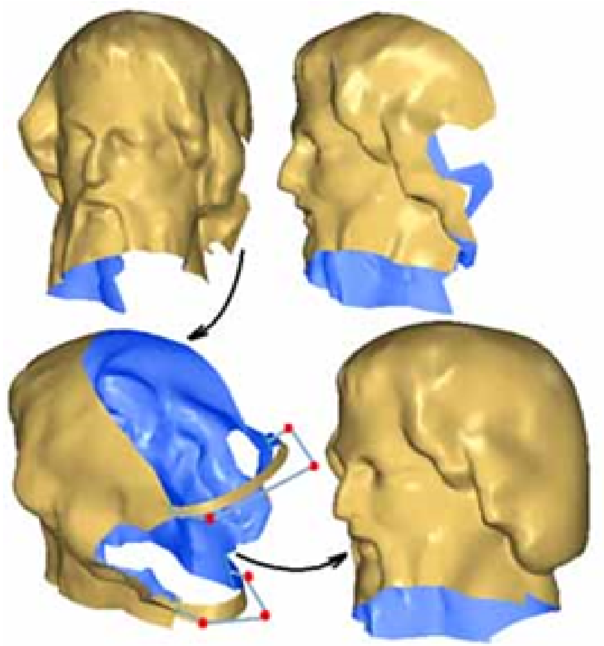
Select the mesh and the three polylines and go to Sharp Edges. The parameter **Cleaning Distance** will delimitate the cleaning area around the polylines, then all the triangles in this area will be changed. If the distance is too small or too big, the result will not be correct. Enter 0.13 m for **Cleaning Distance**.

Press the **Preview** button. The 3 sharp edges will appear on the mesh. Click **OK** to validate. Feel free to enter different values in order to see the difference.

7.2.5 Exercise: Fill holes with curvature filling

Open the file **FillHoles.3dr** and run Elevation View twice.

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 cannot directly use the command Fill Holes because the hole is too large and the neck part should not be filled at all.



46 In the case of the file FillHoles.3dr, 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

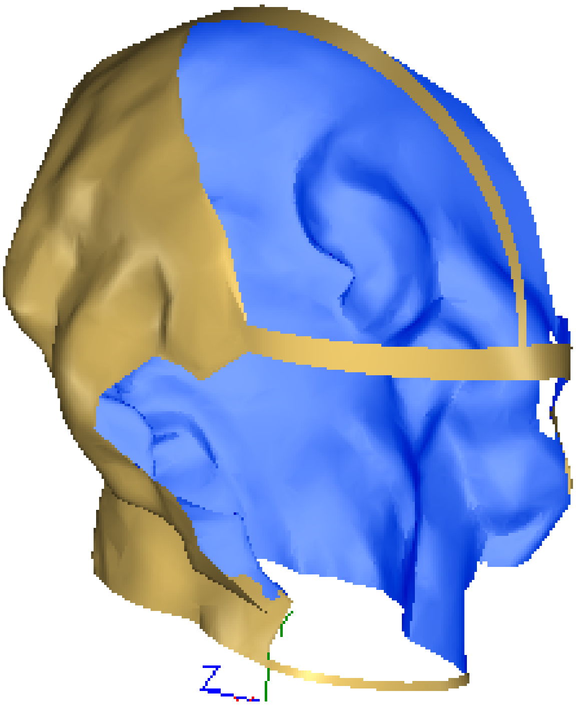
Launch the command Bridge. We will create two bridges: one that is used to delimit the back of the neck, and a second one that will be used to guide the subsequent hole filling process by its round smooth shape imitating the back of the head.

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,
- You can adjust the **Orientation,**
- When you are satisfied with this first bridge, select **Sew** 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.



47 Example of bridges

Fill Holes

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

- Click all the holes except the neck hole,
- Activate options **Add points inside 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,**
- Click the button **OK, Next,**
- Click the neck hole in order to select it,
- Unselect 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 Global Smoothing 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.3dr**.

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 Define Borders. There is only one parameter to set: the **Cleaning 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 in text field to click two points and then enter two points in order to define the distance

Enter, for example, a value greater than 10, like 11. Click on **Preview**. The result is bad because one hole has not been reconstructed due to a too high value, as we can see in the results.

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.

48 Borders have been improved in only one click

8 Sections and Polylines

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

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

- Join 2 Polylines: to compute a curve to link the extremities of two polylines
- Extract All Holes and Borders: to extract the lines of the free contours and the holes in a mesh
- 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 Extract:

- Intersection: to compute various intersections, between polylines, meshes and geometric shapes
- 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 kinds 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.

8.1.1 Exercise: Create a planar section on a point cloud

Open the file **SectionsBuildingPlan.3dr**

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 Planar sections.

- Choose Z for the plane direction.
- Choose **List of values**.

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 (you can use Nearest 3D Projection in the point selection ribbon).

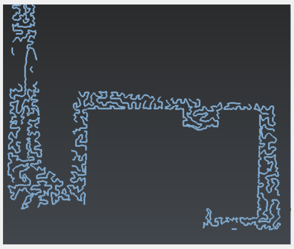
To create sections on a point cloud, additional parameters have to be entered:

- The slice 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 **Noise reduction** in order to have smoother polylines.

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

From the top view (press Z).

Case 1:	Case 2:	Case 3:	Case 4:
▪ **Slice thickness: 0.1** ▪ **Chaining distance: 0.5** ▪ **Check Noise reduction**	▪ **Slice thickness: 0.7** ▪ **Chaining distance: 0.5** ▪ **Check Noise reduction**	▪ **Slice thickness: 0.35** ▪ **Chaining distance: 0.1** ▪ **Check Noise reduction**	▪ **Slice thickness: 0.35** ▪ **Chaining distance: 2** ▪ **Check Noise reduction**
			

1 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 (**case 1**) 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 in **case 2**.

If you enter a chaining distance too low, you will have many short independent polylines as in **case 3**. If you enter a chaining distance too high, all the points will be chained together as in **case 4**. You will obtain long polylines and you may need to cut them for further processing.

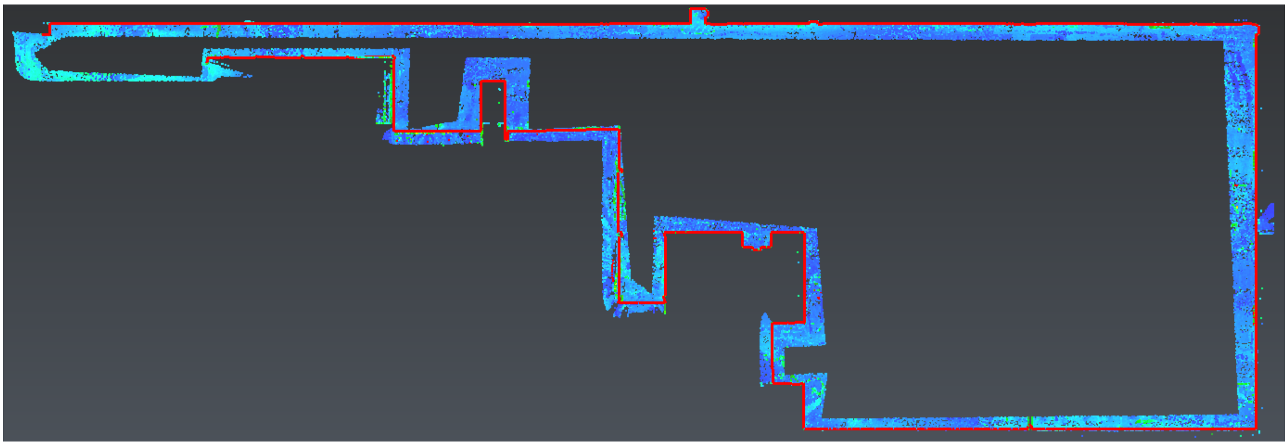
For this exercise:

- Set **Slice thickness** to 0.35

- Set **Chaining distance** to 0.5
- Check **Noise reduction.**

Click **Preview** and **OK** to validate the results. A new folder containing all the polylines is created.

Please see the Manage Polyline and Improve Polyline sections 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.



49 2D plan of a building

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

Open the file **SectionsContourLines.3dr**.

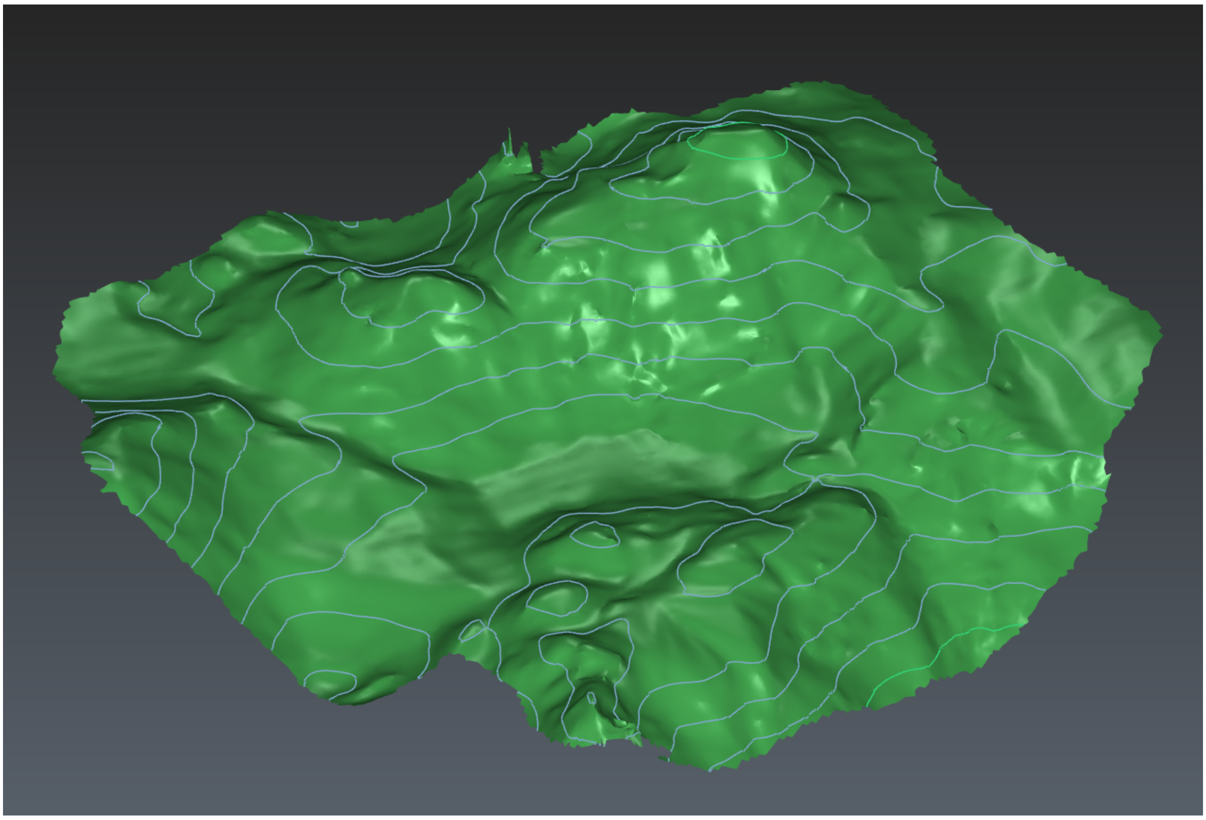
It contains the mesh of a mountain, and we want to create contour lines on it. Select the mesh and go to Planar sections. This command allows to compute one or more sections, all in parallel planes, defined by a given direction.

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 axis direction.
- Choose the option Regular.
- Set the Step to 50 m in order to have a section each 50 meters on the whole mesh.
- Set the Range to All over.

When you press **Preview** a result window appears to inform you that 38 polylines have been extracted. You can hide the display of the planes.

Click **OK**. All the polylines are inserted in a new folder called **Planar sections**. Each polyline is named with its Z value.



50 Create contour lines

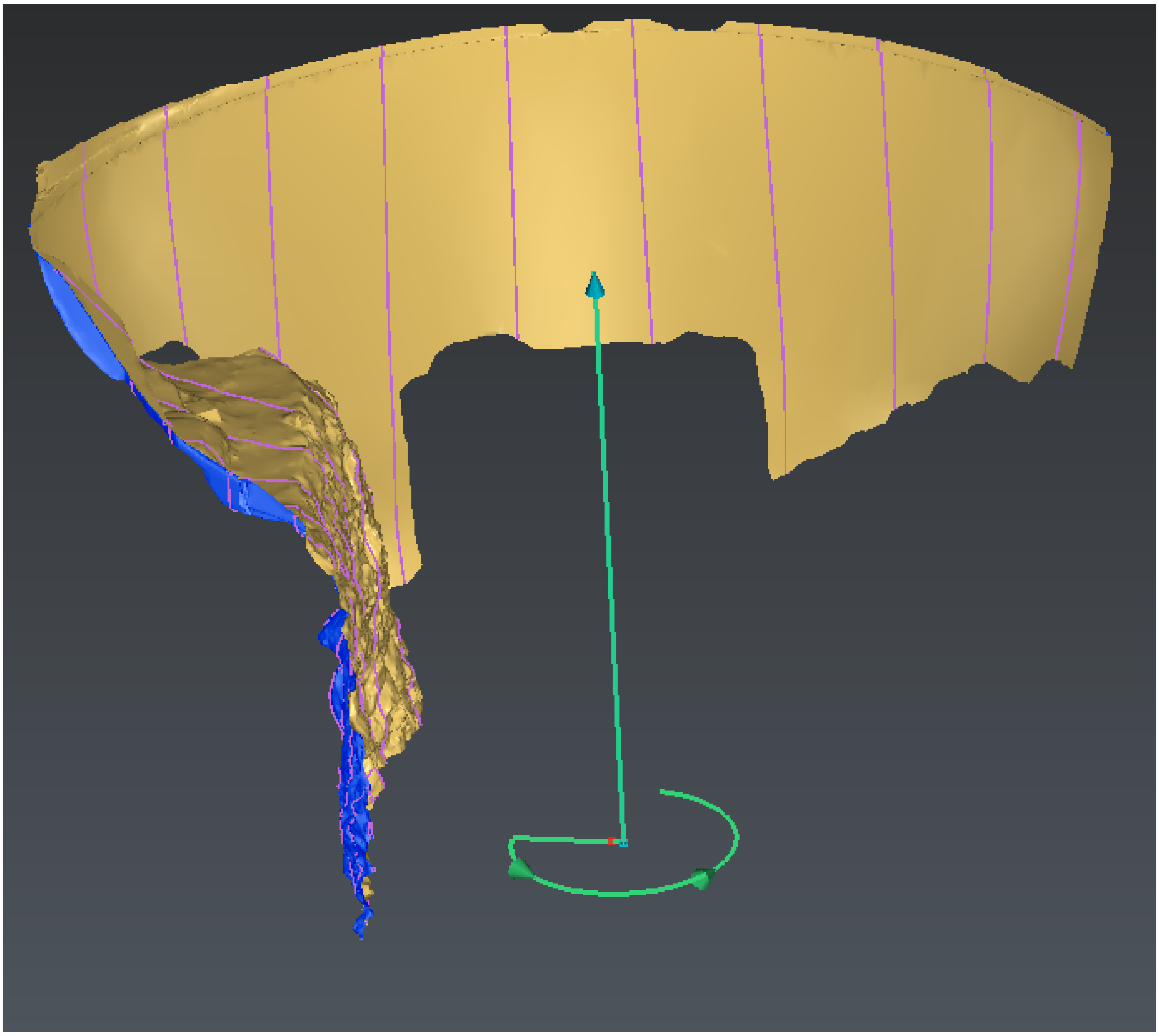
8.1.3 Exercise: Radial sections on a mesh

Open the file **SectionsDynamic.3dr**.

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 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, 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.

Choose **Regular** option. 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).



51 Create radial sections on a mesh

8.1.4 Exercise: Guided sections on a mesh

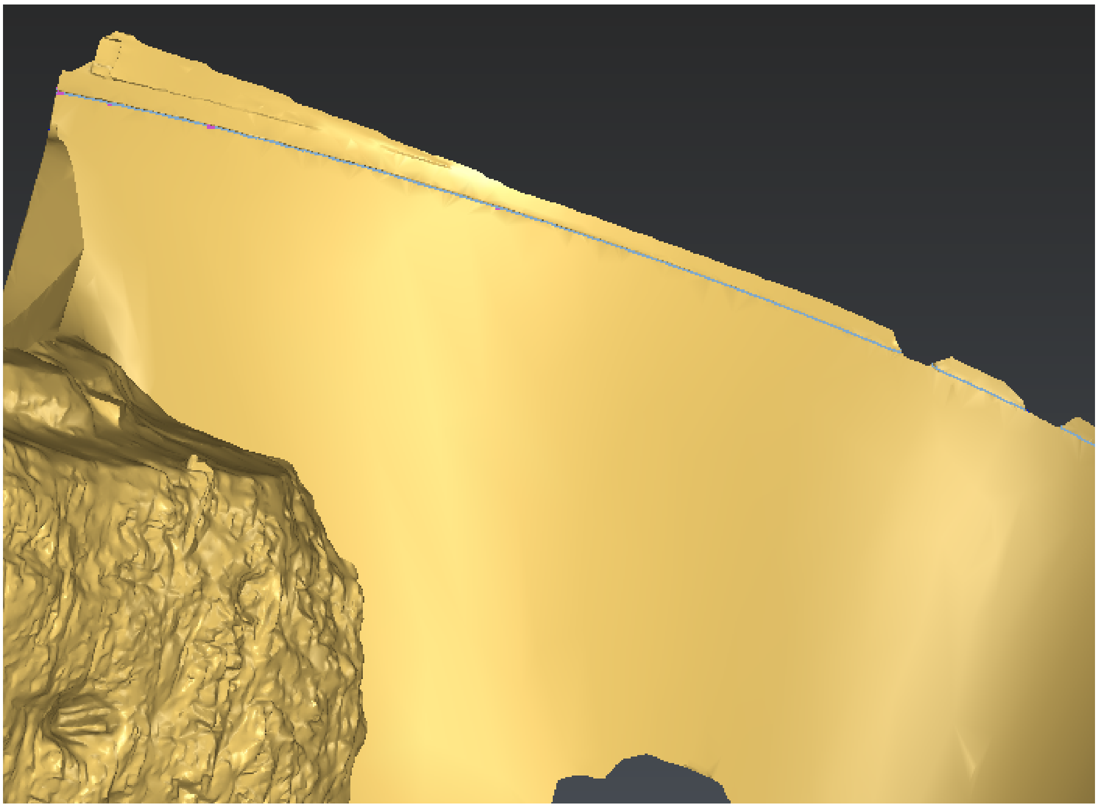
Open the file **SectionsDynamic.3dr**.

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

Freehand sections

Select the mesh and go to Freehand sections. Choose the option **Best plane** 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.

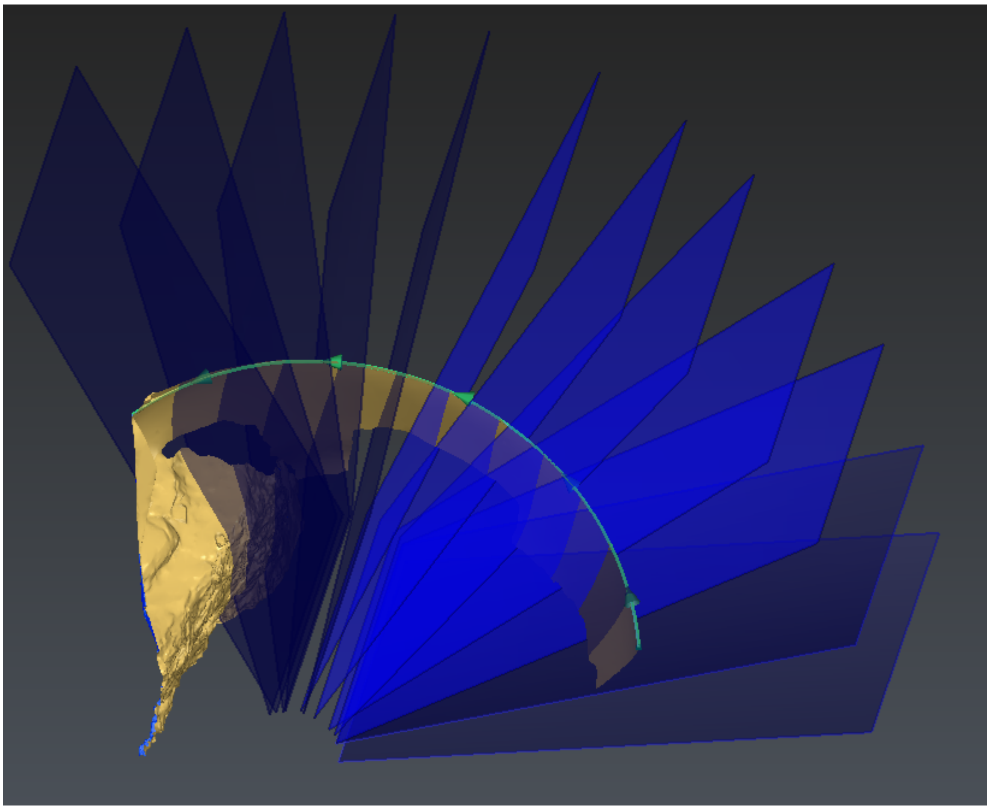
With the option **Projected segments**, each time you click a point on the surface, the segment between this point and the previous one is projected on the mesh according to the view direction. In this way you can force the section to go through specific points that are not all necessarily on the same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



52 Create a planar freehand section on a mesh
Now validate the result. A new polyline is created (or a set of polyline, if so ungroup it and continue the exercise with the longest polyline).

Sections along a curve

Select the mesh and the polyline, created just before, and go to Sections along Curve. You can see an arrow on the polyline indicating its direction.

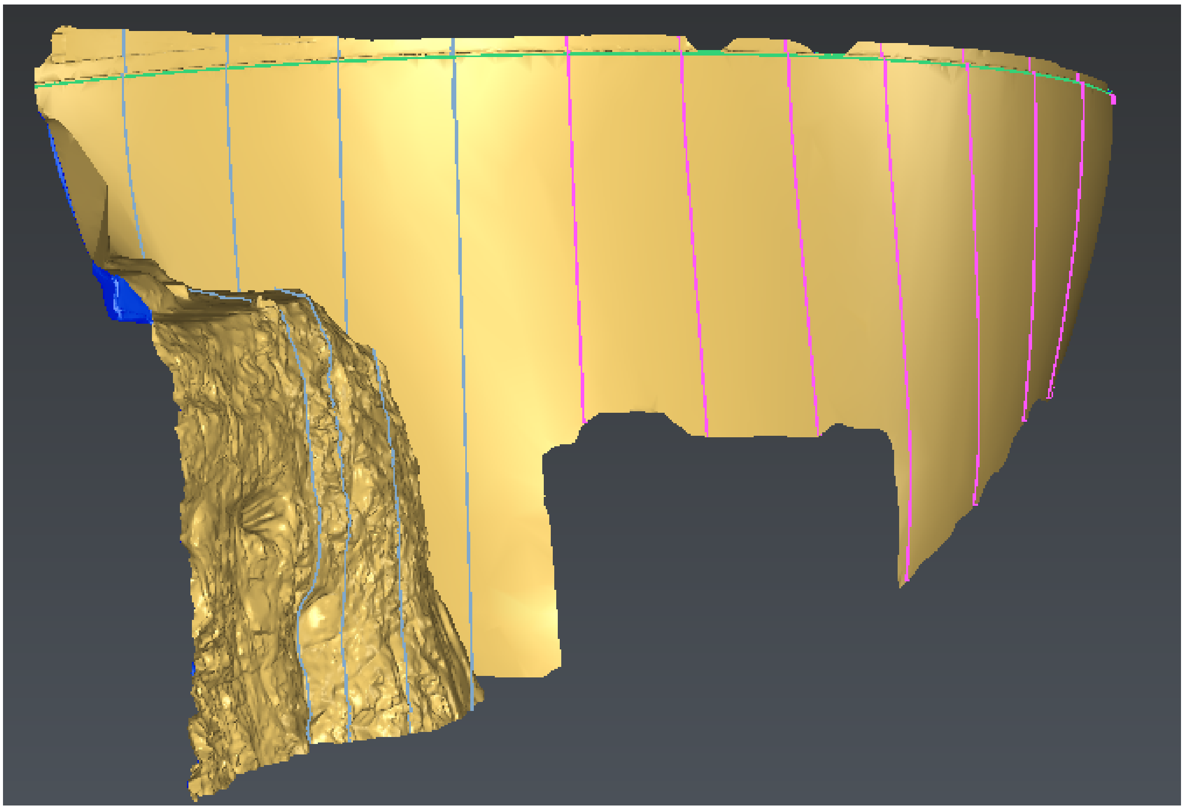


53 Create a planar freehand section on a mesh
You can draw sections with a regular step or by giving a list of distances from the first point of the polyline. All the distances to enter are curvilinear distances along the polyline.

- Choose Regular option.
- Set the step to 10 m.

You can display temporarily the planes where sections will be created in the Outputs section. The planes are locally perpendicular to the polyline.

Click **Preview** then **OK** to validate the results. The sections created are named according to their distance to the first point of the polyline. One section can be either a polyline or a set of polylines depending on the holes in the mesh. In the picture below grey sections are sets of polylines and pink sections are polylines.



54 Polylines and sets of polylines

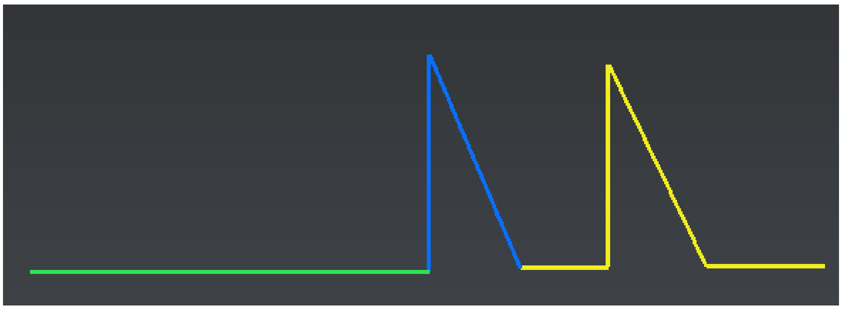
> **Note**
>
> You can explode the sets of polylines by selecting them and going to Ungroup Polyline or by selecting them and right clicking on Ungroup.

8.2 Manage polylines

- Exercise: Cut polylines
- Exercise: Chain polylines

8.2.1 Exercise: Cut polylines

Any polyline can be split at any point. The polylines do not have to be selected before opening: Cut Polyline. Select the appropriate option in the toolbar to cut the polylines at specific points. This can be a way to manually clean a polyline.



55 Polyline cut in three parts

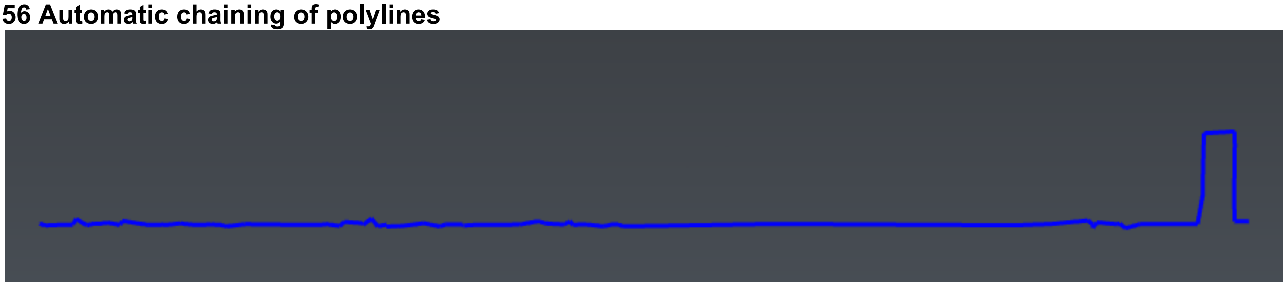
8.2.2 Exercise: Chain polylines

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

Open the file **SectionsBuildingPlan.3dr**.

You can chain the polylines of the group **Lines to chain**. Select the polylines and go to Chain polylines. Uncheck all the options.

56 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 Ungroup Polyline.

8.3 Improve polylines

- Exercise: Improve polylines

8.3.1 Exercise: Improve polylines

Open the file **SectionsBuildingPlan.3dr**.

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 achieve.

Simplify / Resample a polyline

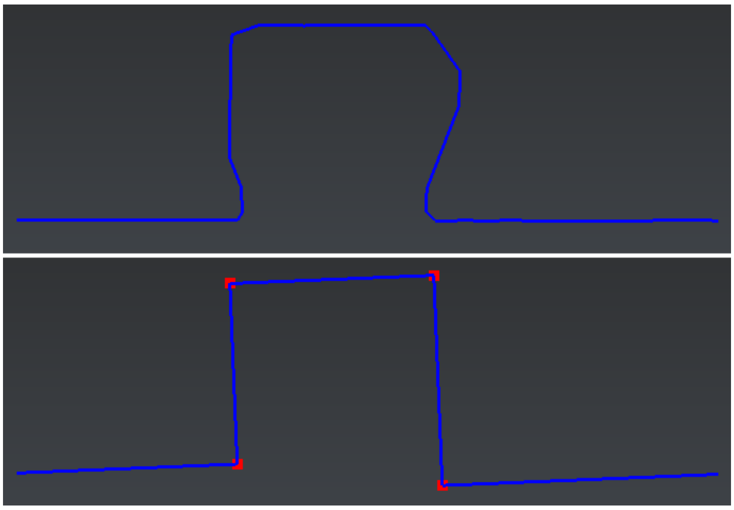
A polyline can be simplified in order to reduce its noise or to recreate right angles.

Select a small noisy polyline (for example RightAngles) and go to Resample Polyline. Set the number of points of the resampled polyline. If you want a straight line, enter **2**. The option **Optimize vertices position** will compute a polyline going through the noise in order to reduce the standard deviation error. This tool can also be used to recreate the right angles on a polyline.

You can try it on the polyline called **RightAngles:**

- Define number of vertices to 6
- Check Optimize vertices position
- Check Make right angles and set the value to 10°

See the result in the picture below.



57 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 option **Optimize number of vertices** 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 **Compute value**.

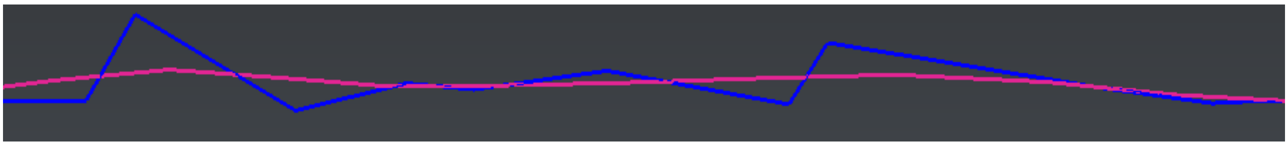
Smoothing

To reduce the noise in a polyline, you can also select the polyline **LineToChain6** and go to Smooth Polyline. Three types of smoothing are available. The smoothing intensity represents the number of iterations of the process.

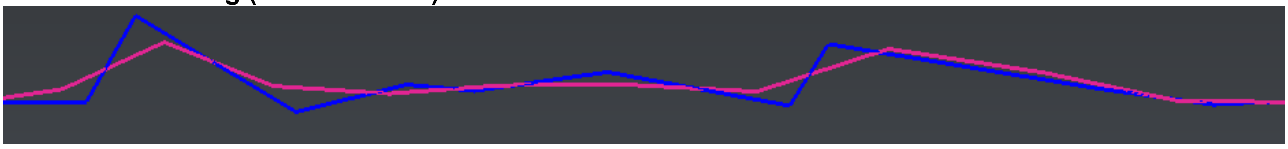
See the pictures below to compare the hard and soft smooth used on the same polyline with the same intensity. The original polyline is in blue and the smoothed polyline in pink. Hard smooth creates a smoother polyline whereas soft smooth tries to keep the general shape of the polyline. With type Bspline, 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 hard smooth; it will tend to a smaller circle. With soft smooth, the radius of the circle will be approximately preserved.

With hard and soft smooth, it is also possible to control the deviation error by entering the maximum deviation authorized between the smoothed line and the original.



58 Hard smoothing (LineToChain6)



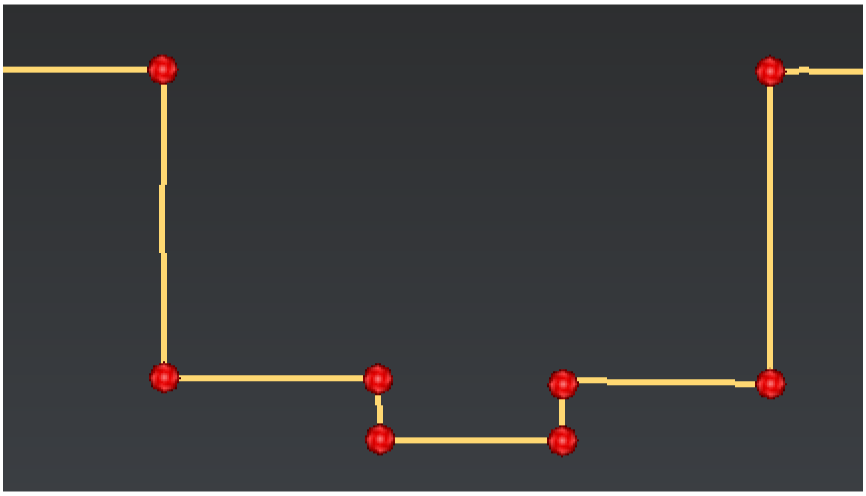
59 Soft smoothing (LineToChain6)

Stretching

Select a polyline and go to Stretch polyline.

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

You can add intermediate control points by clicking specific points on the polyline in the 3D scene (on the right angles, for example).



60 Stretching

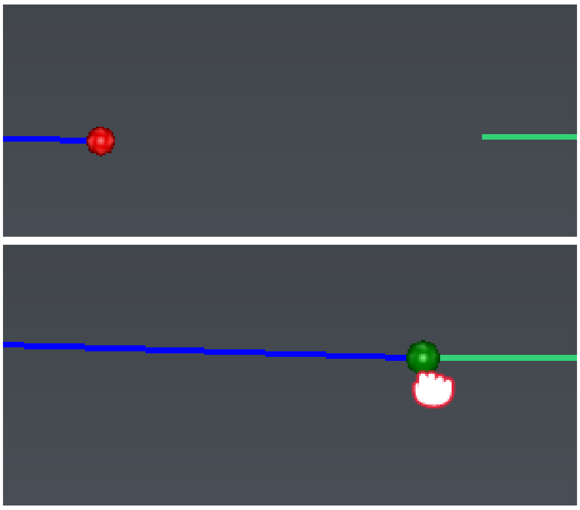
Stretch a polyline with several control points

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

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.

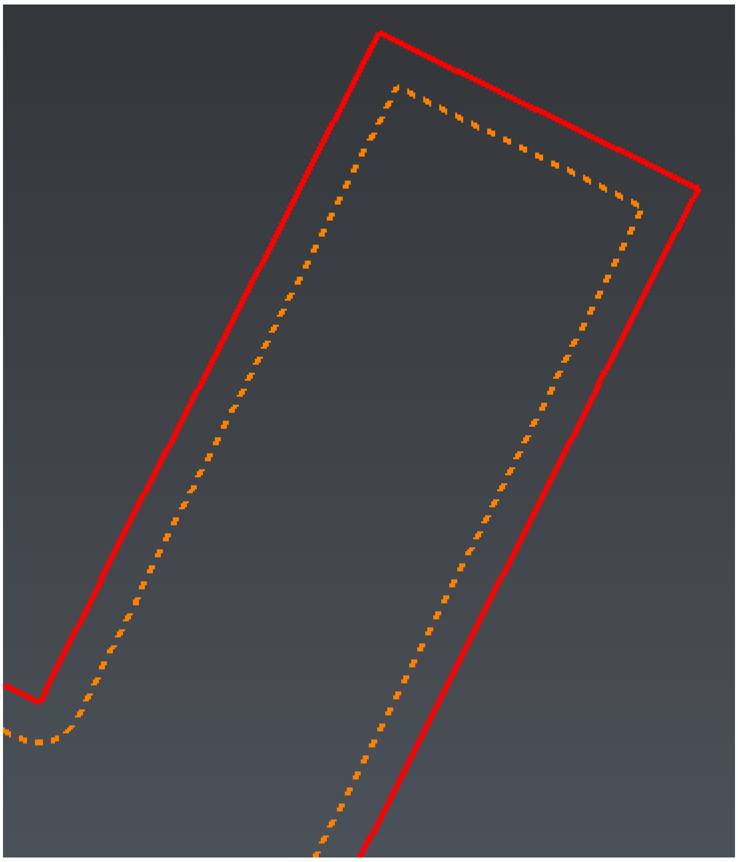
Activate the option **Snap points to other lines** to be able to link two polylines by moving a control point from one polyline on the other. When the control point is green, it means that the polylines are connected.



61 Snap points while stretching

Offset

You can select your final contour or show the object **Final Building Contour** and go to 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 **0.3m** 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.

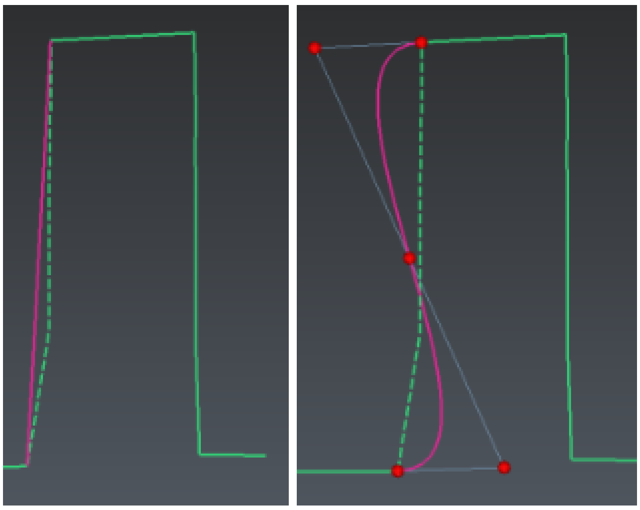


62 Offset a polyline

Replace a portion

You can replace a portion of a polyline with a segment or a 3D editable curve.

Select the polyline **LineToChain1**. Launch the command Replace a Portion. Click two points to define the portion that you want to replace by a straight line or a curve (press Enter to validate)

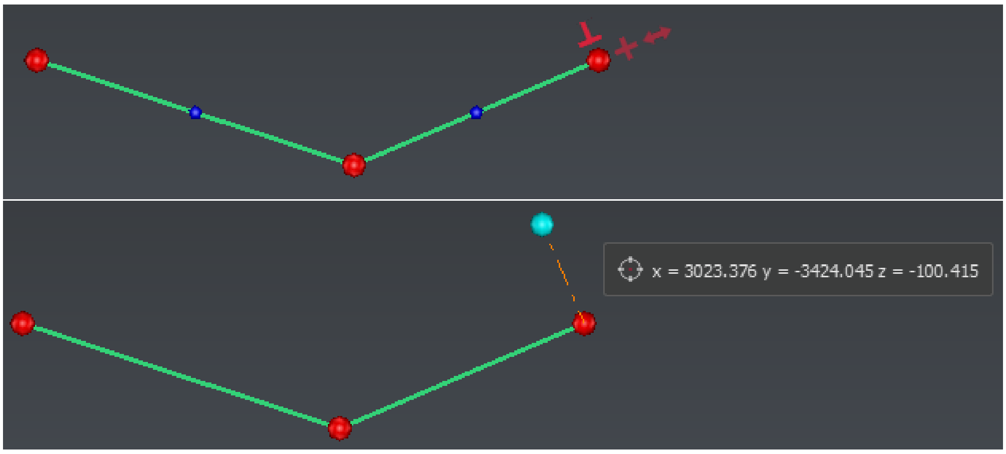


63 Replace a Portion

Manual Edition

You can edit a polyline manually using Edit Polyline. For instance, you can:

- remove one or move vertices,
- insert a vertex,
- extend or shorten end segments.



64 Edit Polyline

Learn how to create 2d building plans: continue with the practical exercise Create building plans.

8.4 Break line (Feature line) and sharp edge reconstruction

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.

Two types of lines can be extracted from a mesh:

- The single **Break line** is the line on the sharp edge.
- The **fictive line** is the line which can be used to recreate the sharp edge (i.e. planar intersection line).

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

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

8.4.1 Exercise: Extract a feature line or a break line from a mesh

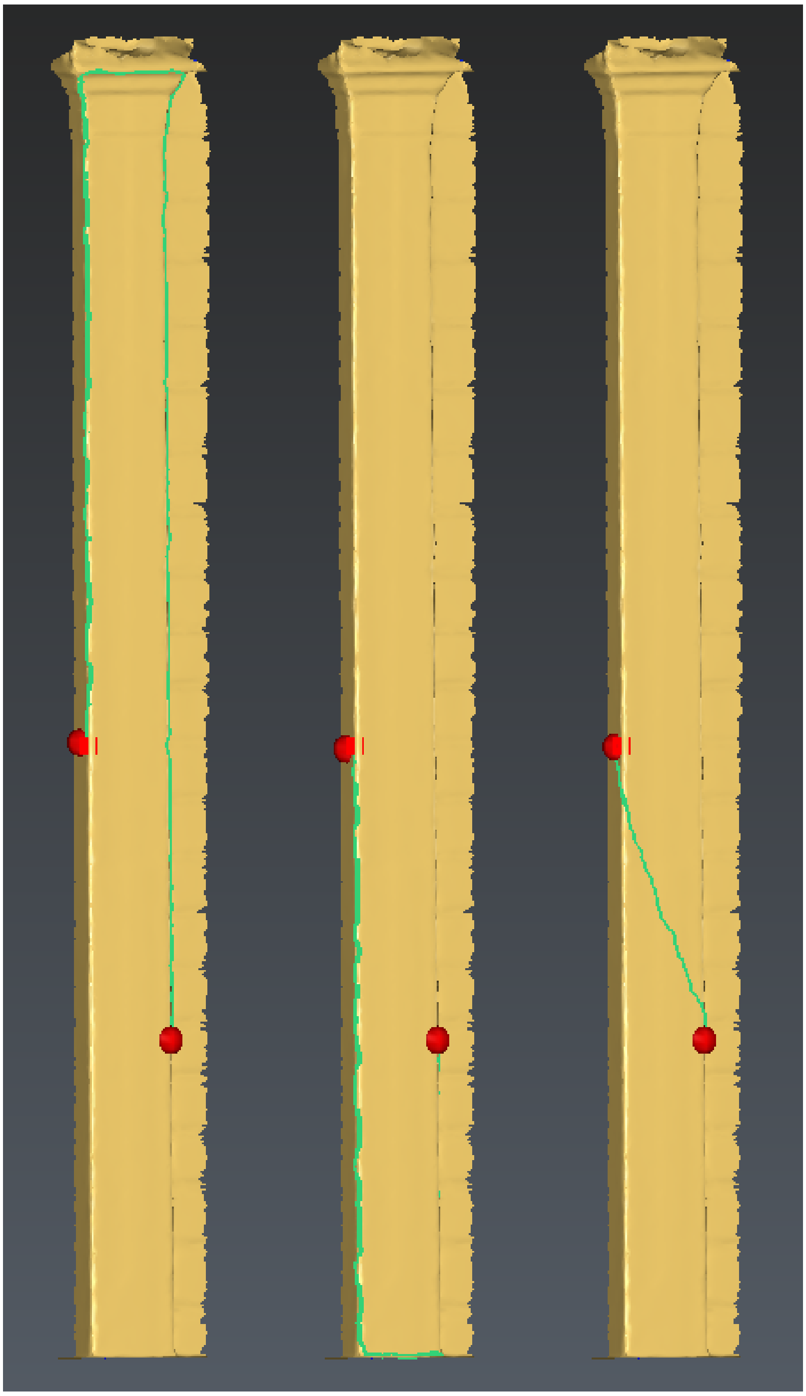
Open the file **FeatureLines.3dr**. It contains the mesh of a pillar and four points which will help you for the first step.

This guide will introduce you to Single Breaking Line extraction command used for the recreation of sharp edges.

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

- **Regardless the convexity**: With this method the path is authorized to be partially concave and partially convex.
- **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.
- **Shortest**: The line will link the two points in the shortest way possible by following the vertices of the mesh.

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



65 Three methods to extract a feature line

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. 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 you 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.

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

Clicked points appear as red balls. You can at each moment move these points by clicking them and keeping the button pressed.

Note for the 3 methods

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

Once the feature line is extracted you can extract another feature line with **OK, Next**.

8.4.2 Exercise: Rebuild a sharp edge using a feature line

Open the file **FeatureLines.3dr**.

Show the points **Point 1** and **Point 2**. Select the mesh and go to Single Breaking 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 **OK, Exit**.

Smooth lines

The idea is to straighten the line.

You can select the created line and launch the command Smooth Polyline.

If you wish, you can modify the smoothing parameters by changing the type of smoothing and the intensity. Choose **Hard smooth** and an intensity of **10.**

Note

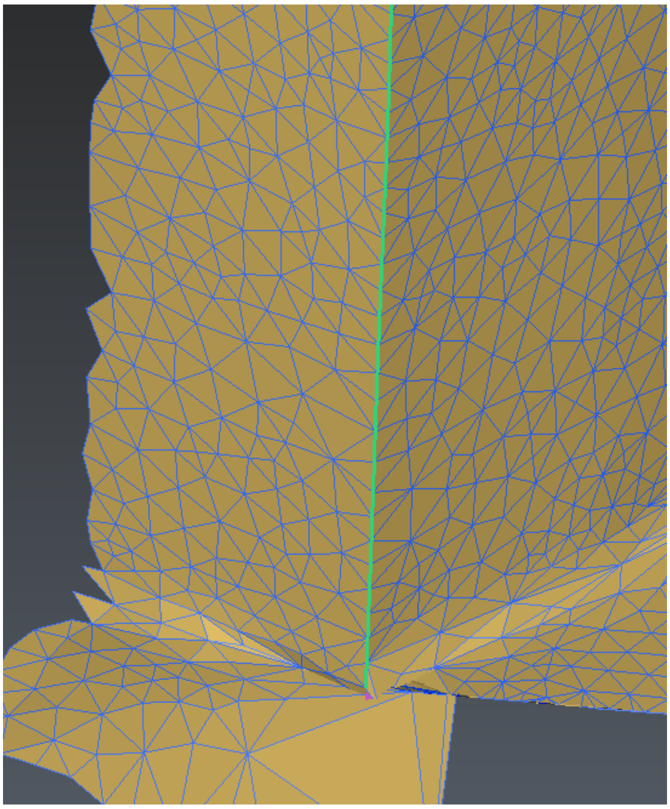
You can also use the other commands to make the line more straight: Replace a Portion, Stretch Polyline...

Sharp edge reconstruction

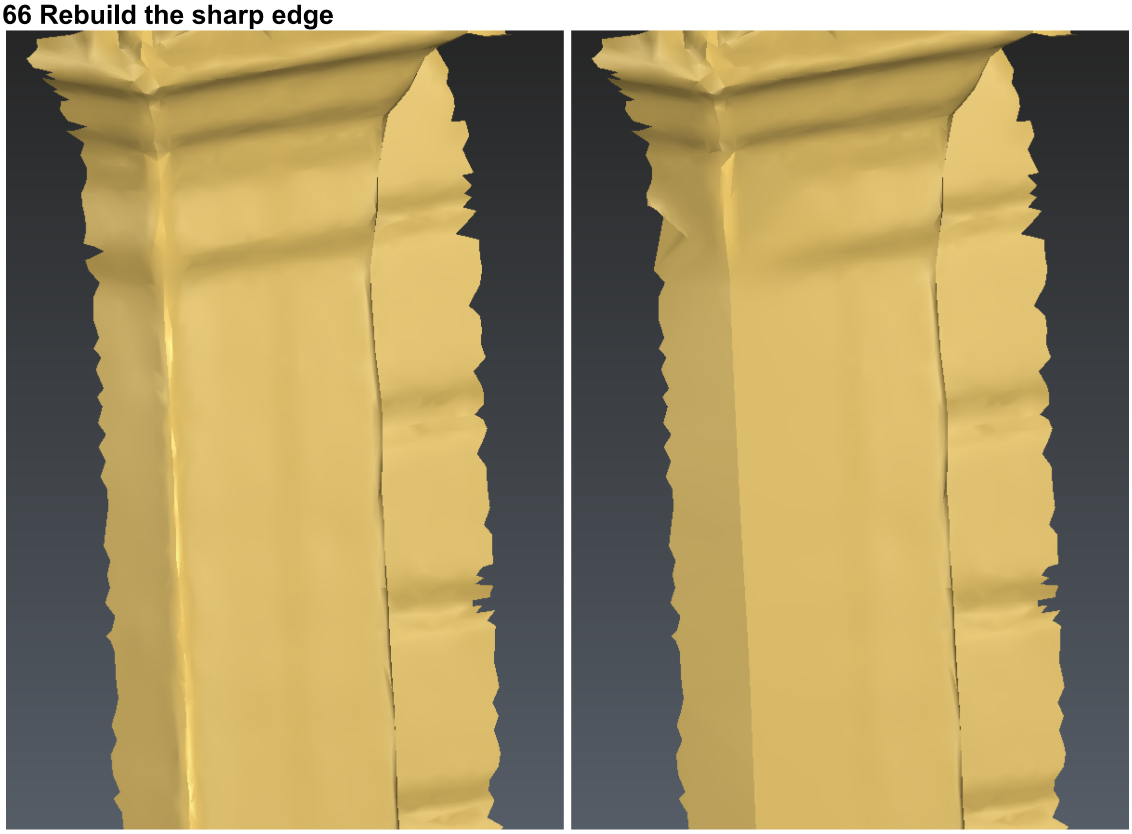
Select the mesh and the line and launch the command Sharp Edges. Set the **Cleaning Distance** to 0.3 m (approximate depth of the pillar).

Click on **Preview**: The mesh is modified, from 0.3 m from both side of the line, in order to respect the given feature line.

The edge of the pillar is now perfectly straight.



66 Rebuild the sharp edge



67 Compare the original edge with the reconstructed edge

You can try the Meshing a facade point cloud (downloadable online), to see the reconstruction of a vault with the same command Single Breaking 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 Intersection, and the edge is rebuilt with Sharp Edges.

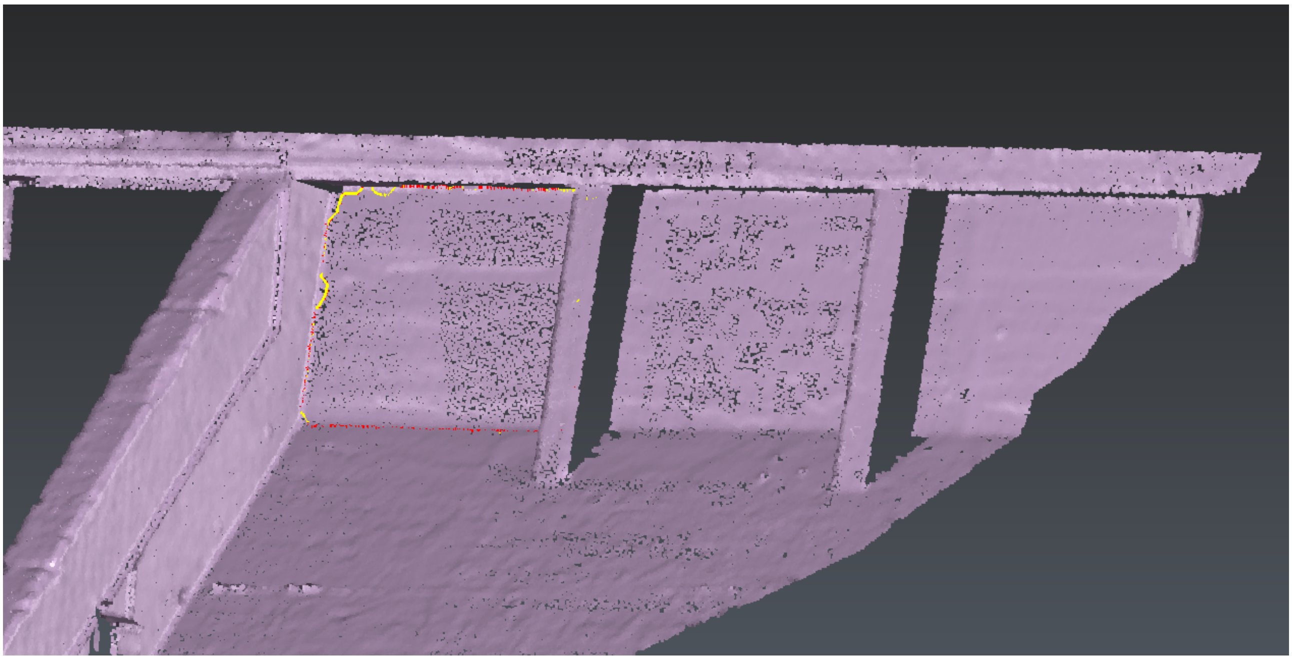
8.5 Polyline extraction

8.5.1 Exercise: Extract planar contours from a point cloud

Open the file **BestFitOnRef.3dr**.

Show only the cloud **Aligned Dam**. Launch the command Planar contour.

Click on one point on the cloud inside the face to extract (see image below). The software will automatically try to find the plane around the clicked point. Click **Preview**: the software identifies its contours: a yellow polyline which is planar. Optionally, you can create the corresponding 3d contour in red (this polyline goes through the real points of the cloud).



68 Planar contours extracted from a point cloud

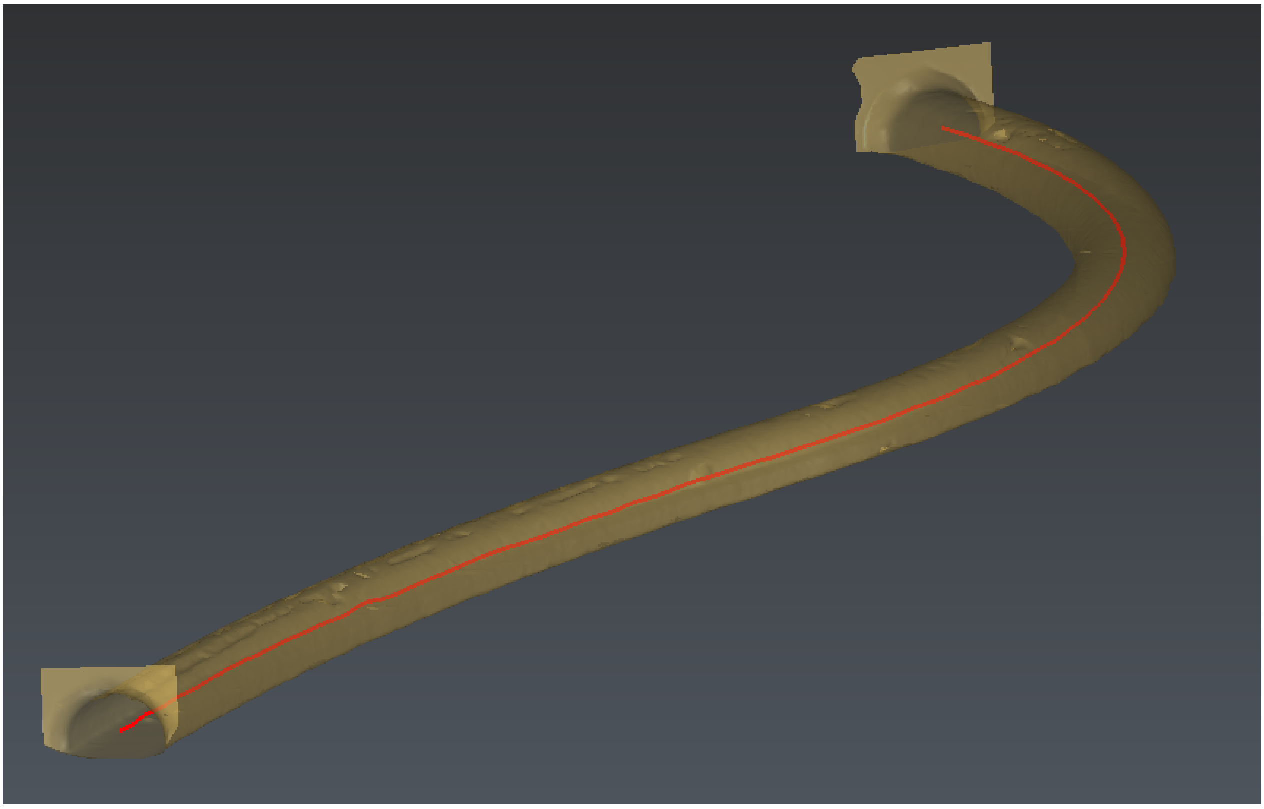
8.5.2 Exercise: Extract the neutral axis from a tunnel

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.3dr**. It contains a measured mesh of a tunnel named **Measured tunnel**. Select it and show it only.

Select it again and go to Neutral Axis.

Optionally, you can run this command with a “Help Line”. For instance, you can draw a polyline following the general direction of the tunnel. Then, select it with the mesh before launching the command.



69 Neutral axis of a tunnel without "Help Line"
Don't change any input and click on **Preview** to see the final axis computed. You can choose to smooth it or not. Click **OK** to validate the result.

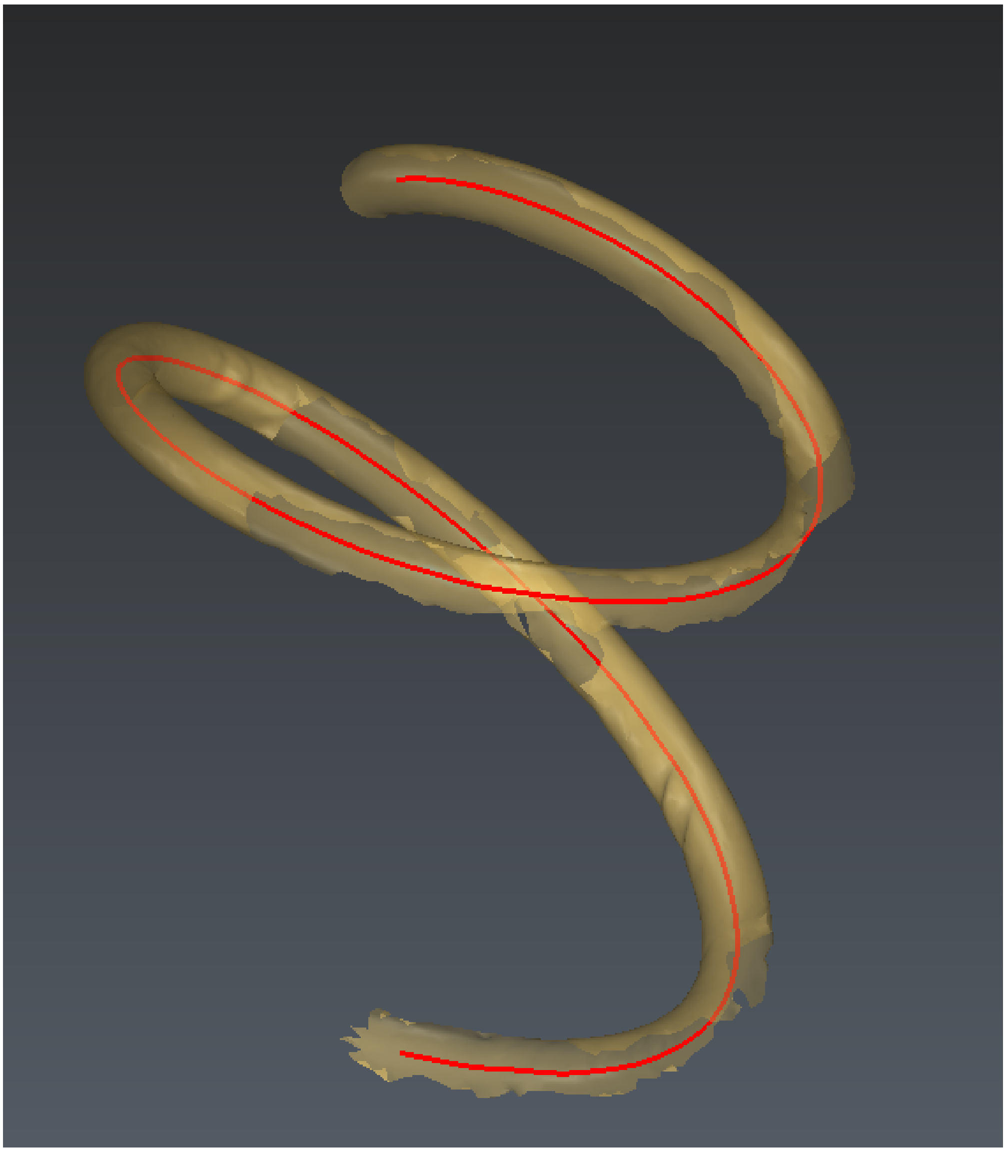
8.5.3 Exercise: reconstruction from neutral axis

Open the file **NeutralAxis&Extrusion.3dr**. It contains the point cloud and a partial mesh of a spring.

First, try the computation of the neutral axis on the **Spring Mesh**. Show only the mesh, select it and go to Neutral Axis.

Optionally, you can run this command with a “Help Line”. For instance, you can draw a polyline following the general direction of the spring. Then, select it with the mesh before launching the command.

Here, launch directly the command without any helpline.

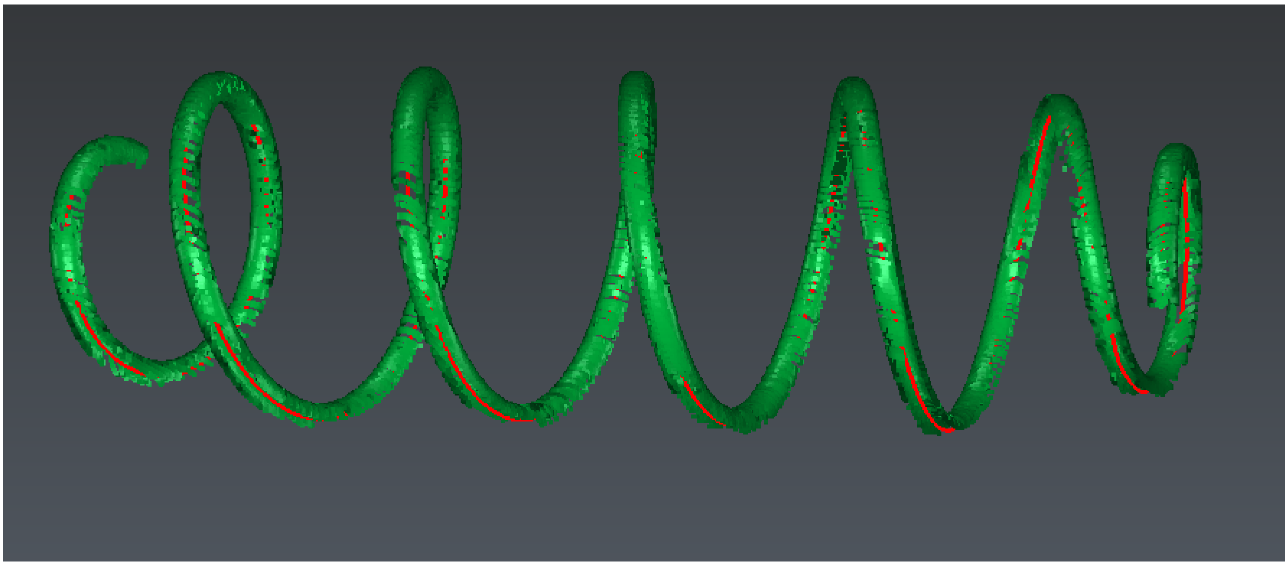


70 Computation of the neutral axis without "help line"
Click on **Preview** to see the final axis computed. You can choose to smooth it or not. Click **OK** to validate the result.

Now, try the computation on the **Spring Cloud**. Show only the point cloud, select it and go to Neutral Axis. To find the neutral axis inside the wire of spring, you have to enter the diameter of the fiber.

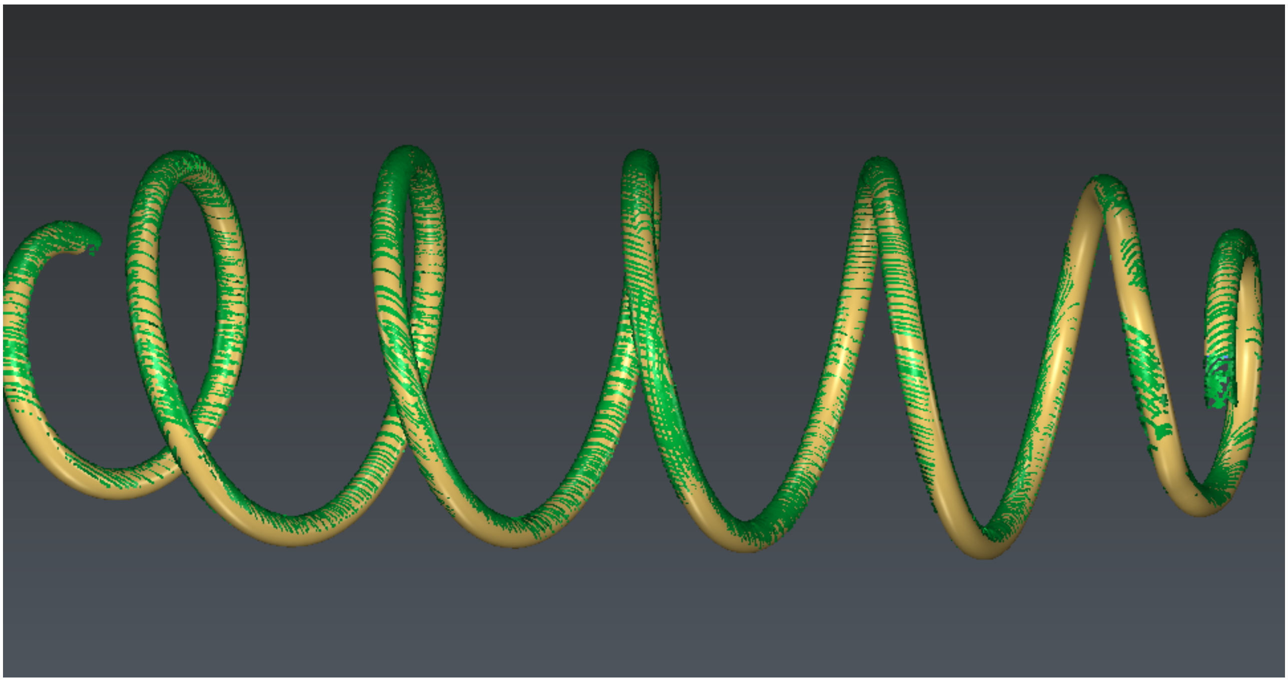
Enter **11mm** in the field **Approximate diameter** and click on **Preview**. You'll notice that the extraction is not perfect at the end of the spring.

Then, expand the **advanced** options, the **Calculation accuracy** value can be used to improve the accuracy of the result (for instance, slide the cursor up to two-thirds) to adjust the neutral axis at the end of the spring.



71 Neutral axis of a cloud

While computing the final neutral axis, it is also possible to create a mesh or a CAD surface corresponding to the shape. To do it, select **Mesh reconstruction** and/or **CAD object reconstruction** in the group **Outputs**.



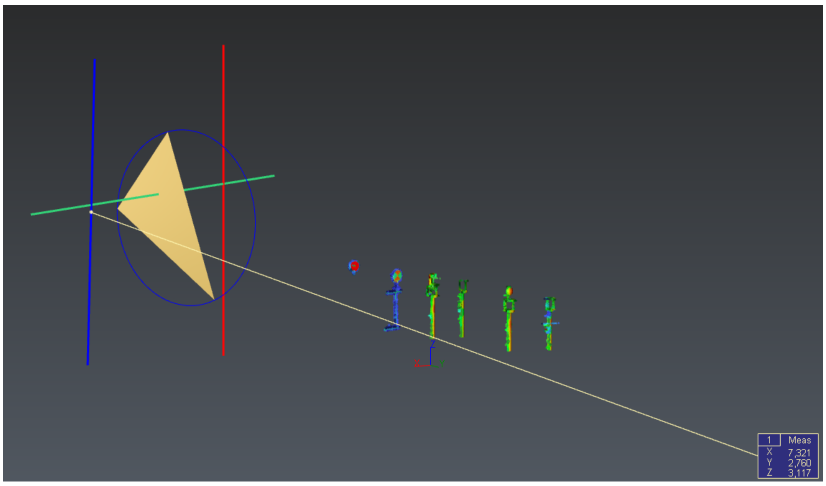
72 Mesh and CAD Reconstruction

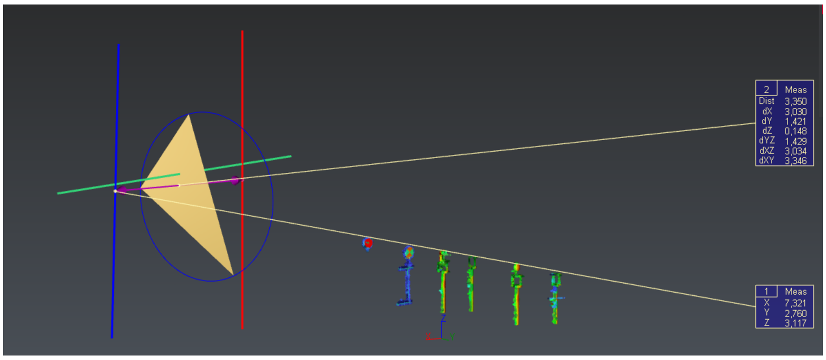
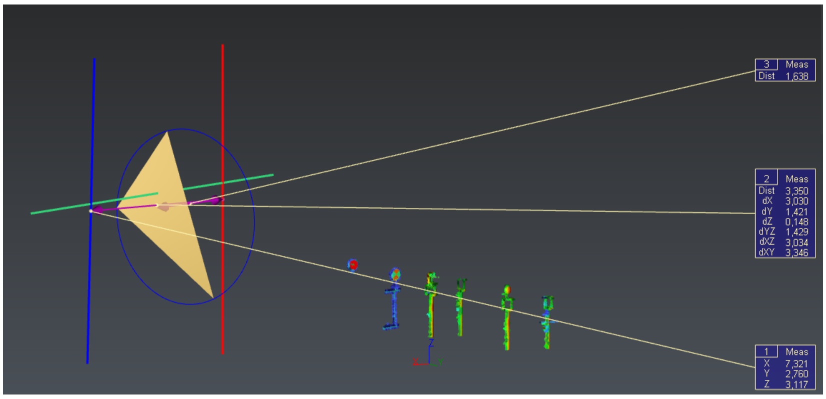
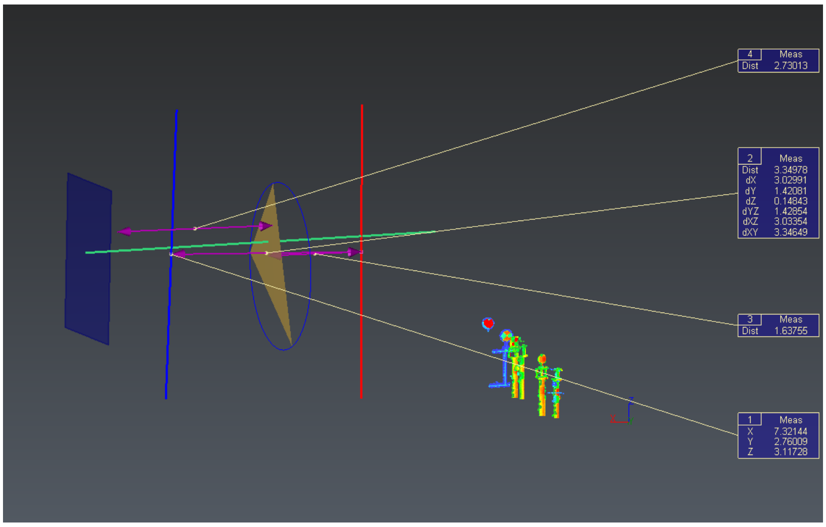
9 Analysis: Measurement, Inspection and reporting

9.1 *Make simple measurements*

Open the file **EnterPoints.3dr**.

There are different tools to measure coordinates, distances, angles or surfaces such as the tools listed below:

Launch the command Measure Point and select the middle of the blue line by choosing the **Middle / Center** option in the toolbar. Click **OK, Exit**, the command provides you the coordinates of the point and create a label (n°1) attached to this point.	

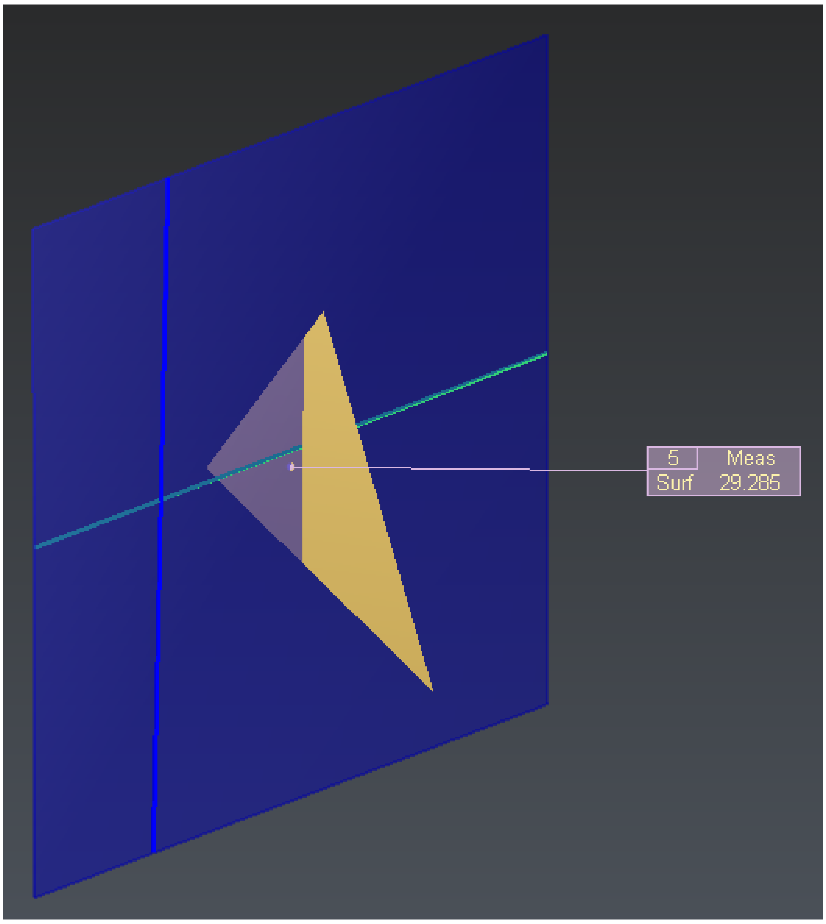
Launch the command Distance between Points and select the middle of the blue line and the middle of the red one. Click **OK, Exit**, the command creates a new label (n°2) and display an arrow to show the computed distance.	
Launch the command Distance Point Plane and select the middle of the red line and the triangle. Click **OK, Exit**, the command creates a new label (n°3) and display an arrow to show the computed distance. You can also measure the distance between a point and a mesh triangle (Distance Point Triangle).	
Launch the command Draw Plane and select the three vertices of the triangle. Validate the command. Now select the plane and launch the command Translation, choose X as **translation axis** and 3 for the **length**. Validate the command. Launch the command Distance between Planes and click on the created plane and the mesh. The command compute the distance between these two planes. Click **OK, Exit**, the command creates a new label (n°4) and display an arrow to show the computed distance.	

Select the triangle, the blue line and the green one, use right-click on the selection to open the contextual menu and select **Show Only**.

Launch the command Draw Plane again and create by clicking on the extremities of the blue line and the green line. Validate the command.

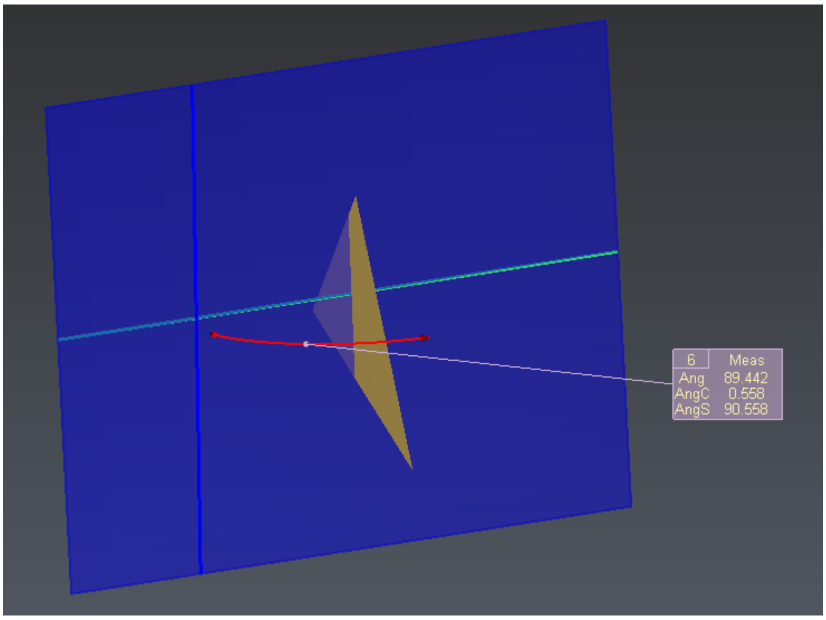
Launch the command Measure Surface and click on the plane, a label with the surface of the plane is created.

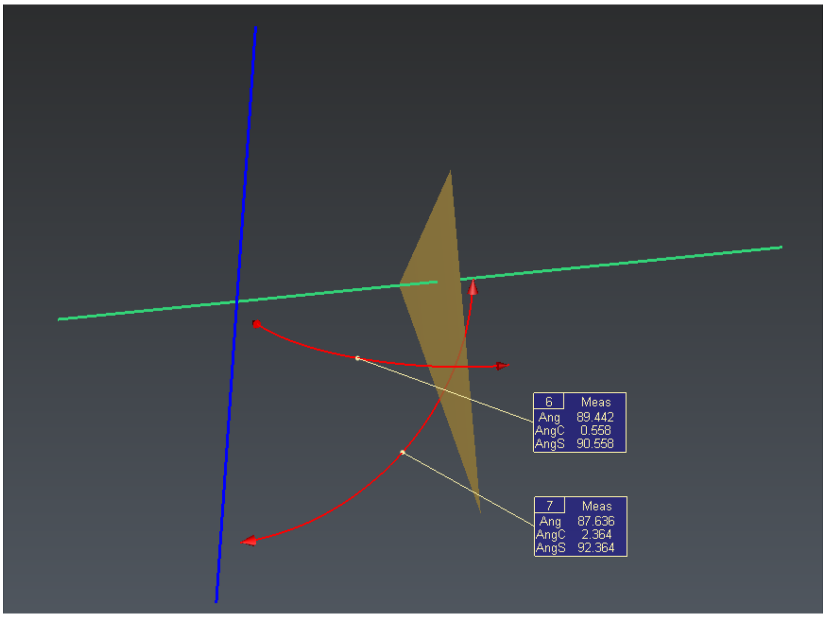
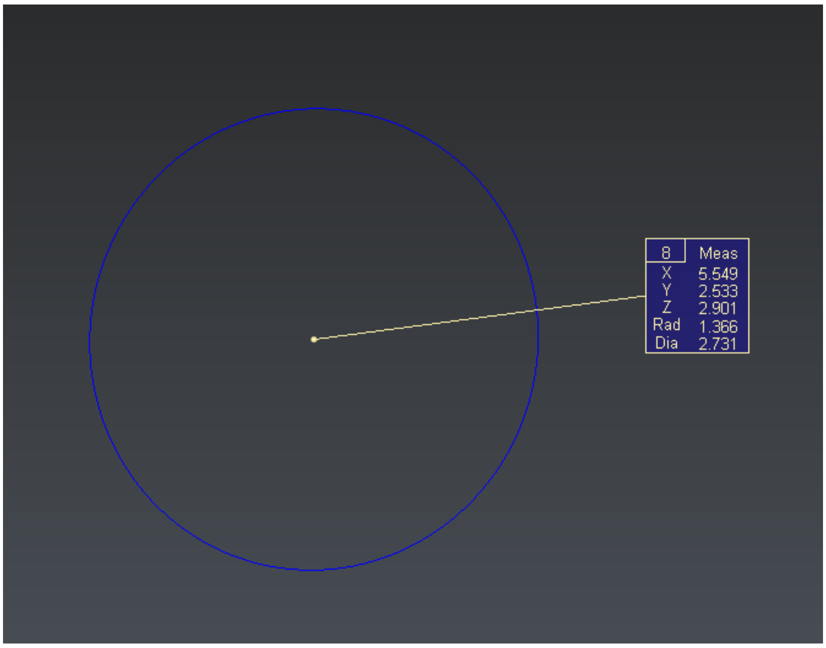
Click **OK, Exit**. the command creates a new label (n°5).



Launch the command Angle between Planes and select the plane and the mesh. Validate.

Click **OK, Exit**, the command creates a new label (n°6) and display an arrow to show the angle.



Now hide the plane and launch the command Angle between Lines and select the two visible lines. Click **OK, Exit**, the command creates a new label (n°7) and display an arrow to show the angle.	
Show only the circle and launch Measure Geometry. As there is no nominal data here, uncheck **Add nominal information**. Click **OK, Exit**. The command creates a new label with the circle definition.	

Measurement commands

In each command, you can choose data you want to keep in your label by checking/unchecking boxes.

Measurement tool in the toolbar

It is also possible to use tools to quickly make measurement. Thanks to the Measurement toolbar in the top right corner of the scene you can measure distance, angle between planes or angle between line faster, without launching a command in the ribbon.

9.2 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, rectangles, cylinders, spheres and 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
- Exercise: Extract features from a point cloud

9.2.1 Exercise: Create a geometric shape

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

- Draw
- Best shape
- Region grow

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

Draw a Circle:

- For example, go to Draw Circle and click points in the scene. Once three points are given, 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: click on the lock **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.

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

- Click on the lock **Center** and enter the point X=10, Y=5, Z=0.
- Click on the lock **Normal** and enter the vector X=0, Y=0, Z=1.
- Click on the lock and enter 2 for the **Radius**.

Using tools to define parameters

You can use the available tools to fix the center (Define points) and the normal (Define 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 check the properties with a right click.

9.2.2 Exercise: Compute best shapes from clouds and polylines

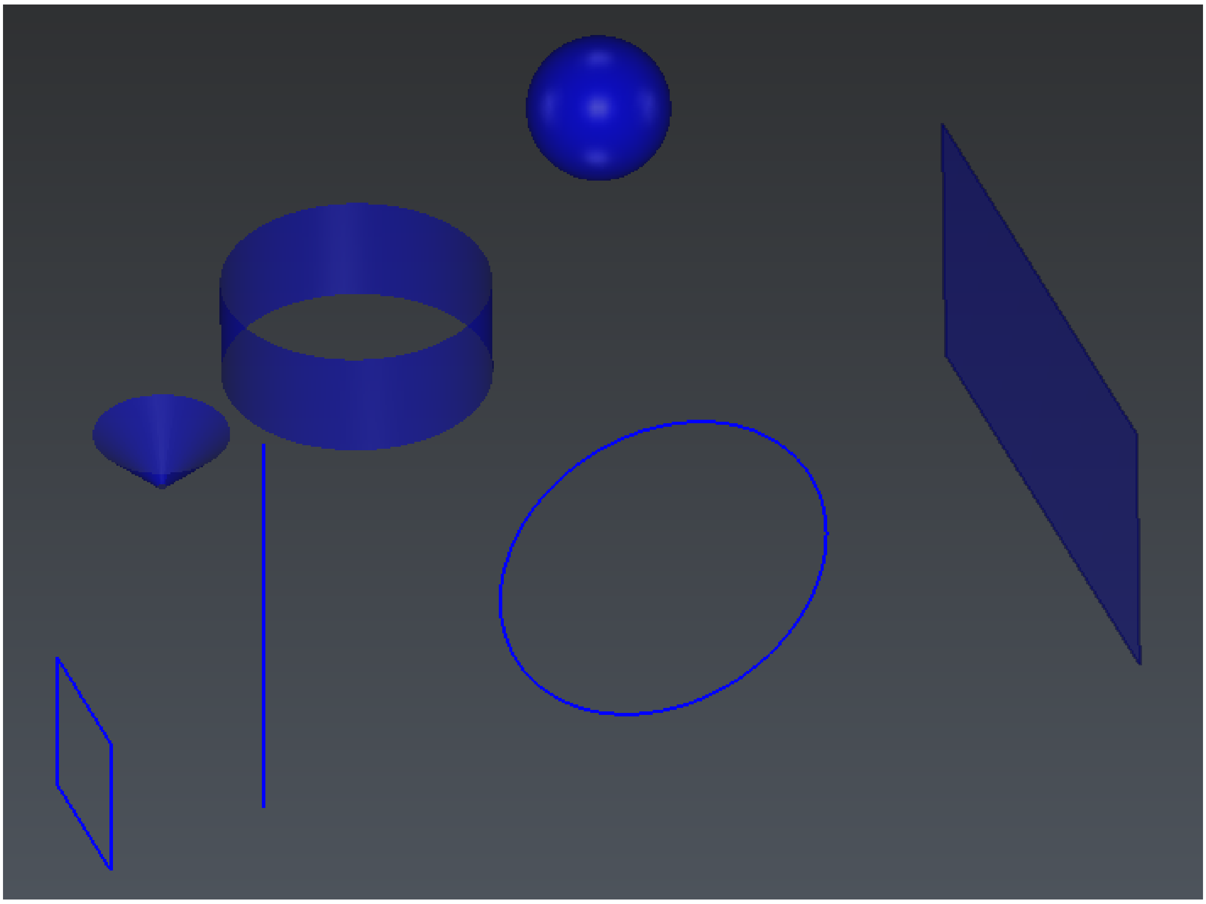
Beside the creation of geometric shapes by clicking points or by directly defining its parameters, you can extract a shape by fitting it to existing objects (meshes, clouds, points, other shapes...).

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

Open the file **AlignOnRef.3dr**.

Show only the folder **BestShapes**. then compute each corresponding best feature. For instance, select the line and open the dialog Best Line:

- You can eliminate noisy points by using the parameter **Eliminate worst points**: eliminate 10 points.
- You can lock some parameters: lock the direction to Z.



73 Best shapes computed from point clouds or polylines

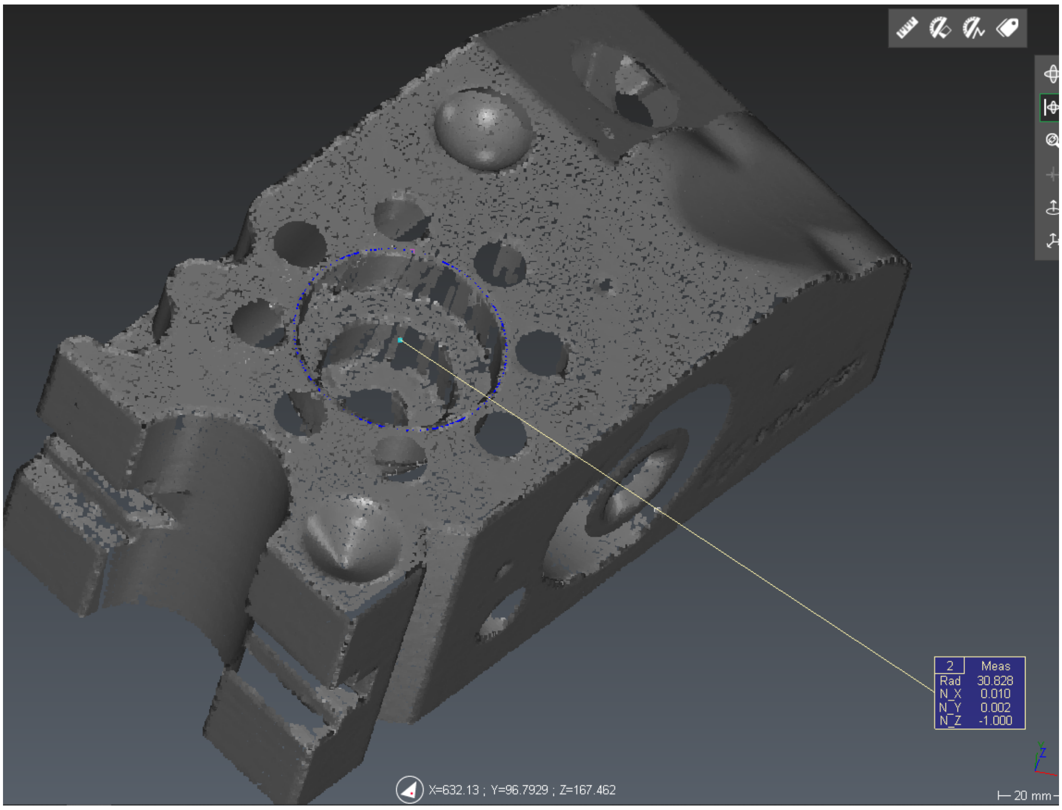
9.2.3 Exercise: Extract features from a point cloud

Unlike the best Feature computation commands which use all points of an object, the commands to extract features try to find the best feature from a seed of point(s).

Open the file **AlignOnRef.3dr**.

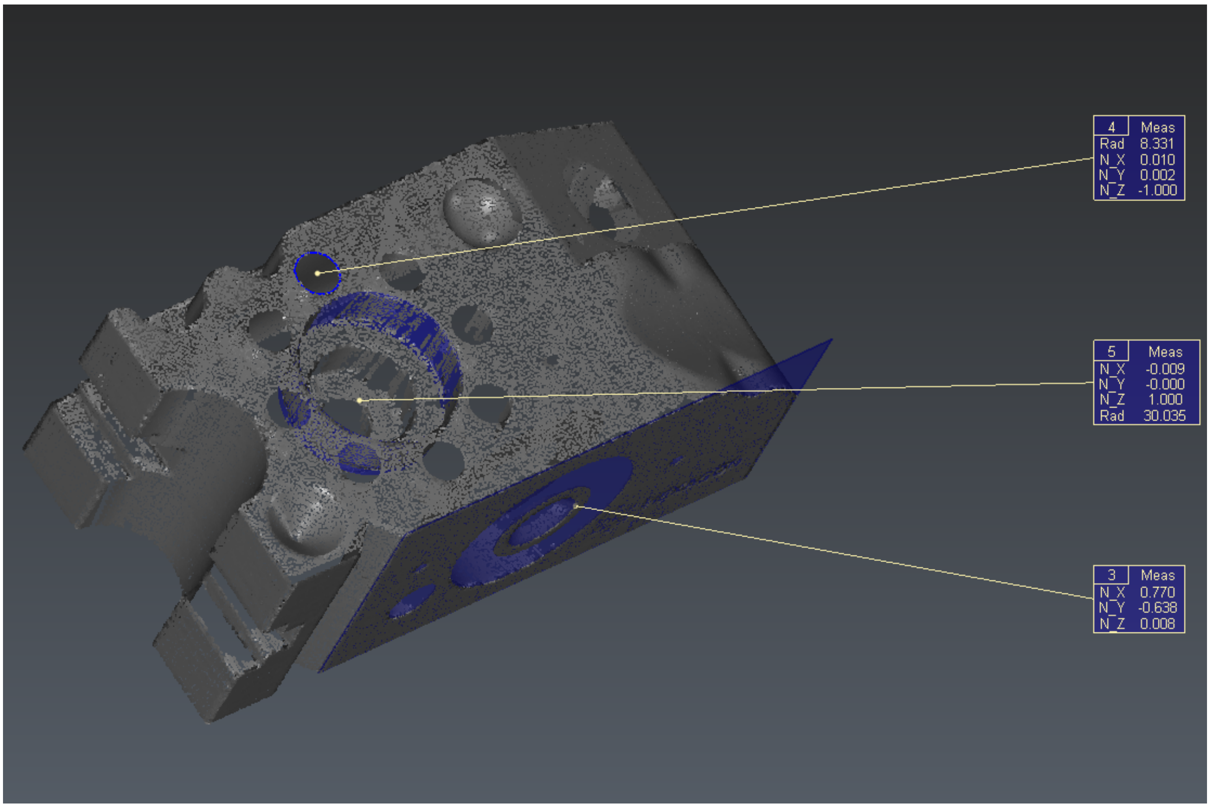
Circle

To extract the dimensions of a hole in this mechanical part, launch Region Grow Circle. Set the **Extraction tolerance** to 0.5mm and click a point close to the border of the hole.



74 Extract Circle - Region Grow

- You can set some parameters (Center, Normal, Radius). The distance between the extracted feature and the original points is visualized using the color gradient.
- You can generate a label or the extracted points if you wish by checking the corresponding options**.**
- Click **OK, Next** to validate the circle or **OK, Exit** to validate and exit the command.



75 Circle, cylinder and plane extracted from the cloud

9.3 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 cloud with another cloud.
- 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 objects (you can use commands Merge Clouds, Group Mesh or Group CAD (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

9.3.1 Exercise: Compute inspection between a surface and a cloud

Open the file **BestFitOnRef.3dr**.

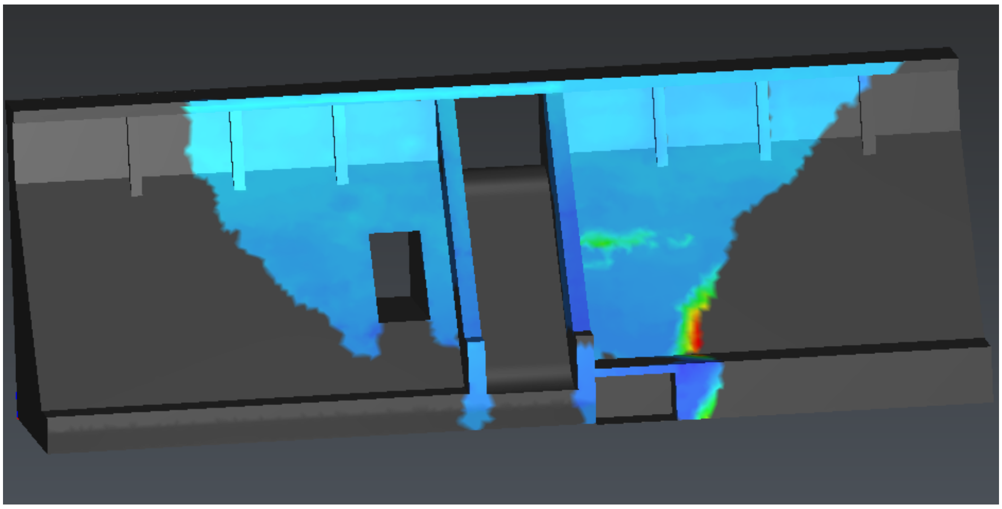
Show only the cloud **Aligned Dam** and the Mesh **Theoretical Dam (good CS)**. Select the mesh first, then the cloud with CTRL pressed, and go to Cloud vs (Cloud vs Mesh).

Choose to **apply colors** on the reference.

The option **Force projection direction** corresponds to a 2D inspection. It 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 **Comparison Theoretical Dam (good CS) / Aligned Dam** has been added in a group named Compare Inspect 2.



76 Comparison between a cloud and a mesh

9.3.2 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 cloud extracted from a best feature (commands **Best** or **Region grow)**, or
- a cloud or a mesh colored along a direction: Along Direction

Open the file **BestFitOnRef.3dr**

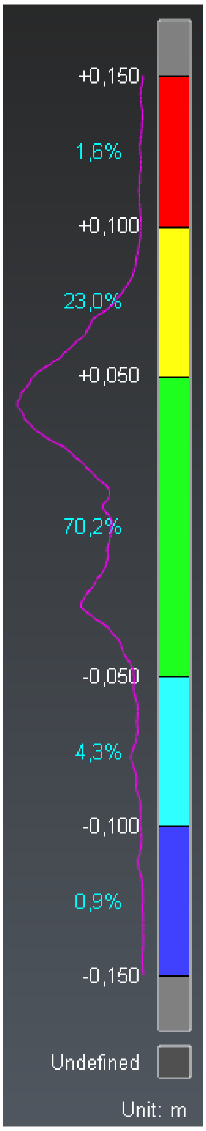
Select the result inside the group Compare Inspect and go to Edit Colors.

You can customize the color scale with different presets or create your own, and the interactions on the gradient directly. You can:

- change colors of each step,
- change the min and max values as well as the interpolation type,
- change the number of steps,
- etc...

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

- choose the **List of Values** preset and set the values to **0.1 0.05 -0.05 -0.1**
- choose **HSL Longest** for Interpolation



77 Edit color mapping
You can save or recall the customized color representation.

9.3.3 Exercise: Compute inspection between polylines

Open the file **CrossSections.3dr**

It contains:

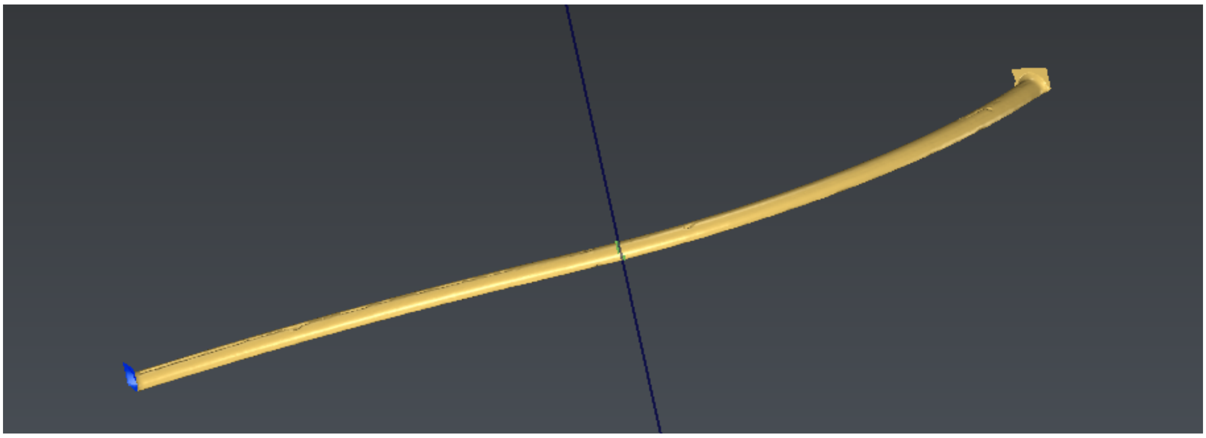
- the mesh of a **measured tunnel**
- a **theoretical section** of the tunnel at a given kilometric point

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 Planar sections,
2. Define the axis of the plane used to compute sections, choose the option and click on the theoretical section,
3. Select the option **List of values** and click a point on the theoretical section,
4. Click OK. A section named **Planar Section 232.26** is added in the Planar Sections group.



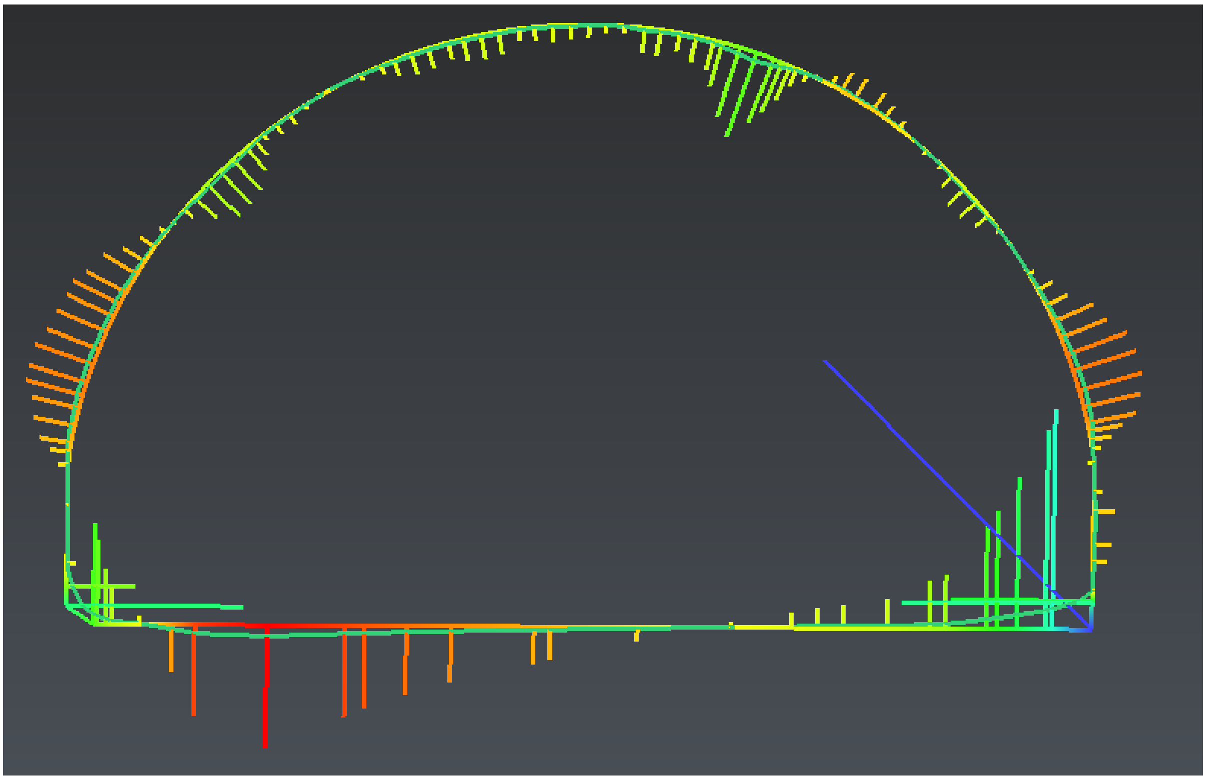
78 Section on the measured mesh

Inspect the polylines

Select and show only the two polylines **Theoretical section** and **Planar Section 232.26**. Select first **Theoretical section**, as it is the reference, and then select **Planar Section 232.26** with **CTRL**, as it is the measure.

Launch the command Compare Inspect Section vs Section.

1. Choose to apply the color on the reference object.
2. Click **Preview** to compute the inspection, then click on **Edit color** to magnify the distances. Set Magnify to 11.
3. Click **OK** to validate the display. Then, click **OK** again to validate the inspection. A new object called **Comparison Theoretical section / Planar Section 232.26** is added in the Compare Inspect group.



79 Inspection between two polylines

9.4 Labels & Reporting

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

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

Open the file **BestFitOnRef.3dr**.

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

Create labels

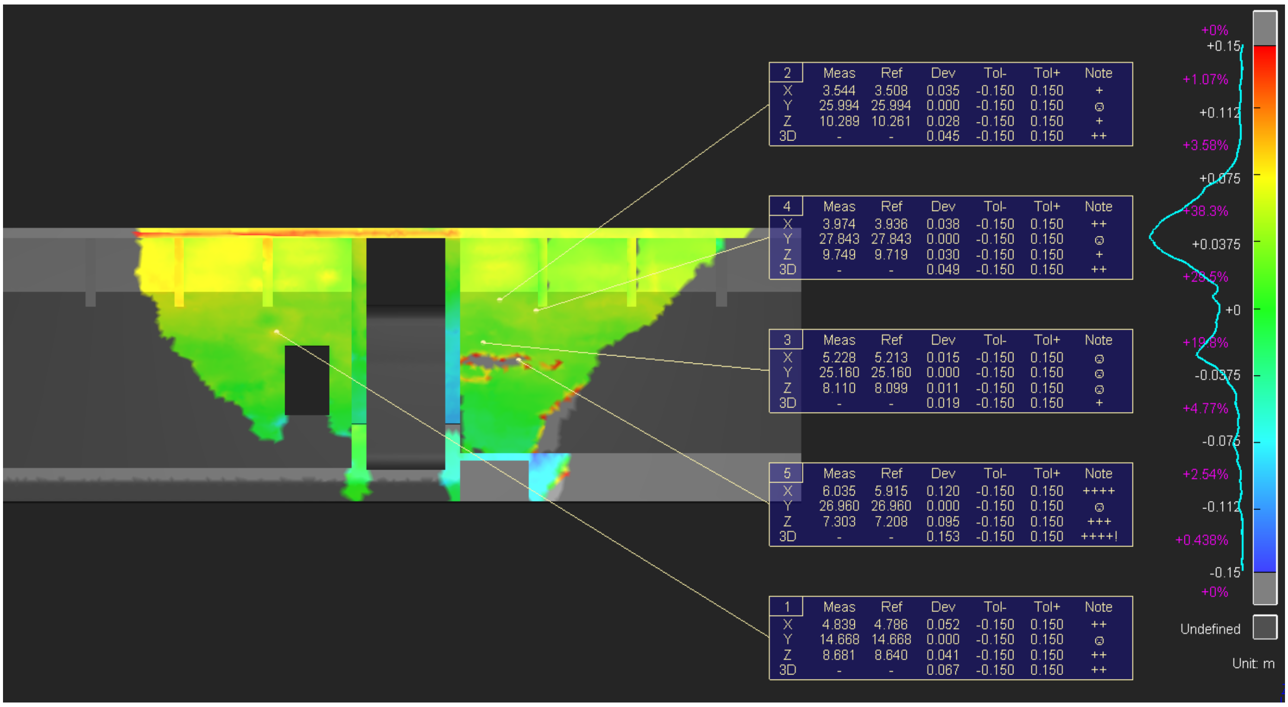
First, launch the command Settings in order to customize labels aspect. Select the label size **long**. Then, click **OK**.

Now launch the command Measure Deviation.

Then select the option **Point on Selection** in the software's central toolbar 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 label(s) you want to comment and launch the command Edit Label and enter a **comment** in the corresponding field in the bottom of the dialog box. Then click **OK** to validate.



80 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 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 Set.

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.

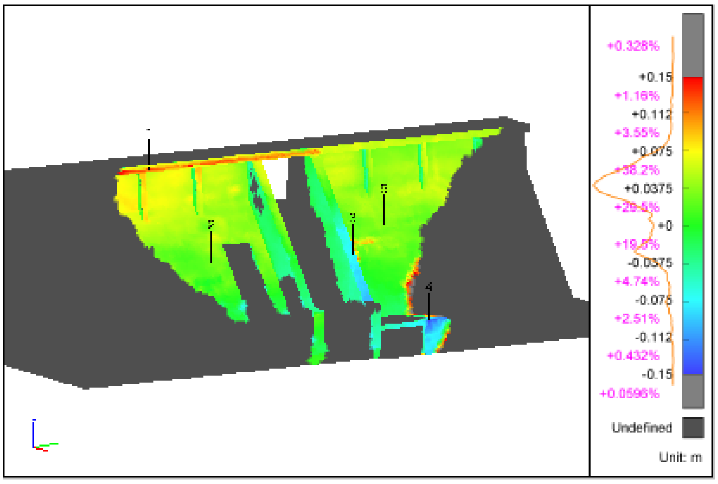
Launch Report Editor or launch the editor thanks to the magnifying glass corresponding to the report data. The chapter **Compare Inspect** has been automatically generated with the Template Library Settings.

First, define the Layout Panel (**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 that 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 columns: 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,
- optionally, insert another cell to display another scene (in 2D mode) using a view set 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	8.78807	23.59082	6.92487	8.84315	23.59081	6.98994	-0.07285	-0.15	0.15
4	12.24698	27.81801	2.17415	12.37047	27.81801	2.17487	-0.12412	-0.15	0.15
5	5.25829	28.28291	8.10025	5.22886	28.28291	8.07882	0.03494	-0.15	0.15

81 3D Inspection report with 3D PDF image

9.4.2 Exercise: Create a BIM inspection report

Import the data

Create a new document and import the IFC file **site.ifc**.

Type "Wall" in the **Filter** field, unfold the found folders, do a right click on the "Wall" node and choose **Load only selection**.

Finally, validate the edition: you have sub-selected components among a BIM project. Note the BIM project still contains the whole data.

> The BIM model can be edited later through the contextual menu (Edit).

Then import the scan **site_aligned.e57** in meter.

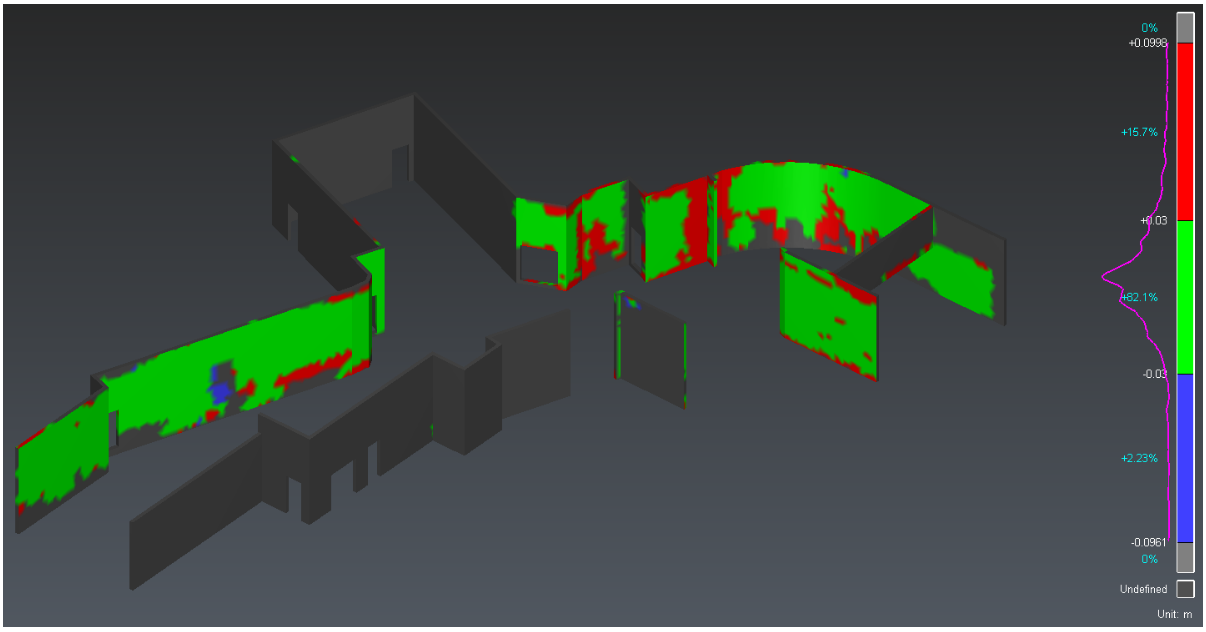
This scan is already aligned with the BIM model, so no further action is required.

Inspect

Select the cloud and the BIM and perform an inspection between a cloud and a BIM (Cloud vs).

Apply colors on reference (site BIM) and check **Max distance** to 10 cm. Press **Preview** to run the analysis.

Click **Edit color** and choose the **Presets Tolerance**. Define the tolerances (3 cm and -3 cm) and validate.



82 BIM inspection

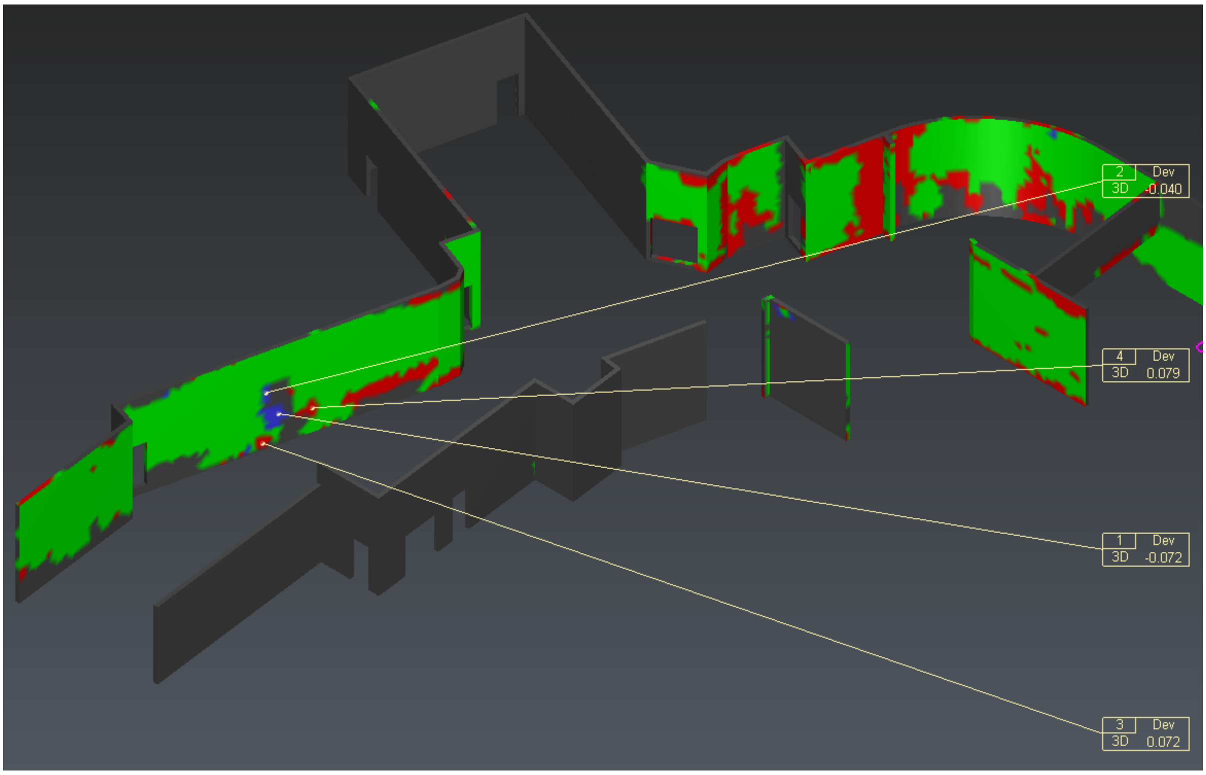
Annotate

Select the inspected mesh and launch Inspection Notes BCF.

Identify a blue or red area and add a note on it thanks to the toolbar at the bottom center of the scene.

You can describe the issue (Title, Description) and set its **Priority**. Then, you can add deviation labels and save viewsets.

Add a note on another wall defect. Note the wall ID (**BIM Component** field), stored in the source file, is automatically stored in the note.



83 Inspected BIM and labels assigned to an issue
Optionally, create another notes. Validate with **OK**.

These notes can also be exported to a BCF file.

Reporting

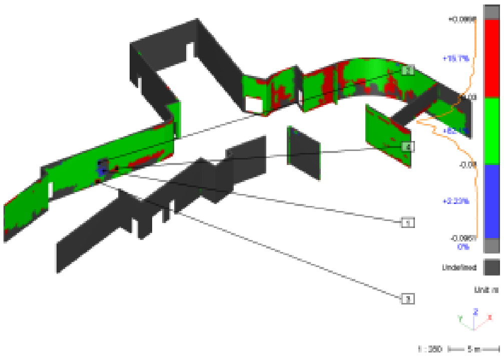
Open the report editor: a chapter called **BIM Inspection** gathers all inspection notes. Click **ToPDF** to generate the report.

BIM Inspection

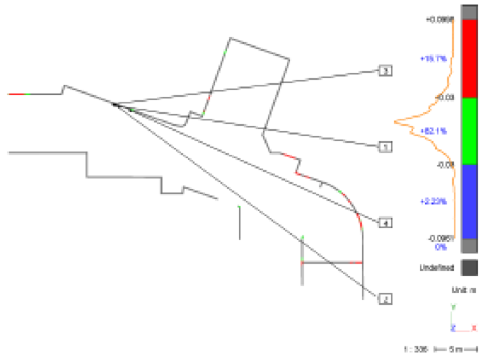
Note 1
First Defect

Summary
Inspected component: Basic Wall:Generic - 8" Masonry:248945
Comment:
Lack of concrete on first wall

3D Views



View 1



View 2

Inspection labels

Label number	Meas X	Meas Y	Meas Z	Ref X	Ref Y	Ref Z	Dev X	Dev Y	Dev Z	Dev 3D
1	241775.583	120241.834	28.740	241775.538	120241.756	28.740	0.024	0.068	-0.000	-0.072
2	241775.055	120241.972	30.002	241775.042	120241.935	30.002	0.013	0.037	-0.000	-0.040
3	241774.879	120241.917	27.548	241774.923	120241.984	27.548	-0.024	-0.067	0.000	0.072
4	241776.875	120241.193	28.413	241776.902	120241.267	28.413	-0.027	-0.074	0.000	0.079

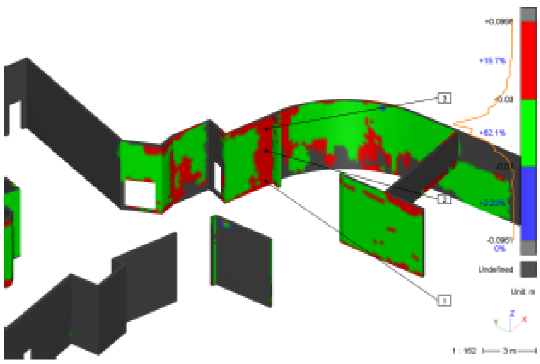
Images

Image 1

Note 2
Second defect

Summary
Inspected component: Basic Wall:Generic - 8" Masonry:247757
Comment:
This area needs work

3D Views



View 1

84 Inspection notes report

9.5 BIM Analysis

- Exercise: make a clash analysis report
- Exercise: BIM Progress Monitoring

9.5.1 Exercise: make a clash analysis report

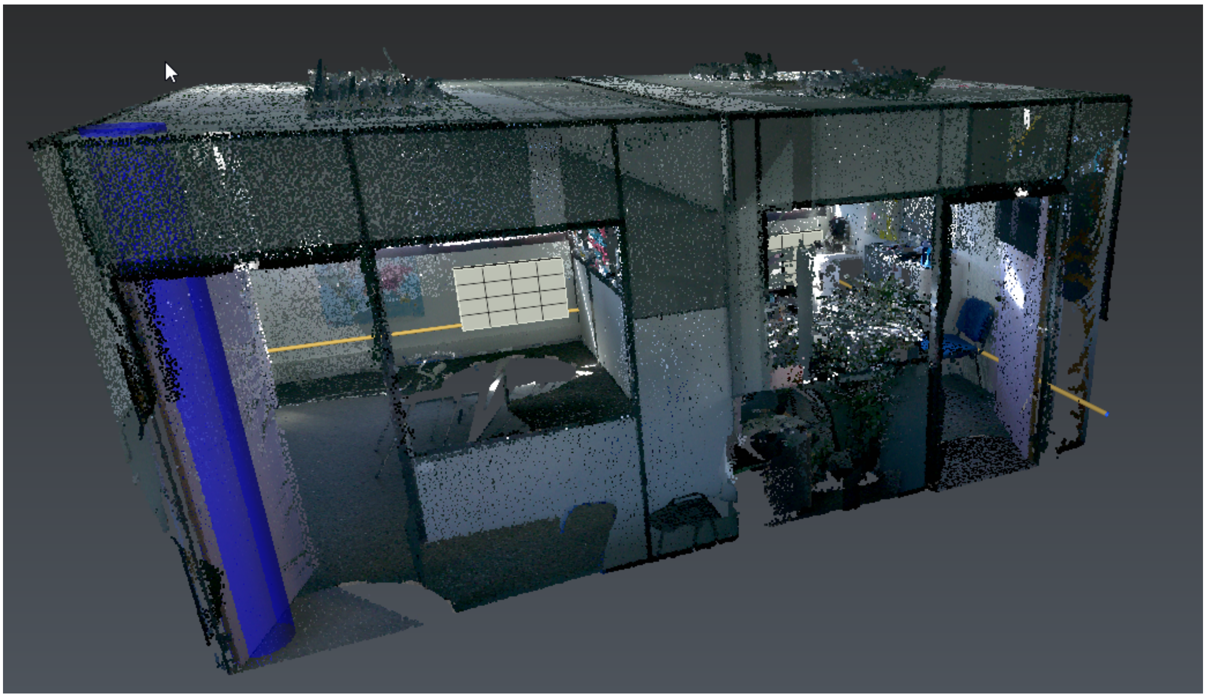
Clash Analysis enables to compute clashes between a cloud and objects, here, between two rooms and hypothetical changes to the building.

Open the file

Open the file **ClashAnalysis.3dr**. It contains a point cloud of a scanned office along with objects representing different planned modifications to the office environment. This file is going to be used throughout this exercise.

Compute clashes

The scene is composed of a 1.8 million points cloud and 4 other objects. Select all objects and the cloud and go to Clash.



85 Current environment and planned modifications

There are two main settings in this command: **Tolerance** and **Cluster Distance**:

- **Tolerance** is necessary to detect points which are very close to an object (but neither laying on nor inside). This parameter is also suitable to take into account the point cloud accuracy. Here, you can set this parameter to **0.01m.**
- **Cluster Distance** splits the colliding points in clusters. For instance, if a pipe collides with the floor and the ceiling, the clash analysis will give two clusters when the distance is lower than the height of the room. You can set this parameter to **0.30m**. This parameter is mainly suitable when analyzing long or grouped objects.

Click on **Preview.** With these settings, you get a list of clashing objects and their corresponding clusters:

Name	Status	Points	Comment
Air duct	Unknown	16588	
Clash 1	Unknown	15476	
Clash 2	Unknown	1112	
Radiator 2	Unknown	1281	
Clash 1	Unknown	1281	
Heating pipe	Unknown	677	
Clash 1	Unknown	343	
Clash 2	Unknown	254	
Clash 3	Unknown	29	
Clash 4	Unknown	27	
Clash 5	Unknown	24	

86 Compute clashes

Clash analysis

The view is split in two:

- on the left, a zoom on the selected object: here, the **Air duct**.
- on the right, a view of the whole cloud where you can localize the **Air duct** in its environment. Note, this view will be used in the report.

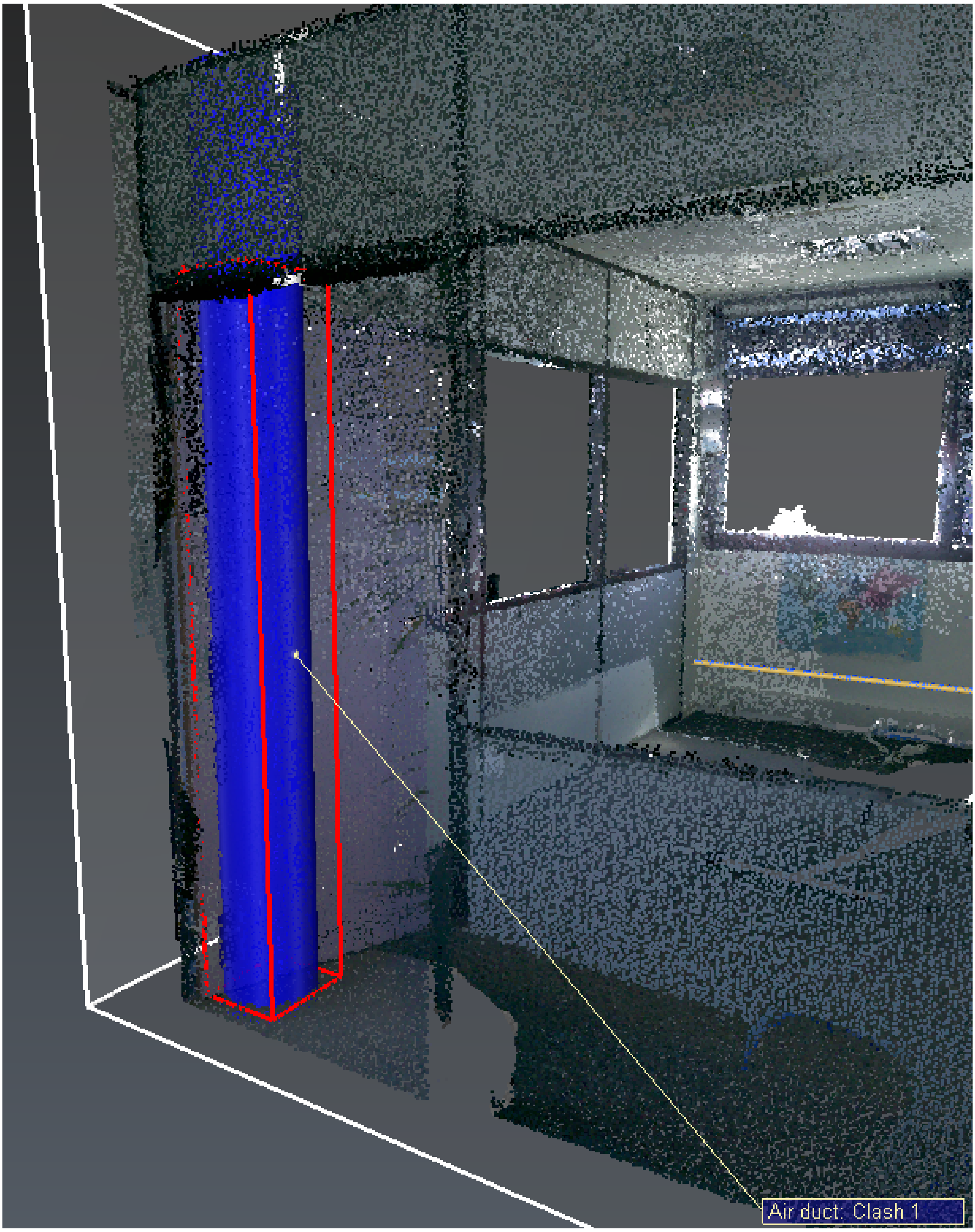
If you select a clash cluster within the table, left view will focus on it.

At this step, the status of all clashes are marked as **Unknown** by default. The clusters are displayed in orange (orange cloud in left view, orange box in right view). Now, you have to check each potential clash.

The **Air duct** has two clashes. Select the **Clash 1**, navigate in the right view: you will see that **Clash 1** corresponds to a door collision. You probably had better to move that component elsewhere, so press **SPACE** to change the status to **Clash** (or click on Unknown). Note the cluster is now displayed in red. Click on **Update Clash View** to replace in the report the default top view by the current one.

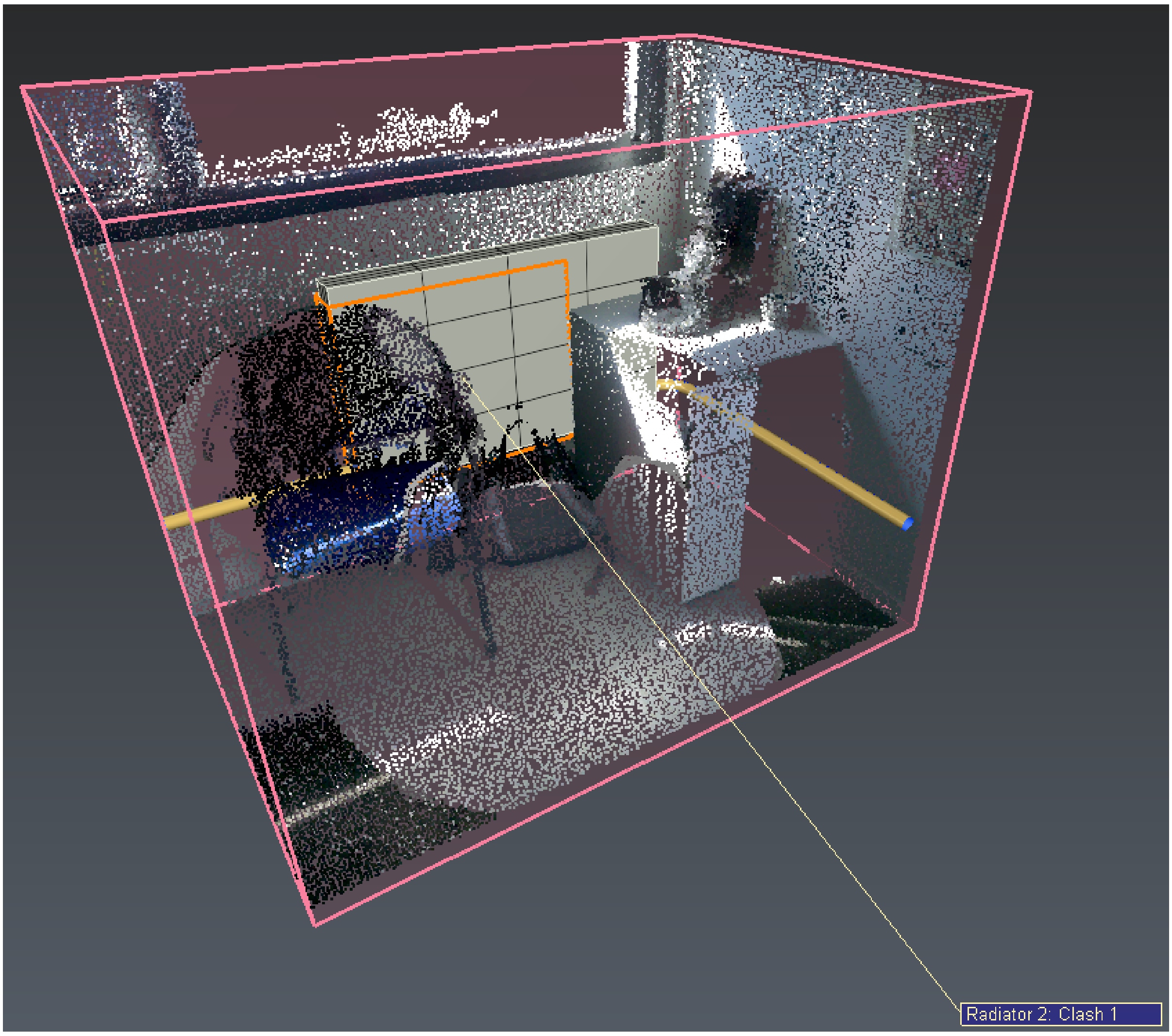
Now, you can optionally add a comment referring to this clash, which is useful for adding practical information about each clash. Click in **Comment** column next to Clash1 and write "Move the air duct". Notes can be associated either with the object or with the clash cluster.

Press **DOWN**, Clash 2 corresponds to the ceiling, set the status to **No Clash** (the air duct continues inside the ceiling). The cluster is now displayed in green but the **Air Duct** stays in Clash due to the **Clash 1**. Update again the clash view and optionally add another note.



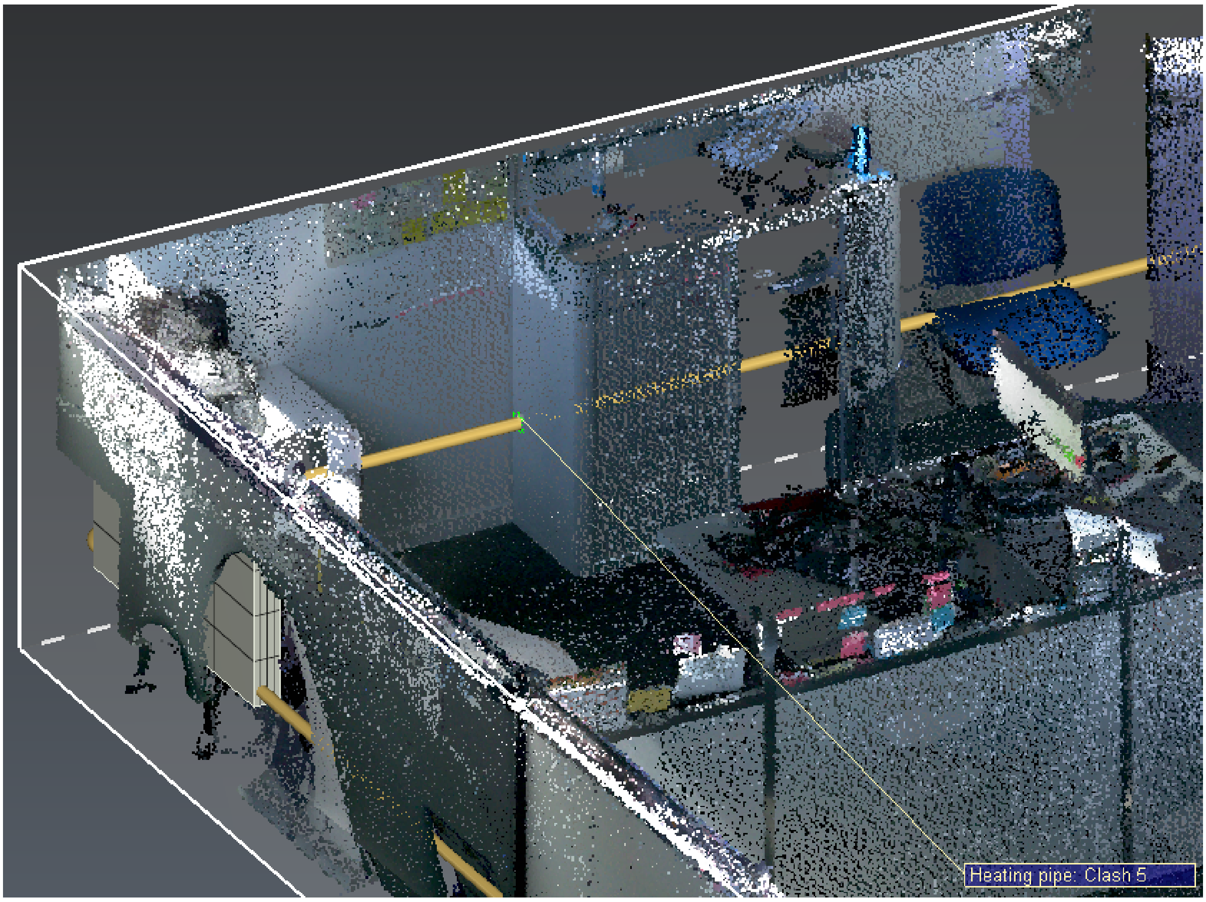
87 Air Duct: Clash 1

Radiator 2 has 1 potential clash with 1281 points. Click on **Clash 1**. The right view has a limit box: press **CTRL+SPACE** to edit it. Reduce the box to the area of interest. In fact the radiator collides a seat and a bag: **No Clash**. Update again the clash view and optionally add another note to the object (not to the clash).



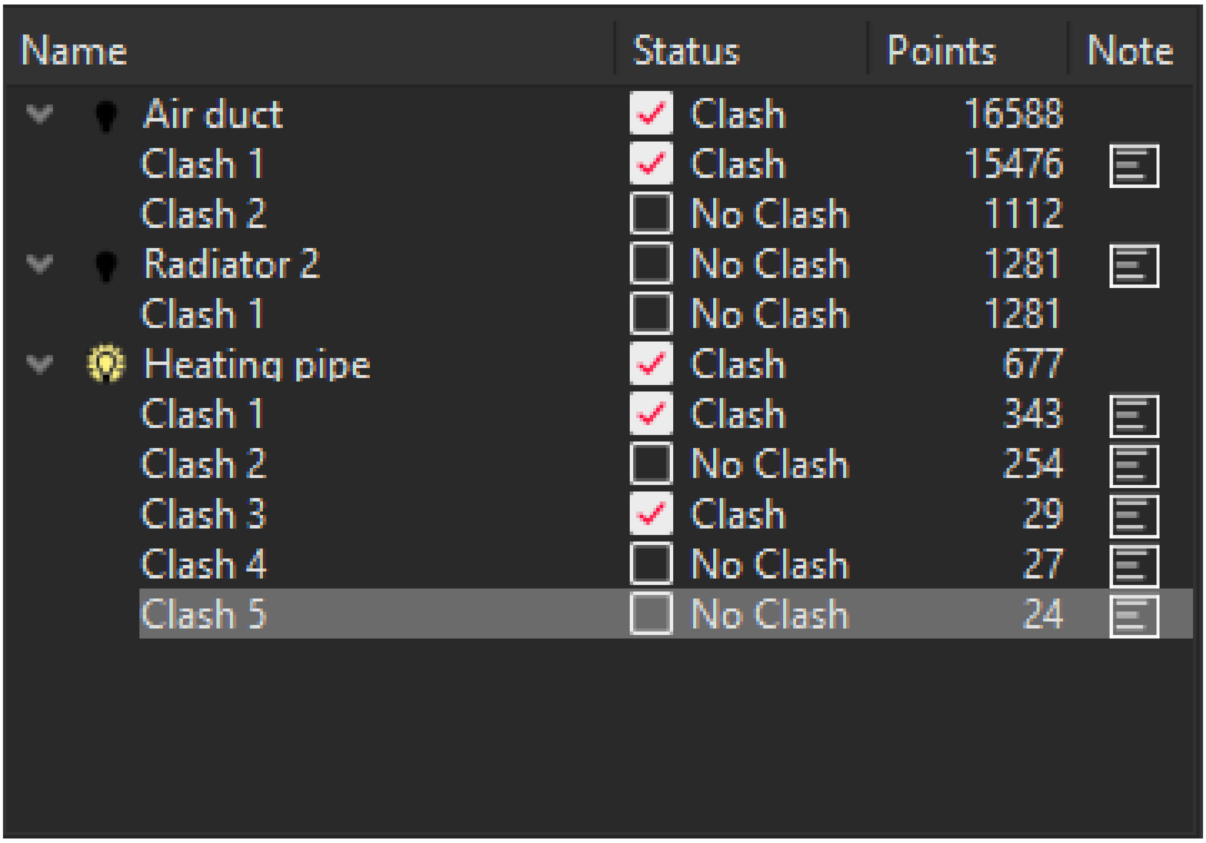
88 Radiator 2: Clash 1
Heating pipe has 5 potential clashes with 677 points. Repeat the same process for each clash:

Clash	Status	Note
1	Clash	Drill the wall
2	No Clash	False clash (bag)
3	Clash	Drill the wall
4	No Clash	Move the closet
5	No Clash	Move the closet



89 Heating pipe: Clash 5
Radiator 1 has no potential clash.

It may happen that your objects clash with the cloud by only a few points. According to the accuracy and density of the point cloud and the wanted level of details, you can filter irrelevant clashes with **Filter clashes** option.



90 Analysis complete
The analysis is now complete: click **OK** to validate and exit the command.

Create and export the report

Firstly, save a view to insert inside the report. Display the project as you want. Within the tree-view, click on the magnifying glass next to the Report data and **update** the main view set.

Open the Report Editor (report menu), you will find a new chapter called **Clash**.

The first page summarizes the parameters, the inputs and outputs of the computation: the Clash Tolerance, the Cluster Distance and the list of analyzed objects, as well as tables named "Clash", "No clash" and "Unknown" containing objects according to their Clash status.

> **Drag a valid data table here.**
>
> If you see on the summary page "Drag a valid data table here", do not worry, it occurs when no object belongs to the given category and this note won't be exported to the final PDF.

The second page lists all the object with the status clash and unknown. You should have 3 different clashes now, as displayed in the top left corner. Use the red arrows to navigate between clashes. Each clash has its page, which is composed of the object name and the clash number in the title, the clash status, the number of clash points, the note you write and a general view of the clash.

Missing binding!

If you see on the page "Missing binding!", do not worry, it occurs when no note has been written on this particular clash and this note won't be exported to the final PDF.

To change the view of the clash present in the report, go back to the command (after selecting **Clash Analysis** in the objects list) and select the clash you want to update. Then change the right view to the point of view you want and click on **Update Clash View**.

Add a limit box

While editing a clash view, you can also edit the limit box by using **CTRL+SPACE** and scaling, moving or rotating.

Once you have edited your report, you're all good to go on and export it! Click on **To PDF** in the toolbar to print the PDF.

91 Report

9.5.2 Exercise: BIM Progress Monitoring

Progress Monitoring allows you to track the progress of a construction project by comparing the BIM model to the scan data.

Open the file

Open the file **ProgressMonitoring.3dr**. It contains two point clouds of a construction site as well as the corresponding BIM model of the beams waiting to be built. The clouds have been scanned at intervals of one week.

Compute progress

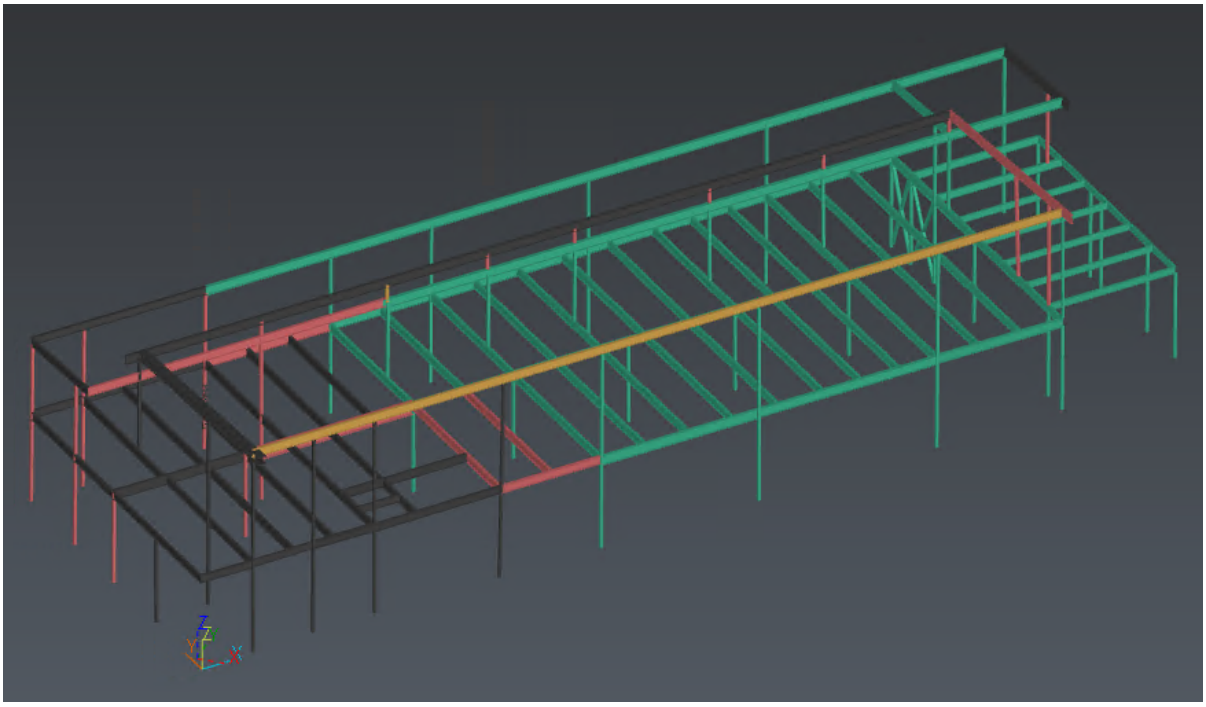
Each sub-element of the BIM file will be individually compared to the point cloud; so it is recommended to remove the unnecessary elements such as screws or ground for example. This will also reduce the computation time.

Select the cloud **Week1** and the BIM **Model structure** and go to Progress Monitoring.

The first step of this workflow, Analyze, will compute the % of completion for each sub-element. Here you can adjust three parameters:

- The **first bar of the Coverage slider** (red to yellow) sets the coverage threshold for considering the sub-element as **"Incomplete"** and no longer as **"Not installed"**. Leave the default value 5%.
- The **second bar of the Coverage slider** (yellow to green) sets the coverage threshold for considering the sub-element as **"Installed"**. Leave the default value 35%.
- The **Tolerance** is necessary to filter out points, by distance, which don't match any sub-element. This parameter is also suitable to take into account the point cloud accuracy. Here, you can set this parameter to **0.05m**.

Click on **Compute**: the inspection is launched. The sub-elements are classified in four statuses: **Installed**, **Incomplete**, **Not installed** and **No data** (green, orange, red and grey respectively).



92 Week 1 progress

> **Pie chart**
>
> You can hover your cursor over the portions of the pie chart to see the ratio of objects classified in each status.

You can see that most of the beams are marked as installed. Click on **Next** to go to the Check step of the workflow.

Check the results

Now, you can find on the left a tree containing all the sub-elements. By default, each node content is sorted by growing coverage.

On the right, two views: the left one displays the overall **Progress View**, and the right view shows specially the selected sub-element.

Click on a sub-element, whether in the tree or in the left view: the right view displays the element in the inspection mode and its matching points.

If you think the thresholds were inappropriate, you can come back to the Analyze step with the **Previous** button and **Reset** the analysis to compute a new one. If necessary, you can modify the automatic classification: select the sub-element you want to edit and press **SPACE**. Check up the notifier to know the key shortcuts.

Finally, you can click on **Next** to go to the final step of the workflow: Export.

Export a report

This step will allow you to generate the reporting data and optionally to export a BCF file containing issues to be imported in your BIM platform and software.

Click **Update Report View** to define the scene view that will be included in the report and in the BCF.

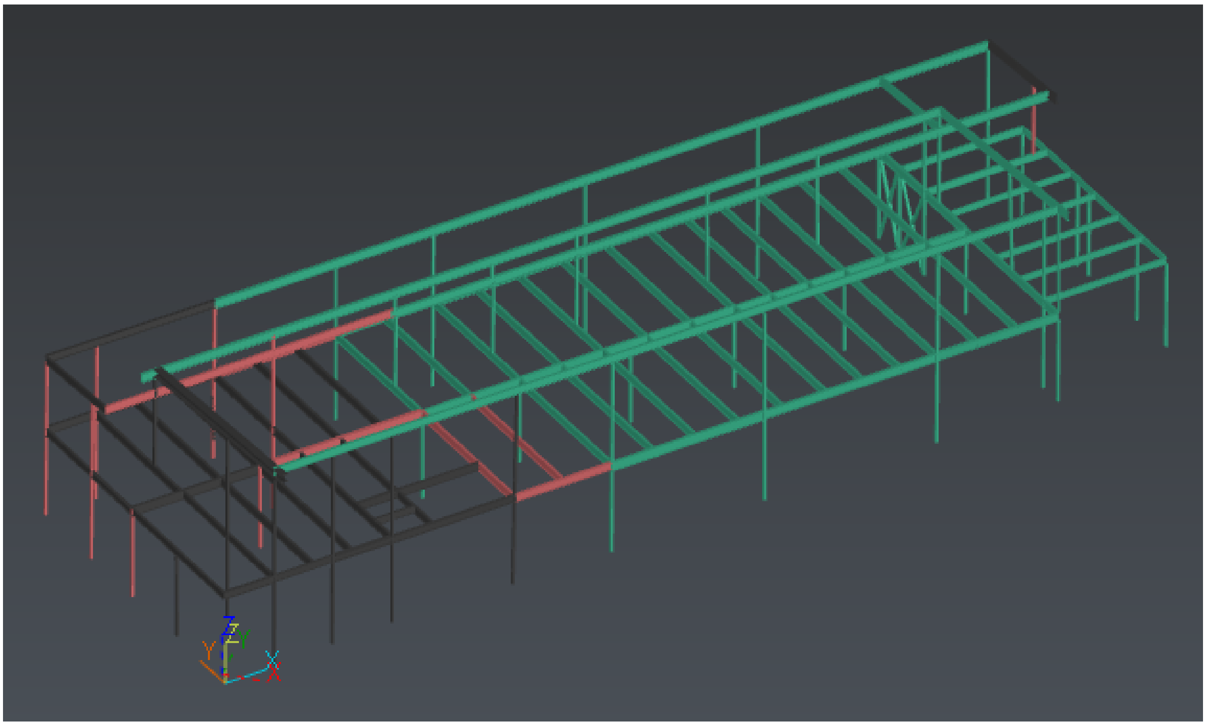
If you want to export a BCF, assign the issues to someone and choose between exporting only the **Summary** or **Summary + detailed issues**:

- The first one will only export one issue with synthetic data of the analysis.
- The second will create a BCF with an issue per analyzed sub-element. You can choose not to export specific statuses to only keep track of what you're interested in.

You can now **Exit** the workflow and go to the report editor so as to generate the PDF report. Note you can edit the analysis by clicking on the **Play** icon in the explorer tree next to the Progress Monitoring folder.

A week later

It is time to check again the progress. Select the BIM object and the second cloud, called **Week2** and repeat the analysis with the same parameters.



93 Week 2 progress

10 Analysis: Surveying

10.1 Cross sections

10.1.1 Exercise: Tunnel analysis

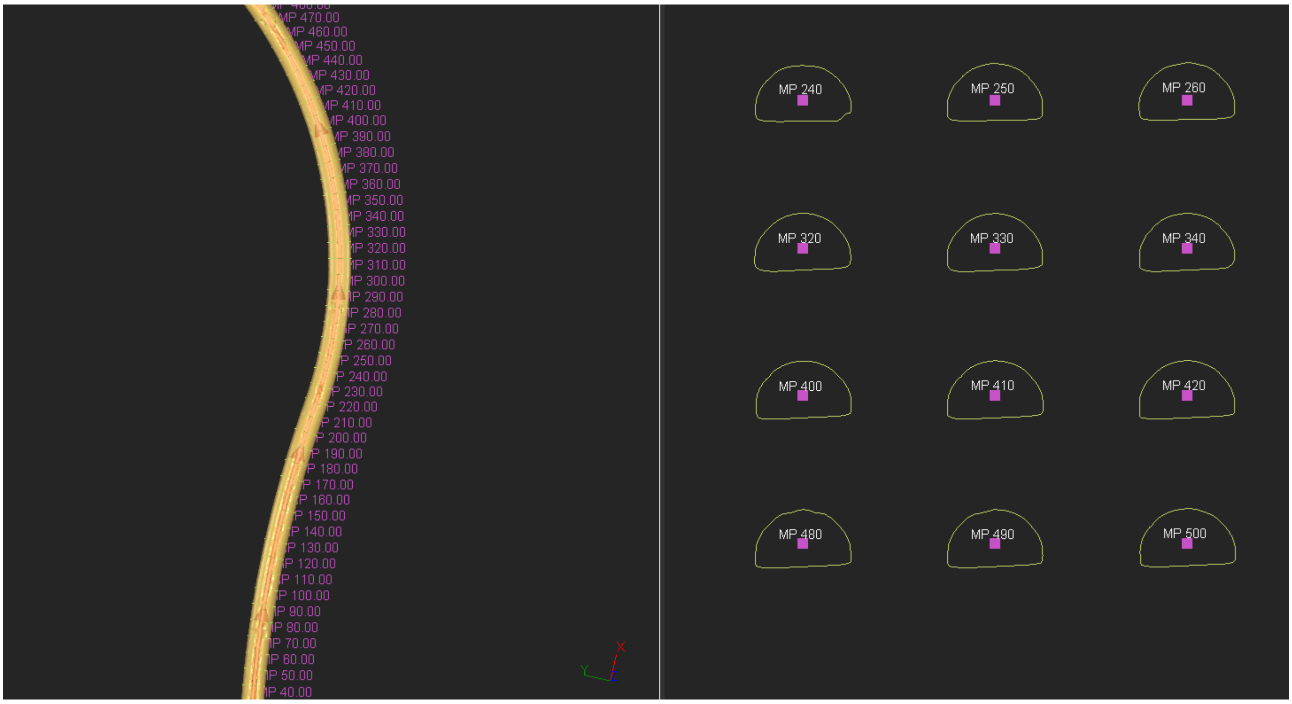
In the surveying field it is common to draw cross sections on a building, on a road or on a structure, in order to inspect it while it is being built or for periodic controls. The following commands have been developed taking into account the specific needs of tunnel inspection. They can of course be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

Open the file

> Open the file “CrossSections.3dr”. 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 Create Profiles 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.



94 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 names composed of an optional prefix and their curvilinear distances on the axis. In this example we have **MP** as a prefix meaning "Milepost". You can write your own prefix in the box **Prefix**.

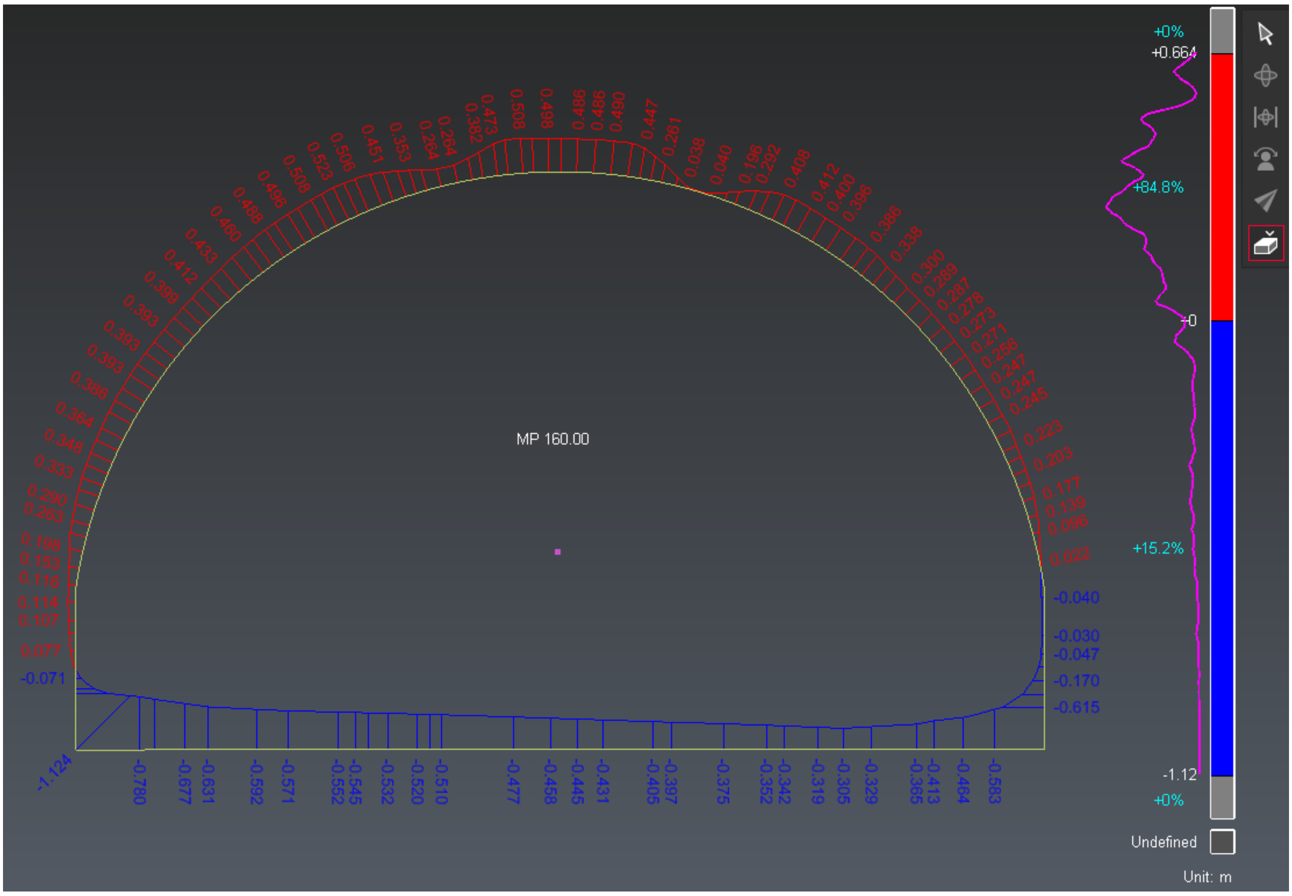
> ⚠ From this dialog box it is possible to export the created sections in their 2D layout by clicking on **2D Preview / Export**.

Click **OK** to validate the results. You can see a new folder 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.3dr". 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 Create Profiles 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 Compare / Inspect Profiles. You could also click first on **OK** to validate the result and then select the created folder containing all the sections and go to Compare / Inspect Profiles.

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.



95 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 Deviations** diagram to display hairs,
- check the option **Show Quotations** texts to display them in the 2D layout,
- check **Show Gradient** 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 **Highlight 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 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 Report 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.

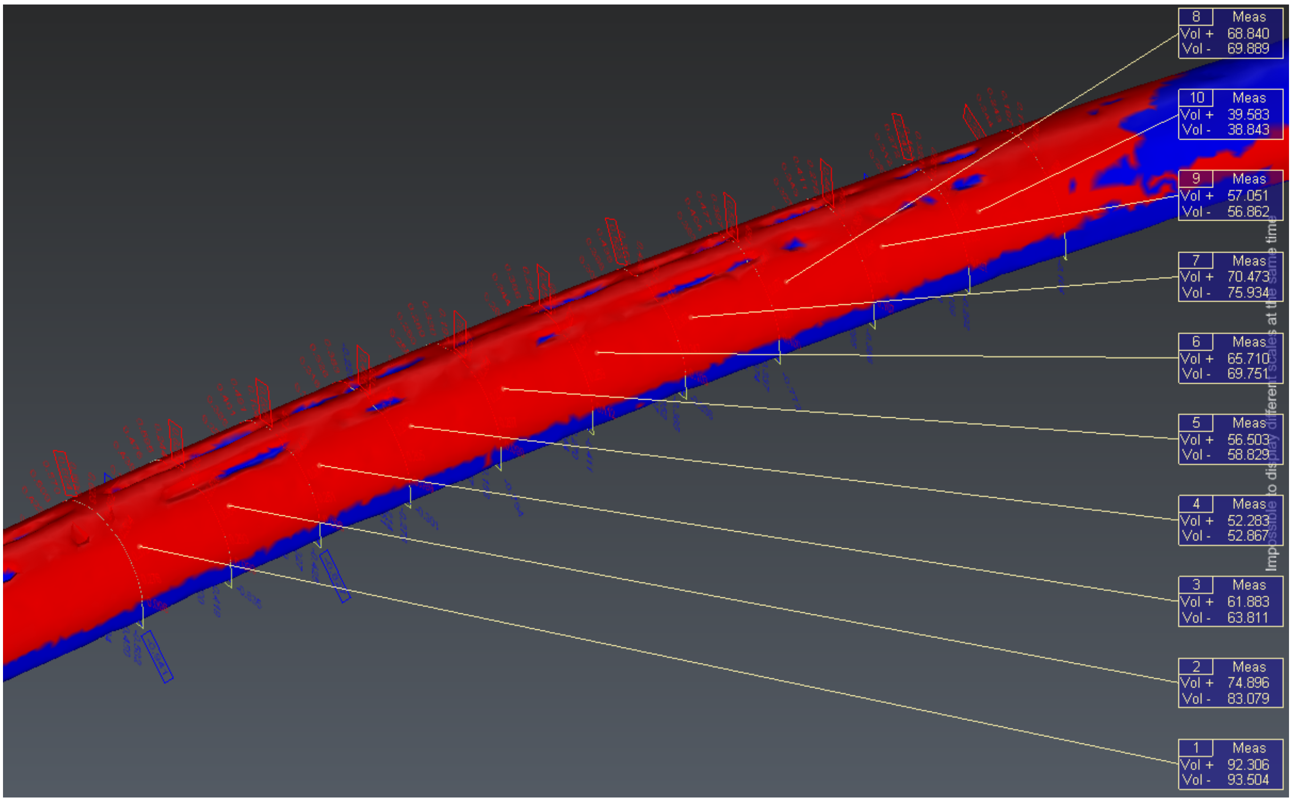
1. define the paper format and orientation in Layout panel: A4+Portrait. Reduce the number of decimals to apply to distances. Apply header and footer everywhere.
2. complete header and footer with logo, date, customer or company name, title, page number, etc. (refer to Template View). You can modify 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.

96 Print cross sections in a dedicated template

Compute the volumes in overbreak and underbreak of a tunnel

Keep using the file “CrossSections.3dr”. We are going to compute the volumes in overbreak and underbreak between the measured tunnel and the theoretical tunnel on specific sections.
Select the theoretical mesh, the measured mesh, the axis and the folder containing the sections previously created, then go to Volume Over/Under.
Choose which mesh is the reference. Here, select the Theoretical tunnel. Set the **Step** to 0.5m. 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 computed between the couples of cross sections according to the Step value.
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. Detailed computation can be added to the report.



97 Compute volumes of overbreak and underbreak

10.1.2 Exercise: Draw a longitudinal profile

In the surveying field, it is common to draw ground sections following an axis. This can be achieved using the command Unroll along Axis.

Open the file

Open the file **TextureParam&CameraPath.3dr**. It contains a mesh and a polyline. This file is going to be used through this whole exercise.

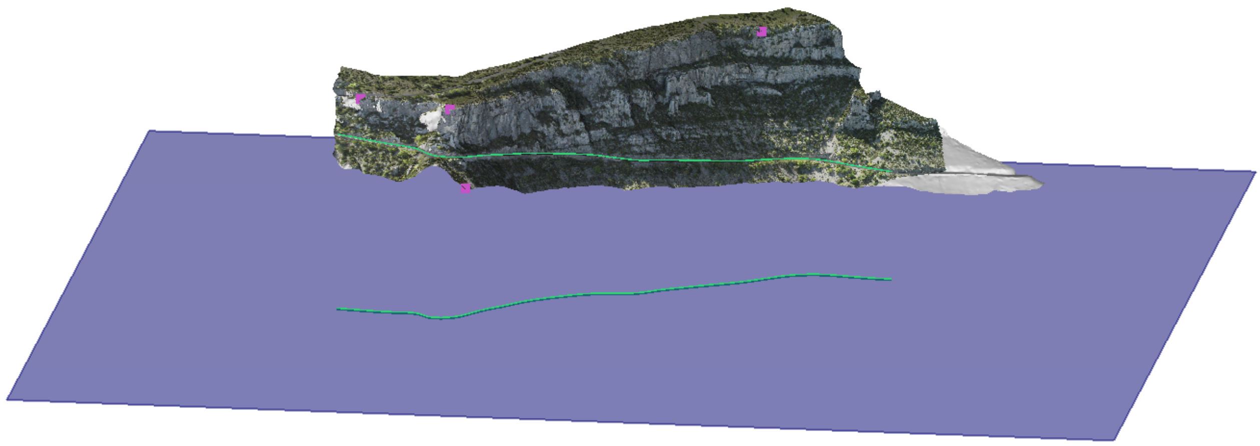
Project the axis onto the ground

To make quotations later, we first need to create a reference plane. Launch the command Draw Plane and define a horizontal plane (**Fixed normal**: X=0; Y=0; Z=1). Set the **Fixed center** by clicking one point on the mesh, then modify its Z coordinate to 400 for instance. The plane **Width** and **Length** can be set to 1000m in this example. Lock **Main axis** to complete plane definition. Click **OK, Exit**.

Now, we are going to project the polyline called **Camera path** onto the mesh **CliffTextured** and onto the plane.

Select the polyline and launch Projection. Click on the mesh and define a vertical projection direction. Check also **Both sides** and Follow the mesh shape by checking the option **Add points to better follow the shape**. Click **Preview** and **OK, Exit**.

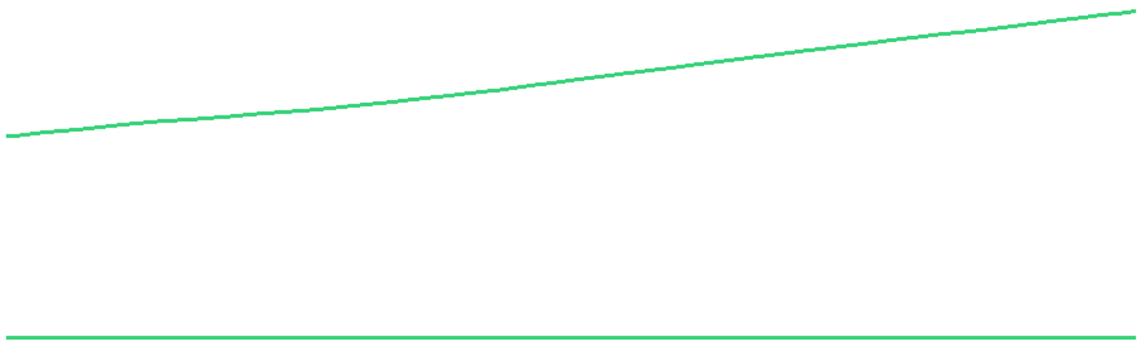
Repeat the same workflow to project the polyline called **Projected Camera path** onto the plane. Thus, you will create two polylines with the same number of vertices. This will help you creating quotations.



98 Fig.1: projected polylines

Unroll the polylines

Select both polylines and launch the command Unroll along Axis. Then, you have to choose the polyline which will be used as parameter for unrolling (in other words, the axis). Here, both stand for the axis. Click **Preview** and **OK**. Show only the polylines which have been added to the folder **Unroll along axis**. Press **Shift+Y** to display the profile from the correct direction.

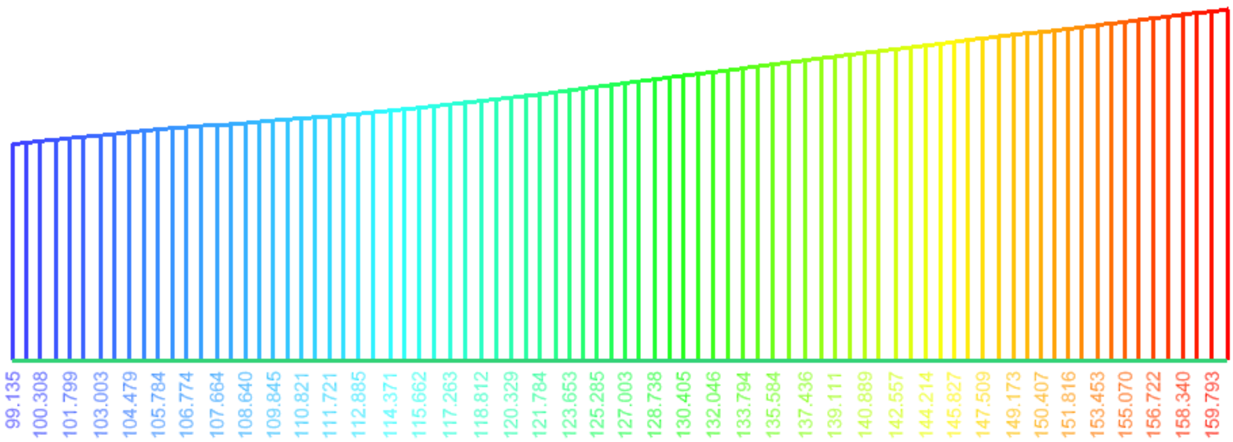


99 Fig.2: unrolled polylines

Add quotations

Select both polylines starting with the upper profile and launch an inspection between sections (vs Section). Here, you can make a **Point to Point comparison**. That is why we have created two polylines with the same number of points. Choose to color the upper profile.

Now, click on **Edit color** in order to check **show quotation**, to modify the **number of decimals**... Note the quotations stand for the heights above the reference plane (Z=400).



100 Fig.3: longitudinal profile

10.2 Surface analysis

- Exercise: Complete analysis of a concrete floor

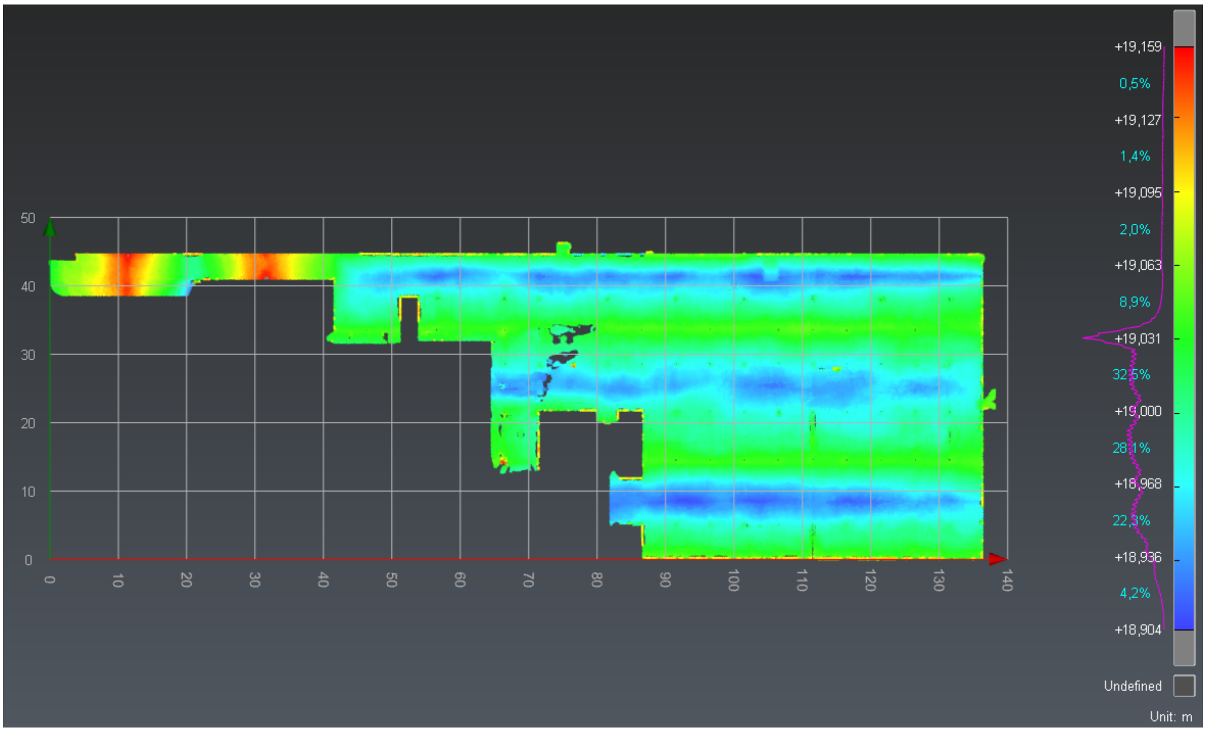
10.2.1 Exercise: Complete analysis of a concrete floor

In this command 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.3dr**. It contains one cloud of the walls and one cloud of the floor.

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

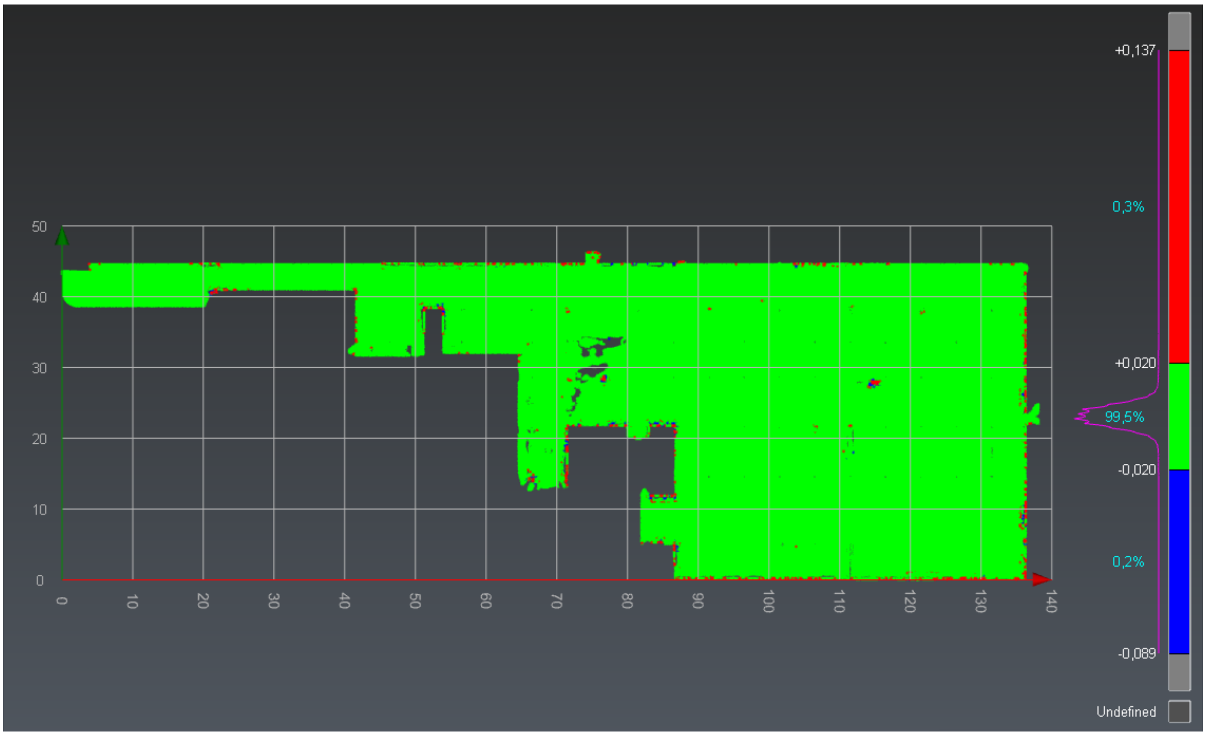


101 Check the levelness of a concrete floor

Check the flatness of a surface

This command allows checking if there are bumps or holes in a planar surface. This check can be done on a horizontal surface like a floor, but also on any other surface.
Show only the cloud **Floor** and select it again then go to 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 2 centimeters 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.

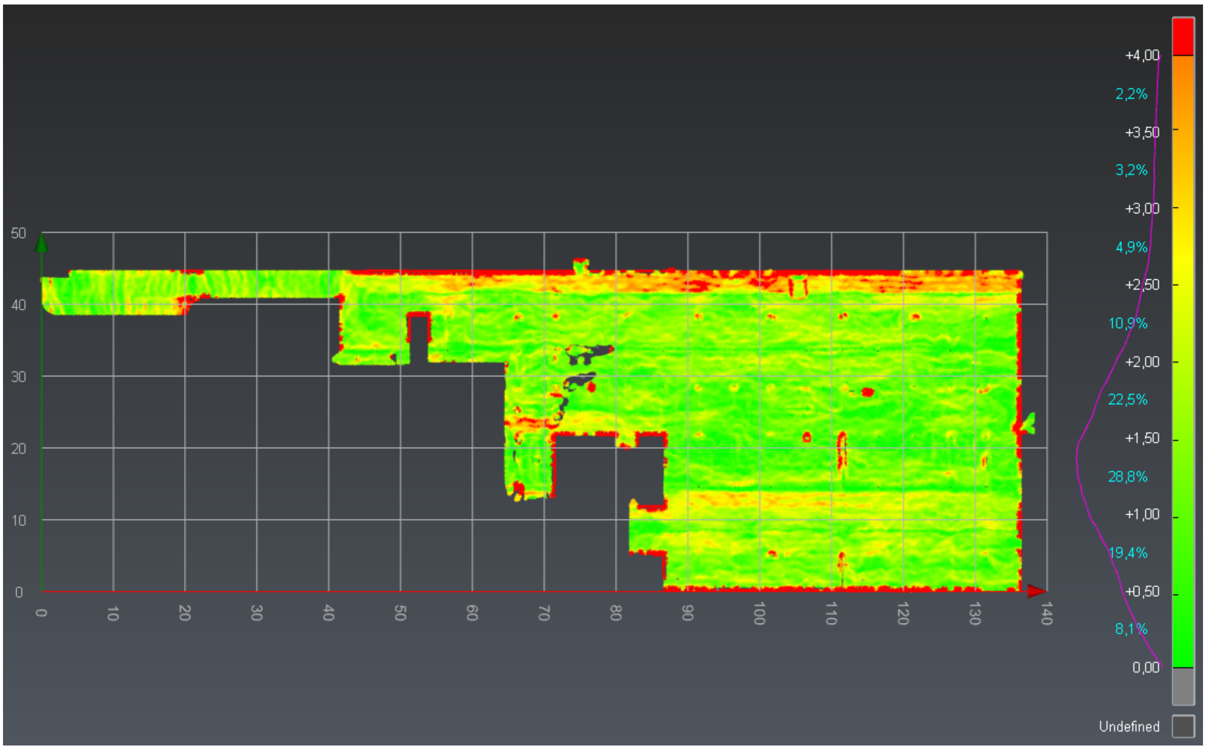


102 Check the flatness of a concrete floor

Check the local slope on a surface

The last command allows measuring the local slope on each point of a cloud or a mesh.
Show only the cloud **Floor** and select it again then go to Slope Analysis. 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 40 cm wide. Click **Preview** to preview the results.

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



103 Check the slopes on a concrete floor

Make a report

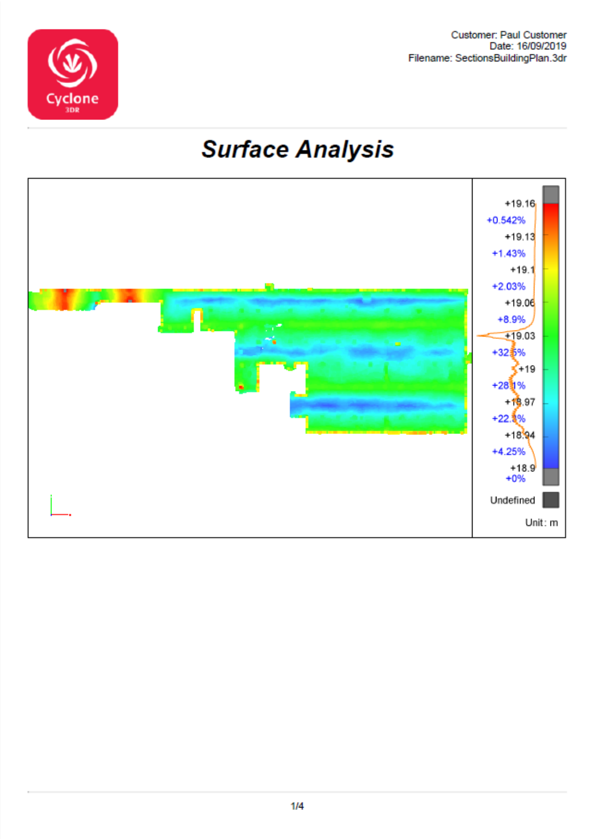
Now, you can create a pdf report with this analysis. Launch the Report Editor by clicking on the report menu in the ribbon. 4 chapters have been added to the report:

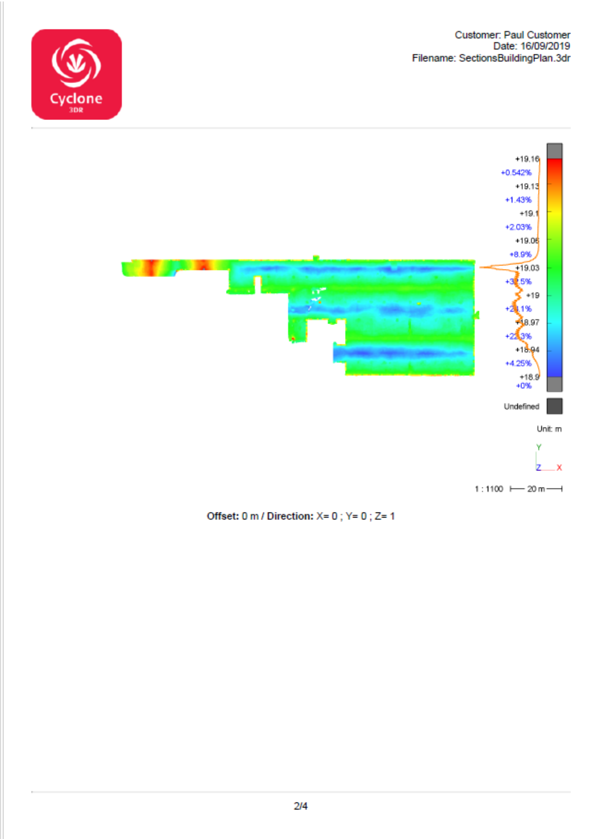
- cover
- surface Flatness
- surface Levelness
- slope Analysis

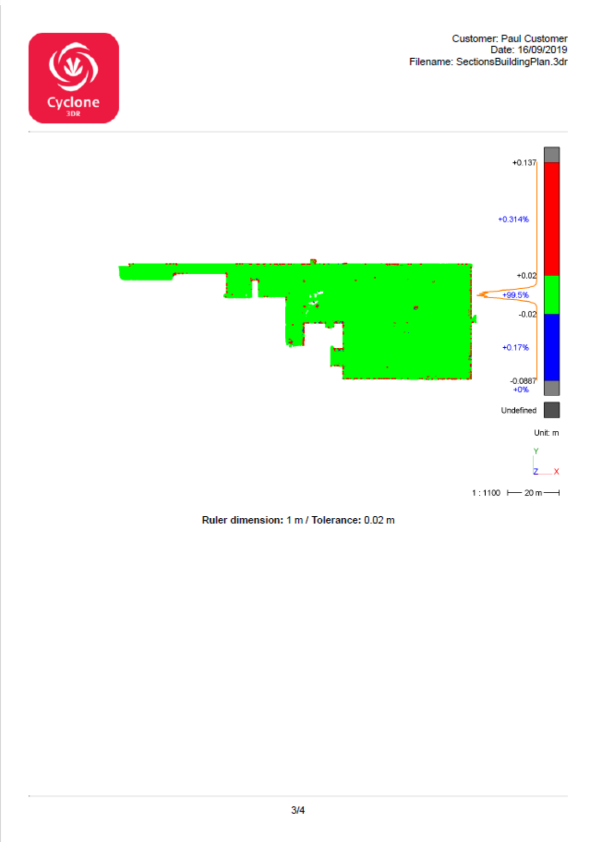
Click **New Chapter** to create an empty one called "Title page". Remove the chapter called Cover. Drag and drop the chapters so as to sort out your report as the following: Title page; Surface levelness; Surface Flatness; Slope Analysis.

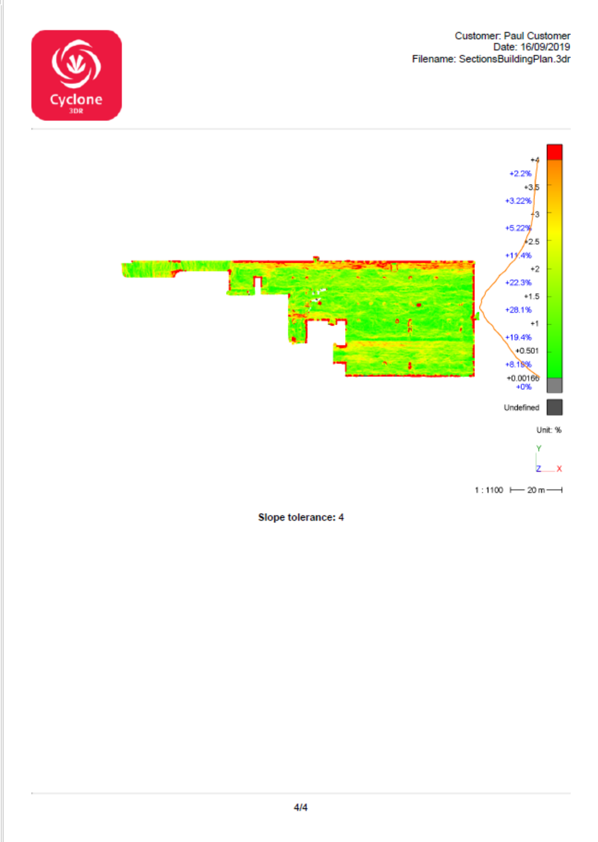
1. Choose a portrait **orientation** inside the **layout panel.** Apply header and footer for all the pages.
2. In the header, keep only two cells.
3. Add your company logo in the header left cell: import your logo double clicking the Company logo icon in **Environment data** (**data panel**). Then, insert it in the left cell.
4. In the header right cell, clear the content. Then, 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.
5. In the footer: keep one cell, add the **current page** and the **total page**. Align center these texts using the text toolbar.
6. In the body area: keep one cell, transform the cell into a **text area** and write the title report "Surface Analysis". Format it using an appropriate style.
7. Add a cell below and insert the levelness viewset in **3D Mode**.
8. In other chapters: add a cell containing the **chapter title**, format the texts.
9. Modify 2D scenes so as to set the **scale** to 1:1000. Choose to display the grid.
10. Optionally, export the chapters as new templates using **...** icon.

11. Finally, click on **To PDF** to generate the report.


Cyclone 3DR
Customer: Paul Customer
Date: 16/09/2019
Filename: SectionsBuildingPlan.3dr
Surface Analysis
+19.16
+0.542%
+19.13
+1.43%
+19.1
+2.03%
+19.06
+8.9%
+19.03
+32.5%
+19
+28.1%
+18.97
+22.3%
+18.94
+4.25%
+18.9
+0%
Undefined
Unit : m
1/4


Cyclone 3DR
Customer: Paul Customer
Date: 16/09/2019
Filename: SectionsBuildingPlan.3dr
Undefined
Unit: m
1 : 1100
20 m
Offset: 0 m / Direction: X= 0 ; Y= 0 ; Z= 1
2/4


Cyclone 3DR
Customer: Paul Customer
Date: 16/09/2019
Filename: SectionsBuildingPlan.3dr
+0.137
+0.314%
+0.02
+99.5%
-0.02
+0.17%
-0.0887
+0%
Undefined
Unit: m
1 : 1100
20 m
Ruler dimension: 1 m / Tolerance: 0.02 m
3/4


Cyclone 3DR
Customer: Paul Customer
Date: 16/09/2019
Filename: SectionsBuildingPlan.3dr
Undefined
Unit: %
1 : 1100
20 m
Slope tolerance: 4
4/4

104 Report

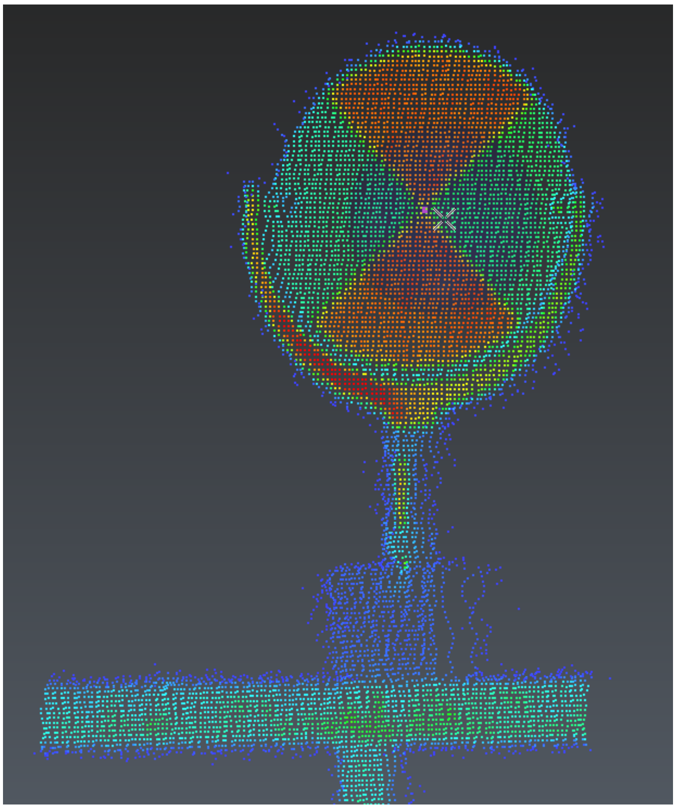
10.3 Surveying extraction

- 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.3dr**".

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.
Launch the command Point and select the option **Surveying Target.** Select **With checkered pattern** in the type list and a **diameter** of 0.1m. Click a point on the target close enough to the center (**Target 5 - Checkered pattern**). The center will be automatically computed and a new point created.



105 Automatic extraction of target center

The target extraction works on black and white targets or blue and white targets only when the "Inspection" representation mode is available, 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 and import the file "**ExtractGround.nsd**".

You can directly drag and drop it in the 3D scene, or import it with the function Import. Be sure to set file unit to **meter**. 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 DTM.

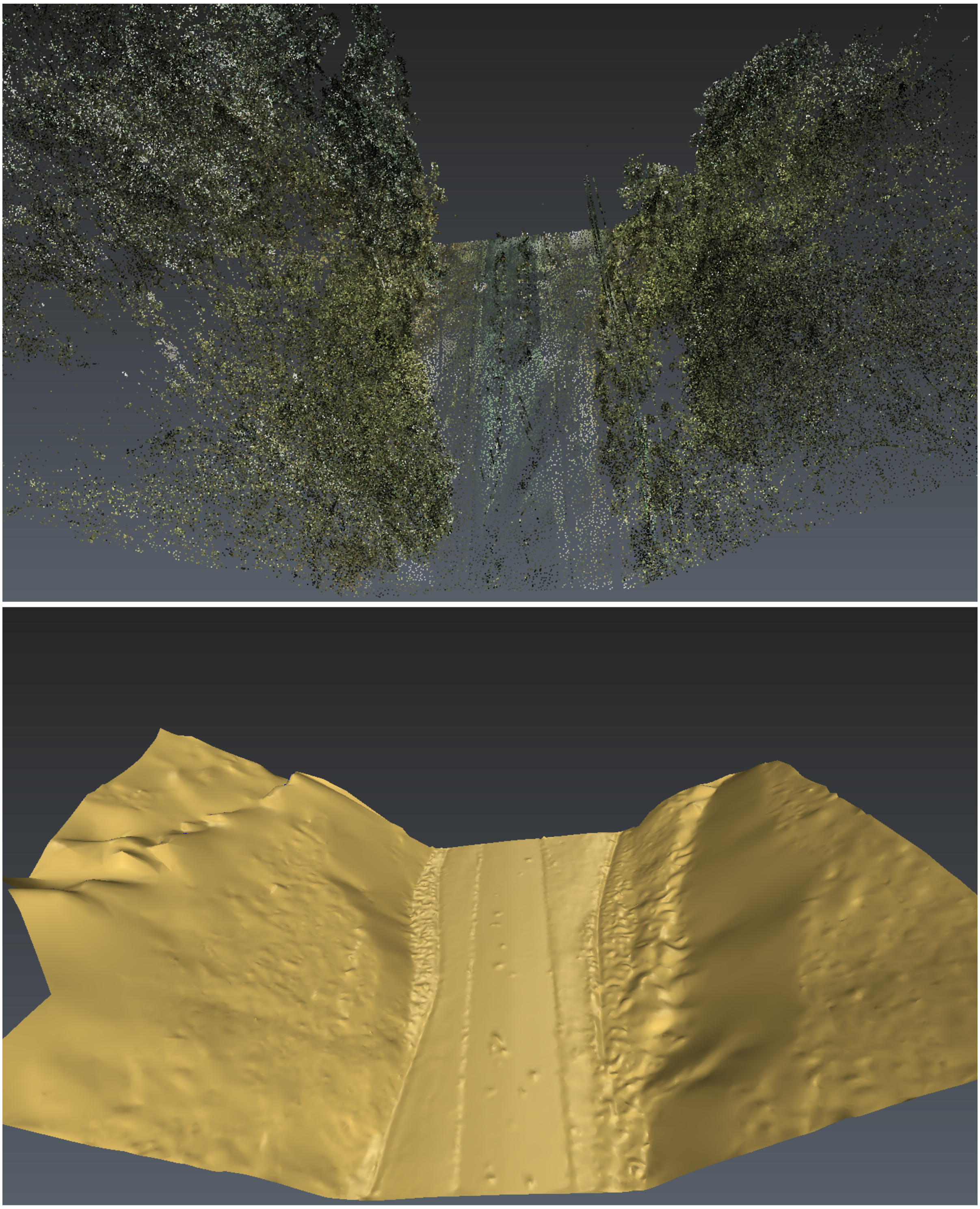
Set the maximum slope that can be seen in the terrain to **45°**. Choose **Z axis** as the direction for the computation. The **Extraction grid size** will give you the level of details in the final mesh. This value is computed by default, but you can set it to **0.15 m**.

Choose **Check noisy points** as extraction strategy.

Choose the **Refined** meshing strategy in order to improve the result.

Click **Preview** to get the resulting mesh. You can show or hide the initial cloud in order to better see the result.

It is also possible to extract the points that are on the ground, the points that are not on the ground or the noisy points. Click **OK** to create the mesh.



106 Automatic creation of a DTM

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.3dr**”. This file contains two point clouds of a building, also used for the alignment exercises.

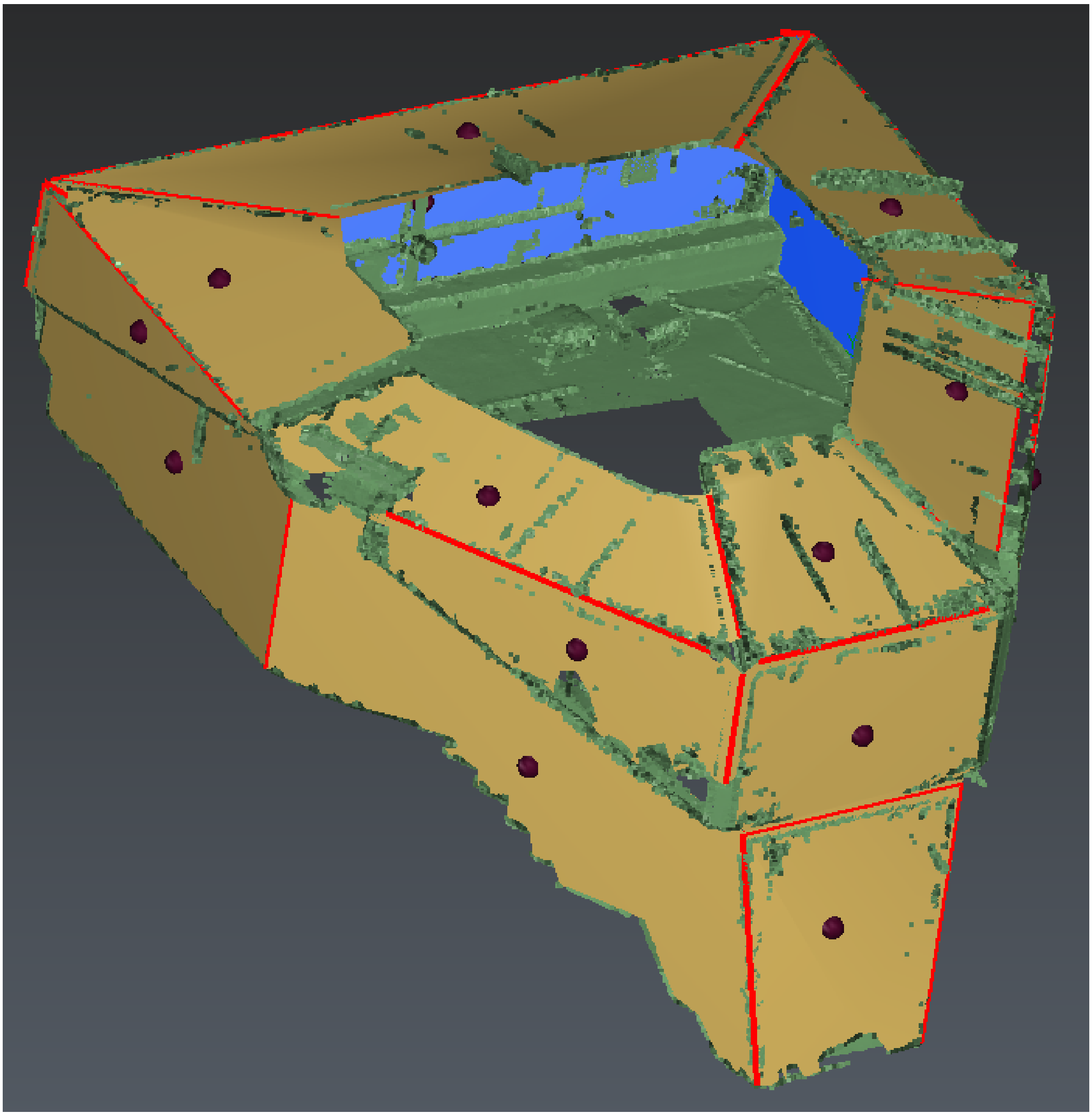
Show only the cloud **Fusion Cloud**. You can let the representation mode as **Inspection** or turn it to **Smooth** to not see the colors over the point cloud.
Select the cloud and launch the command Building Extractor.

The software 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 slider 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.

While the process and after having clicked on a surface, if the automatic contour is not satisfaying, you can click on the space key to launch the manual contour, re-draw the selected surface and then click Enter to proceed.

Continue like this to model the entire building.



107 Create a model based on planar faces

10.5 Volume and cubature

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 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

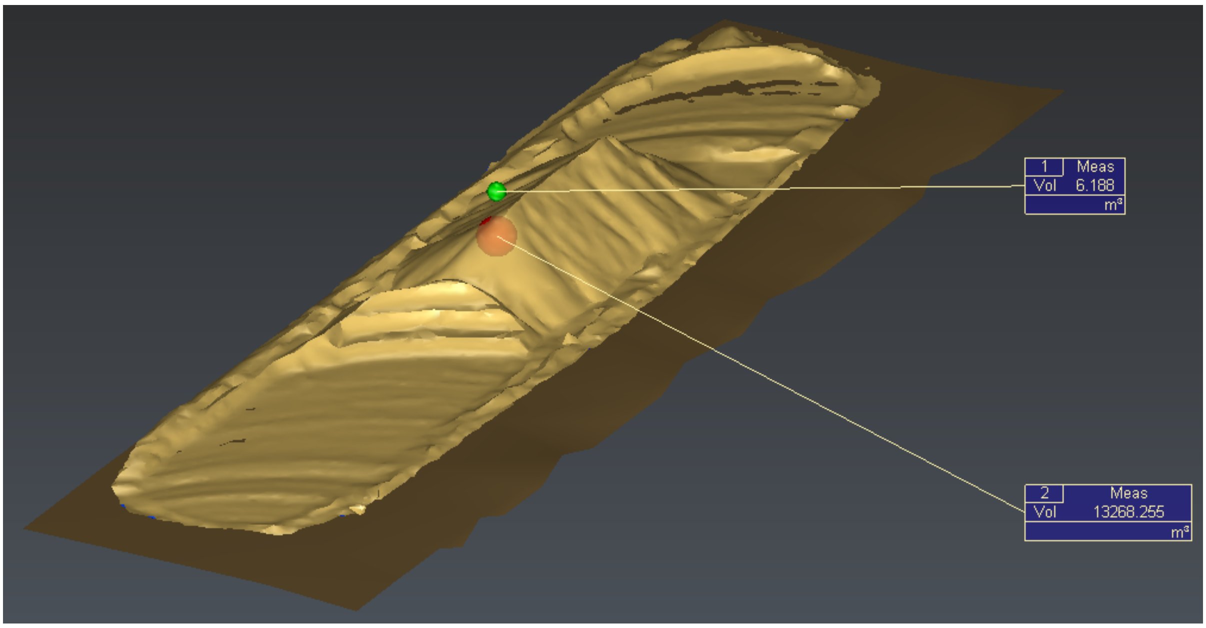
10.5.1 Exercise: Compute cubature between two open meshes

With the command Cut and Fill, 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.3dr**". It contains the mesh of a stockpile and the mesh of a reference ground.

Select both and go to Cut and Fill.
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, the command tries to find an appropriate direction to see both entire surfaces.

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



108 Compute cubature between two open meshes

Volume output

```
Volume above Reference Ground and below Stockpile:
13268.255 m³

Volume above Reference Ground and below Stockpile:
6.188 m³

Difference of the two volumes: 13262.067 m³

If the reference surface is Stockpile,
Excavation volume of 13268.255 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 13268.255 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 do exercise 3D Meshing & Cubature 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.3dr**".

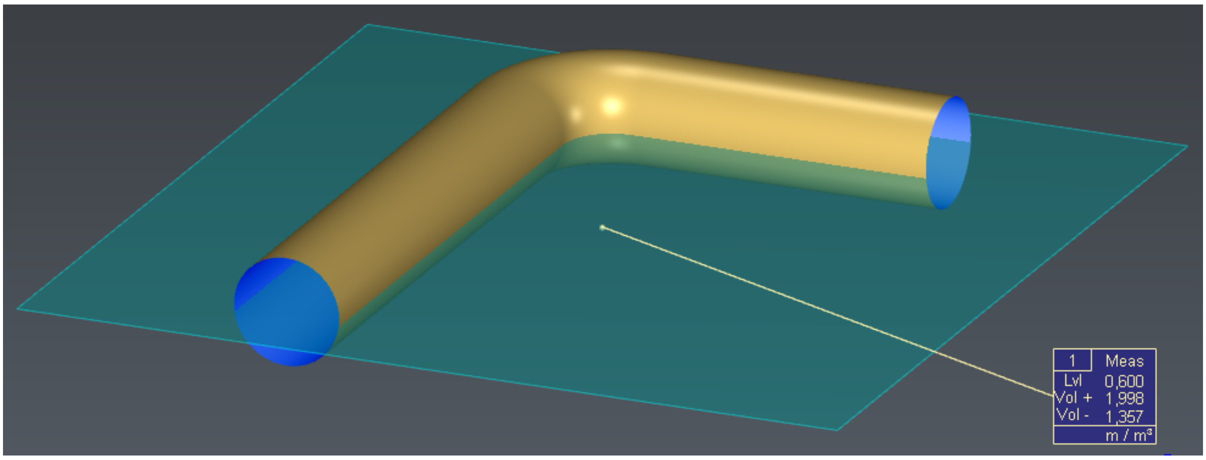
Select the open bent pipe and go to Volume from Elevation. This command can be used to compute volumes over and under a level of liquid inside a closed or an open mesh.

The dialog box opens and you can see that there are two ways to create planes: the Regular way and the List of distances method. The default method is the Regular creation of planes with a 0.01 m step. The selected pipe is way higher than 1 m high, so another option should be used for the exercise.

Select **List of values** for the **Extraction method** and enter **0.60** for example. You can see that a plane representing the water level is displayed in the scene.

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.



109 Compute cubature with a level of water

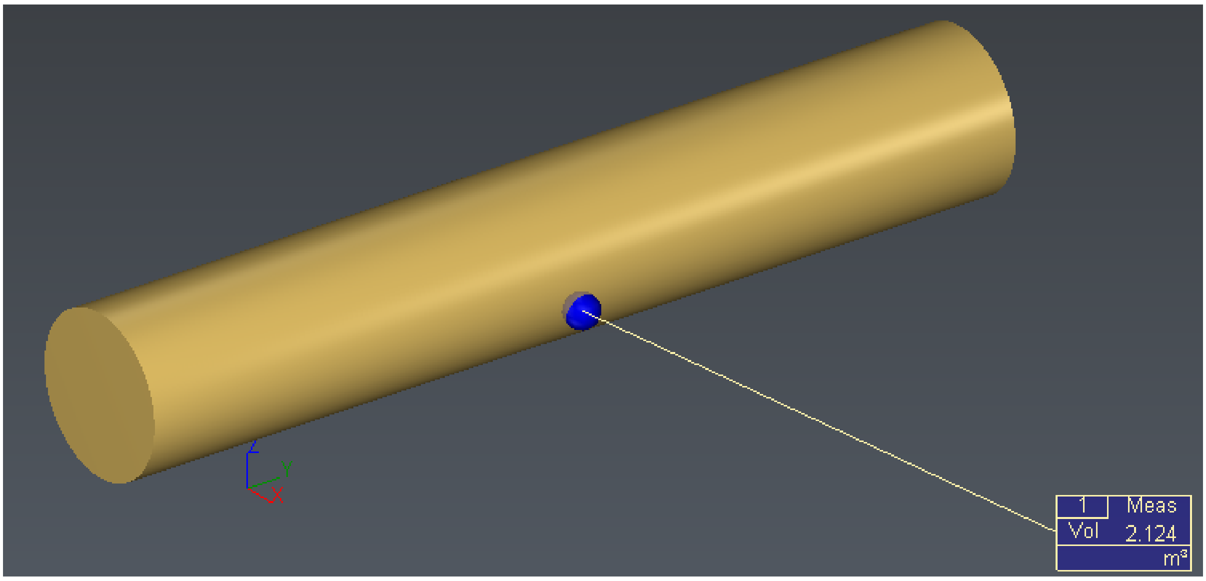
You can try the command with the closed Bent Pipe and with the open Bent Pipe. In this case, if the volumes are computed for the same level of liquid, it makes no difference whether the mesh is closed or not because the pipe is closed vertically.

With the option **Range**, 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

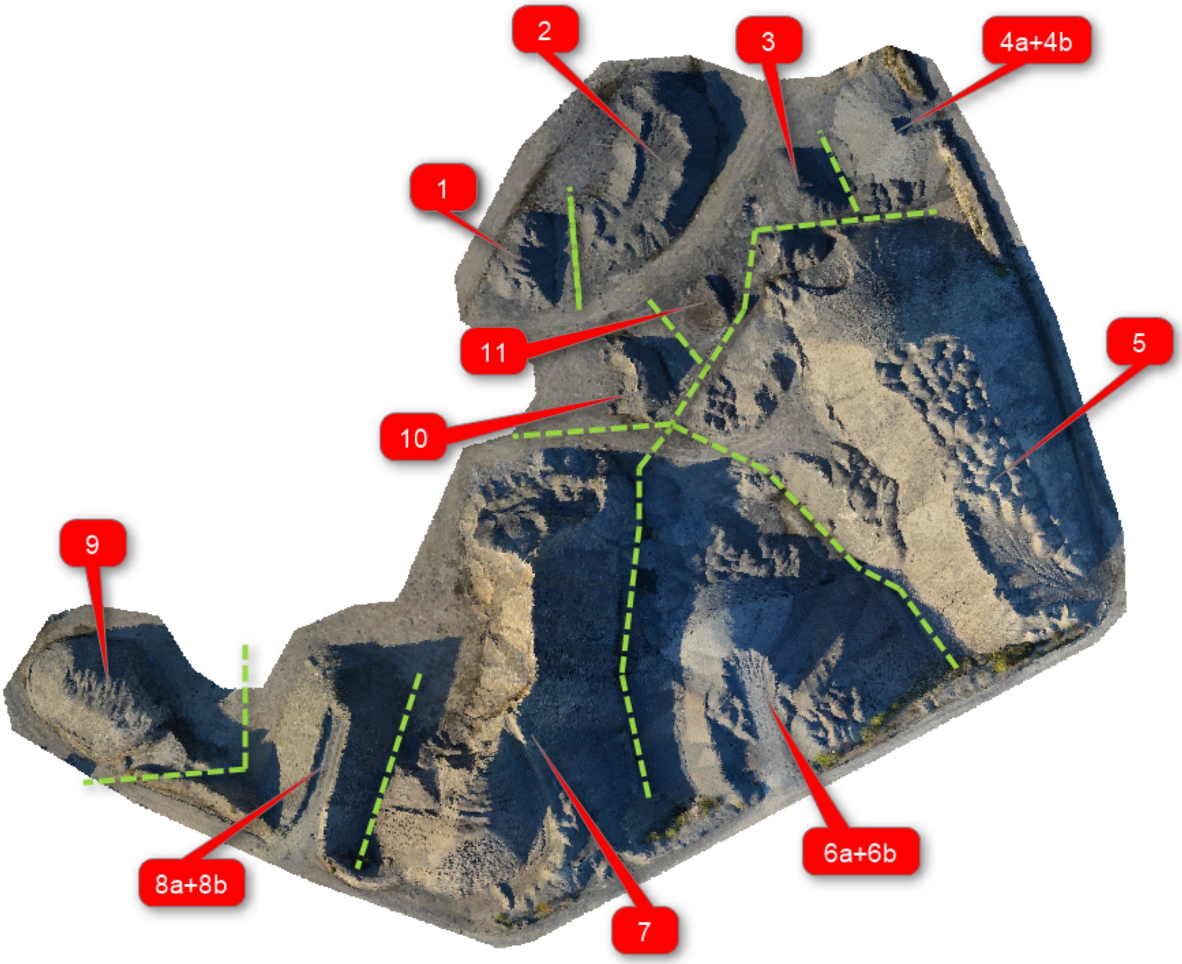
Open the file “**VolumeClosed.3dr**”. It contains the meshes of two sections of a pipe.

Select one mesh and go to Volume 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.



110 Volume of a closed mesh

10.5.4 Exercise: Create a stockpile project



Open the file

Open the file “**Stockpile.3dr**”. It contains a cloud with several gravel stockpiles. This file is going to be used through this whole exercise. Select **Stockpile cloud** and launch the command Stockpile.

Tip & Trick

To help you draw or select the contours, polylines have been added to this sample.

Create stockpiles

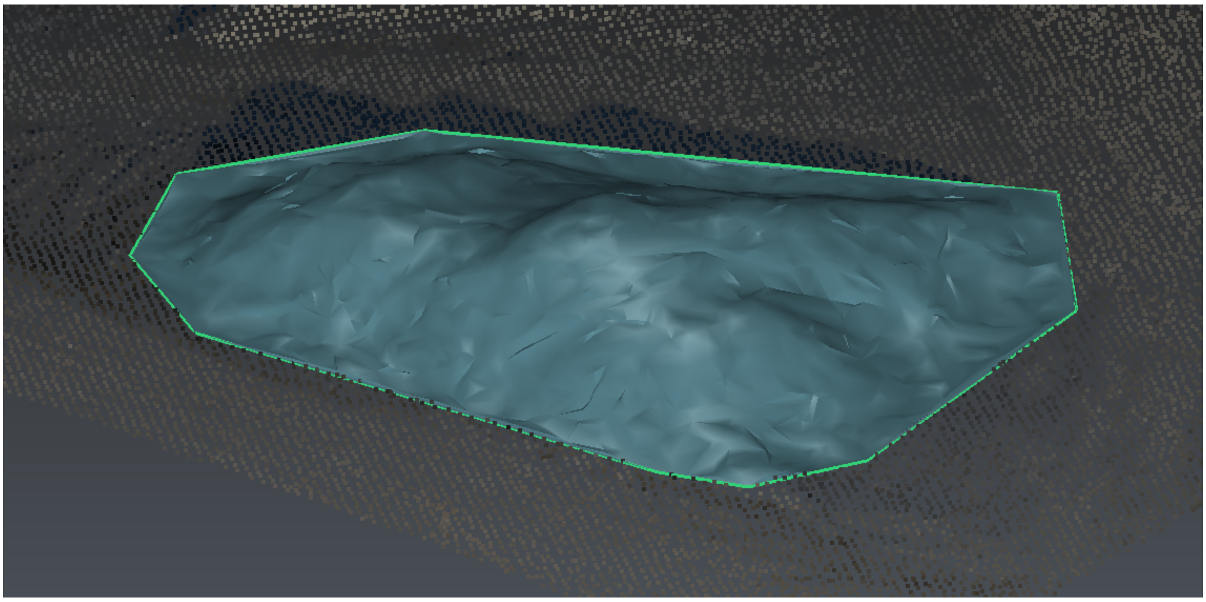
Stockpiles n°1, 2 and 3

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

- material name: 10001 (for instance a material ID),
- material nature: gravel
- 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 manually or select its contour (refer to the stockpile map, to the polyline Contour 1 and to fig.1). Close and validate the contour with **ENTER**.



111 Figure 1: stockpile n°1

Select the **Best plane from contour** method to mesh the initial ground below the stockpile. 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.

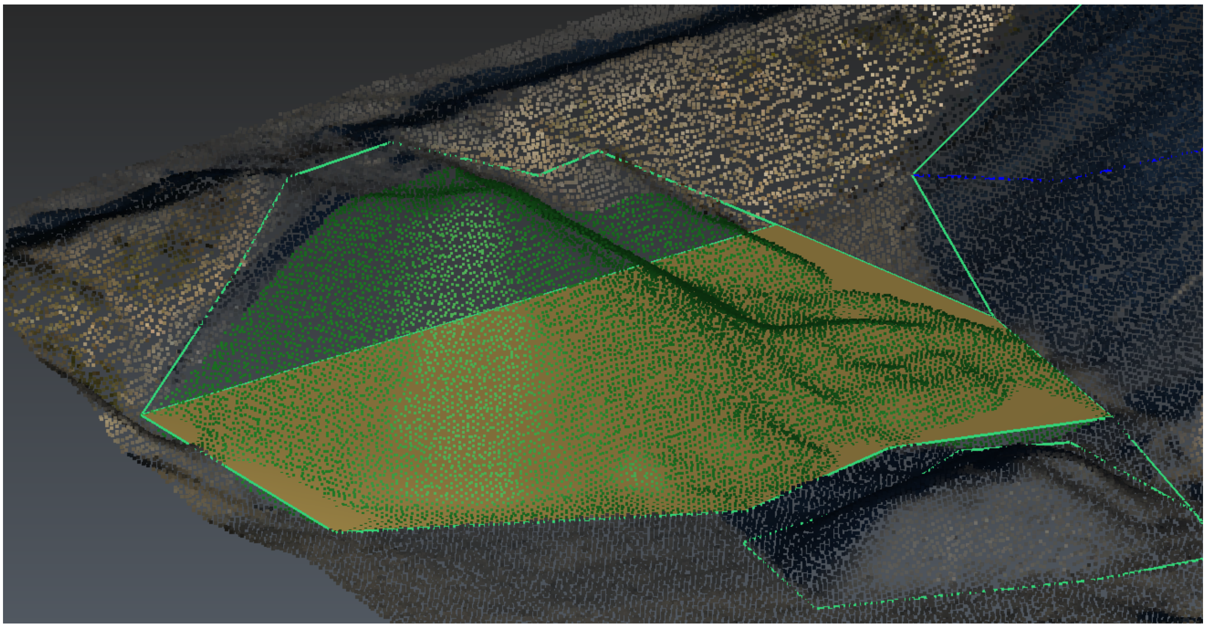
Stockpile name	Material name	Material nature	Grain-size	Method	Spike elimination	Cut Volume
1	10001	gravel	4/6	Best plane from contour	None	$422m^3$
2	10001	gravel	4/6	Best plane from contour	None	$2365m^3$
3	10002	gravel	4/6	From contour points	None	$294m^3$

Note

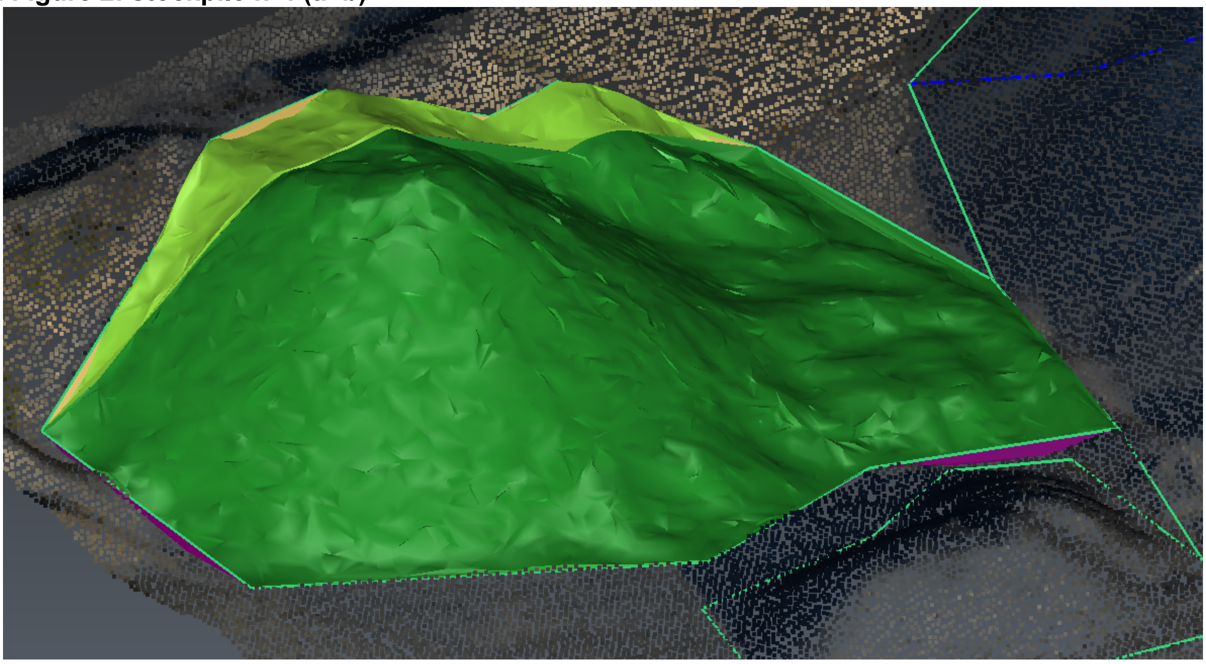
Some stockpiles are along embankments. Consequently, you have to find out the initial ground. Sometimes, you can reconstruct the embankment shape by splitting the stockpile (n°4, 6, 8). Sometimes you can't (n°5 and 7) but you have to choose the less inaccurate method.

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.



112 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	1874m^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

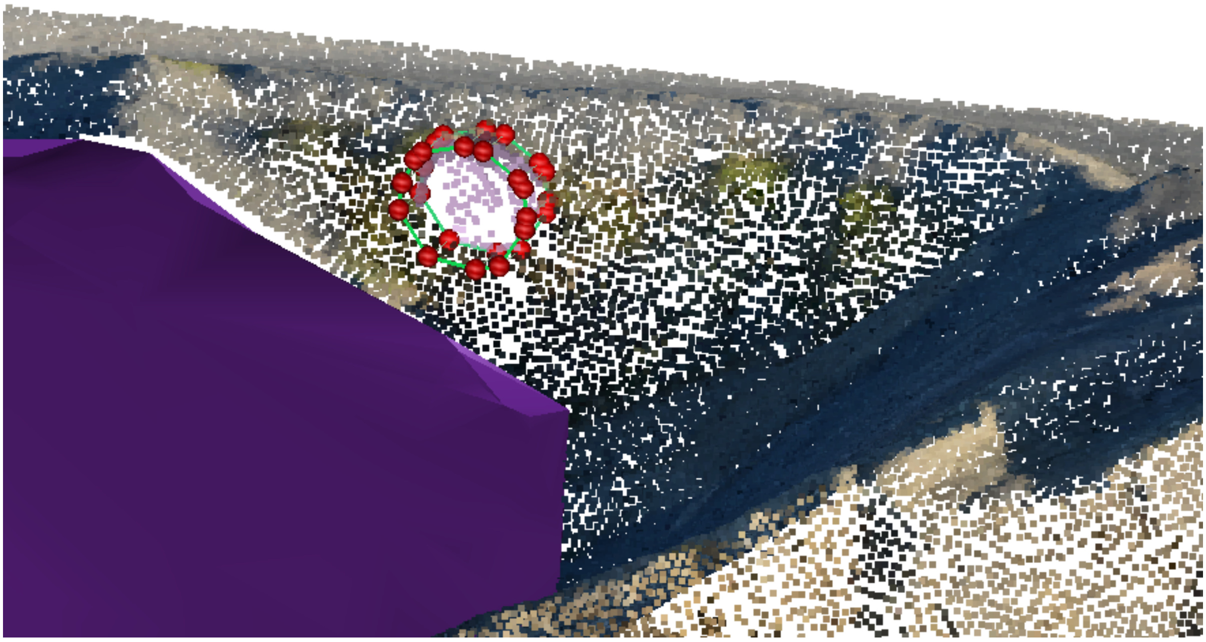
Repeat the same workflow for each stockpile. You can also go directly to the next section.

Stockpile name	Material name	Material nature	Grain-size	Method	Spike elimination	Cut Volume
5	20001	stone	100/200	Horizontal plane from lowest point	Medium	91363m^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	50292m^3
8a	10004	gravel	0/31.5	Horizontal plane from lowest point	Medium	5773m^3
8b	10004	gravel	0/31.5	From contour points	Medium	318m^3
9	10005	gravel	4/10	Horizontal plane from lowest point	None	5082m^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 Clean / Separate to remove the points.

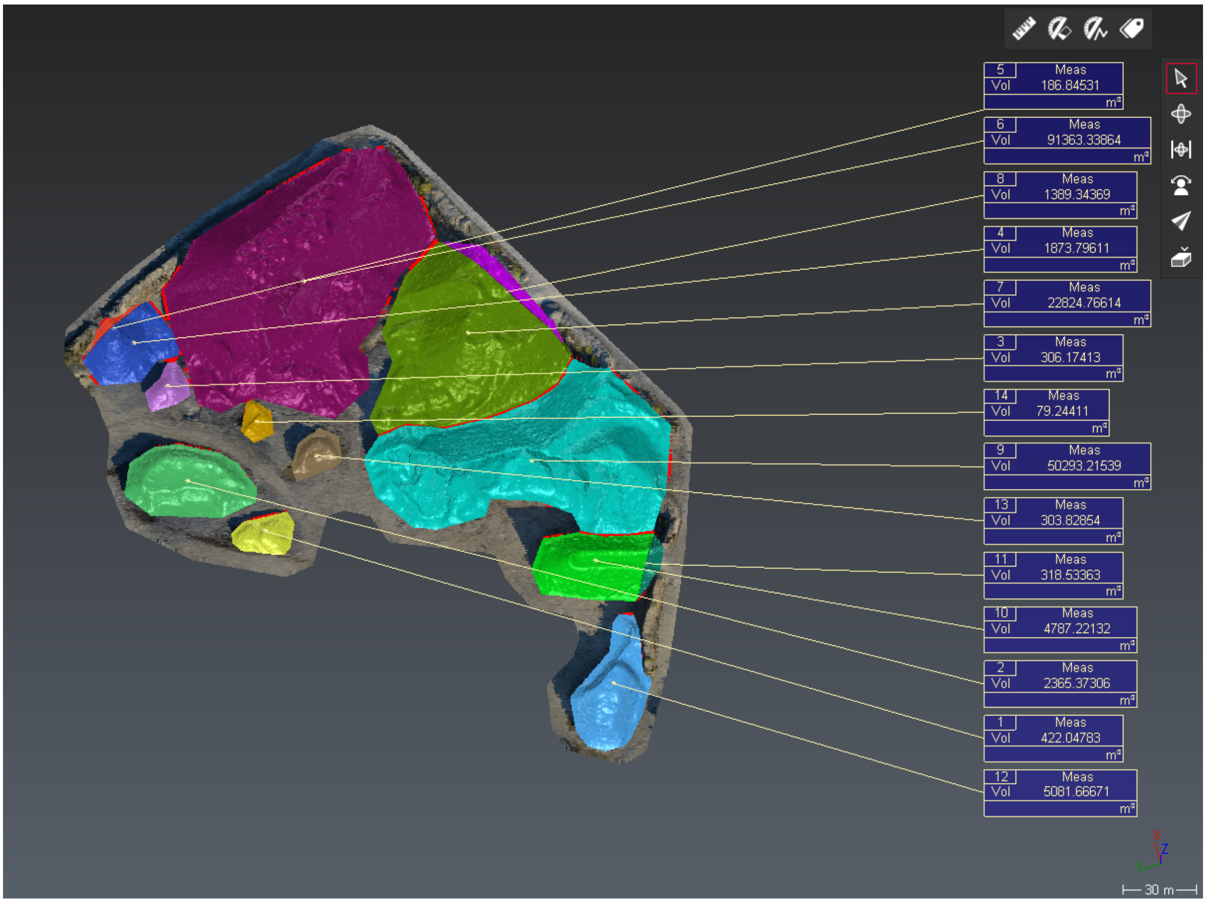


113 Figure 3: remove noisy points

> **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.

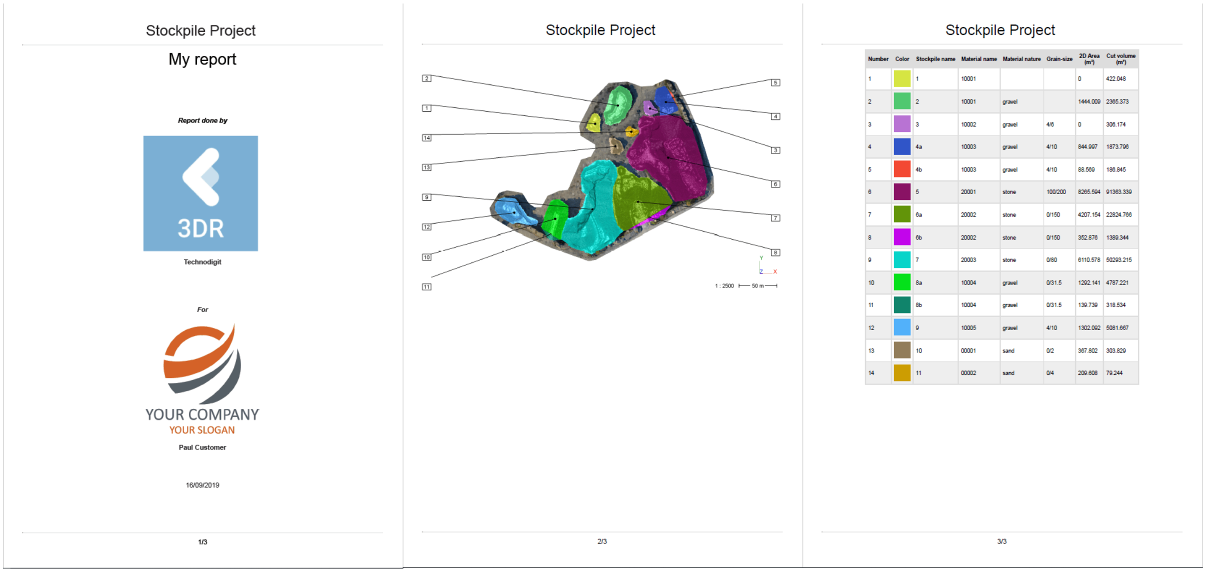


114 Figure 4: colored map with labels

Create a report

Launch 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 another size for labels (**Minimum**),
- 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.



Number	Color	Stockpile name	Material name	Material nature	Grain-size	2D Area (m²)	Cut volume (m³)
1		1	10001			0	422.048
2		2	10001	gravel		1444.009	2365.373
3		3	10002	gravel	4/6	0	306.174
4		4a	10003	gravel	4/10	844.997	1873.796
5		4b	10003	gravel	4/10	88.569	186.845
6		5	20001	stone	100/200	8265.594	91363.339
7		6a	20002	stone	0/150	4207.154	22824.766
8		6b	20002	stone	0/150	352.876	1389.344
9		7	20003	stone	0/80	6110.578	50293.215
10		8a	10004	gravel	0/31.5	1292.141	4787.221
11		8b	10004	gravel	0/31.5	139.739	318.534
12		9	10005	gravel	4/10	1302.092	5081.667
13		10	00001	sand	0/2	367.802	303.829
14		11	00002	sand	0/4	209.608	79.244

115 Fig. 5: report

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).

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 standard 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.

The Tank license is mandatory to use this module.

11.2 Define the project

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

11.2.1 Start the project

Open the file **TankInspection.3dr**.

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

Select the mesh **Tank mesh** and launch 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 click point tool.

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.

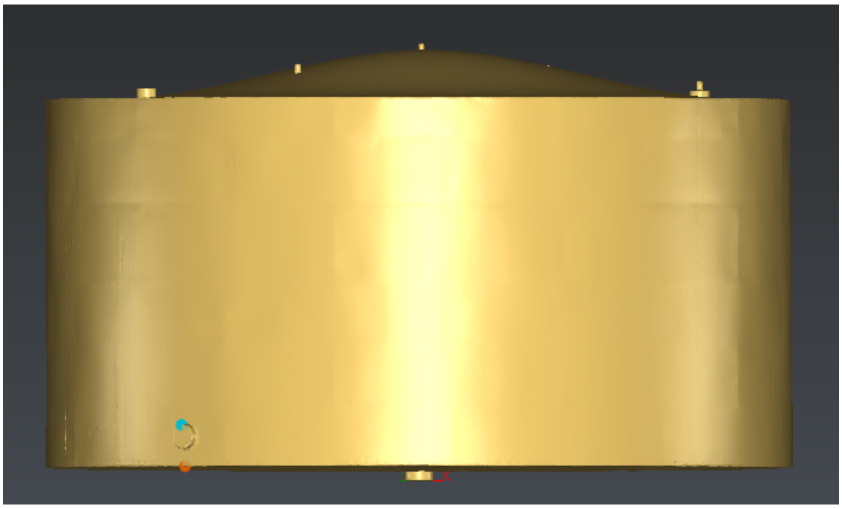
Finally, enter the theoretical height of the tank: **11 m**.

Before validating the project, you can also generate the best cylinder shape of the tank, by clicking on **Compute best cylinder shape**. Do not launch this command. The best cylinder will be extracted during the next step of the workflow. Compute the best cylinder

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

- **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 Cyclone 3DR.



116 Creation of a Tank project

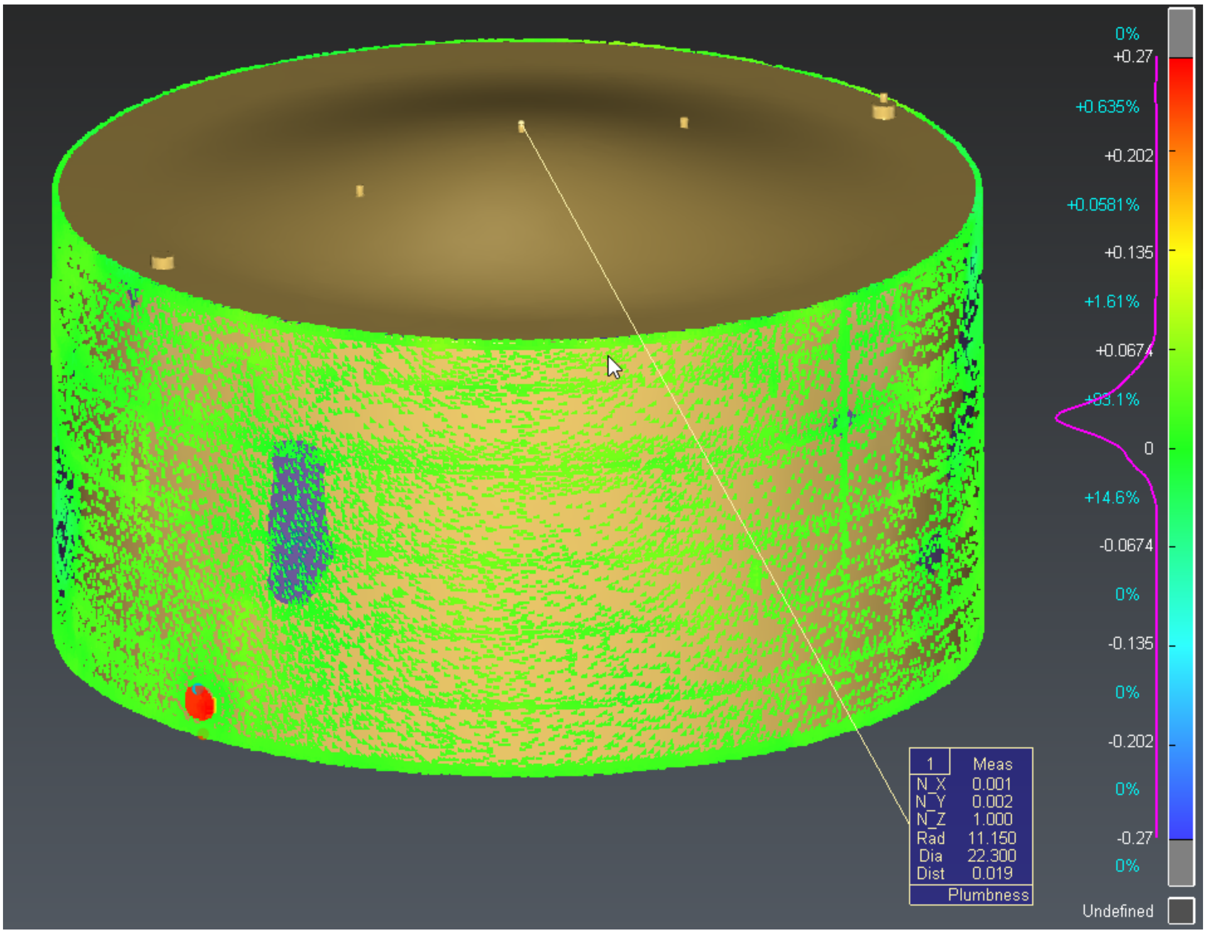
11.2.2 Compute the best cylinder

In the previous step Create/Edit Project, you had the choice to compute the best cylinder from the tank mesh. If you did not, you still have the possibility to launch the command after you created the project.

Click on 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**.



117 Compute a best cylinder on the tank

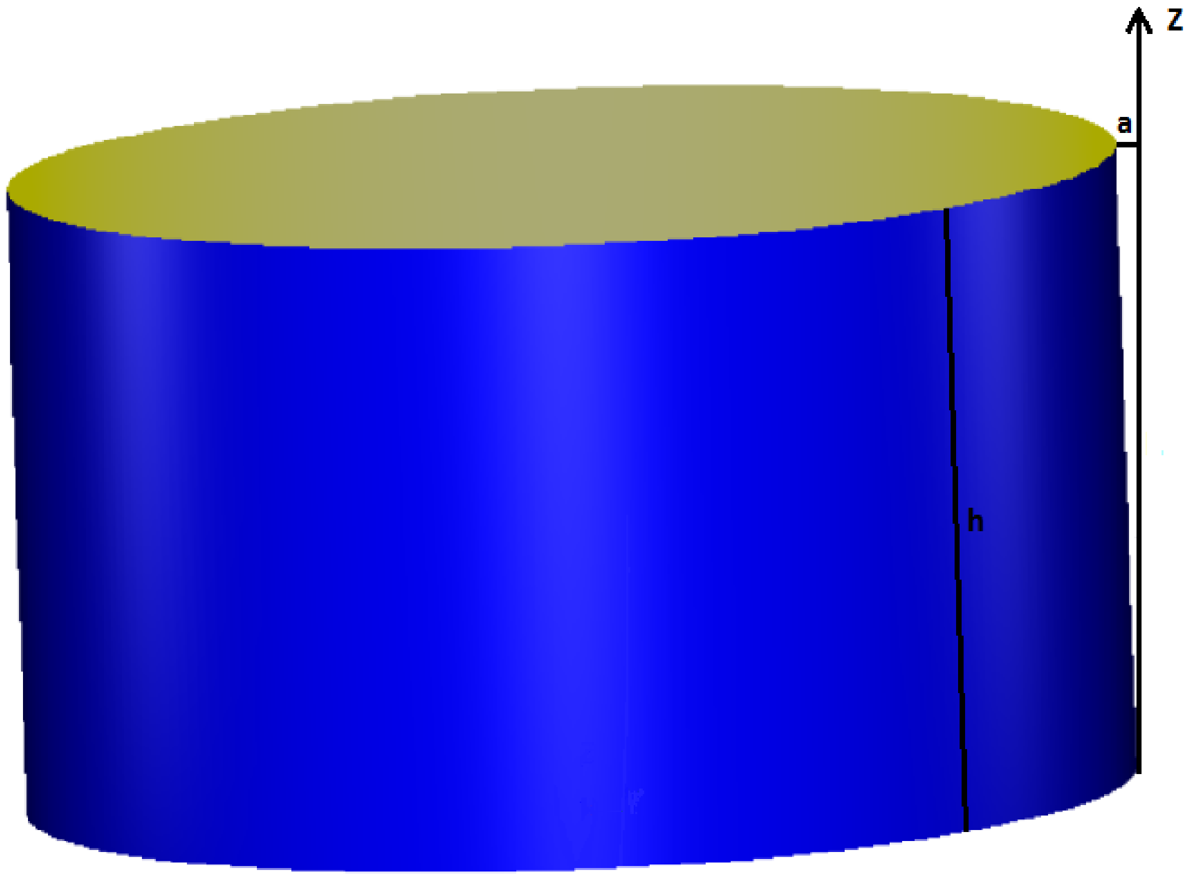
It is possible to eliminate the worst points, for example if the mesh is not perfectly clean. Here we can keep the computation with all points.

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

A **Results** section in the dialog 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.



118 Definition of the plumbness (a) of a tank

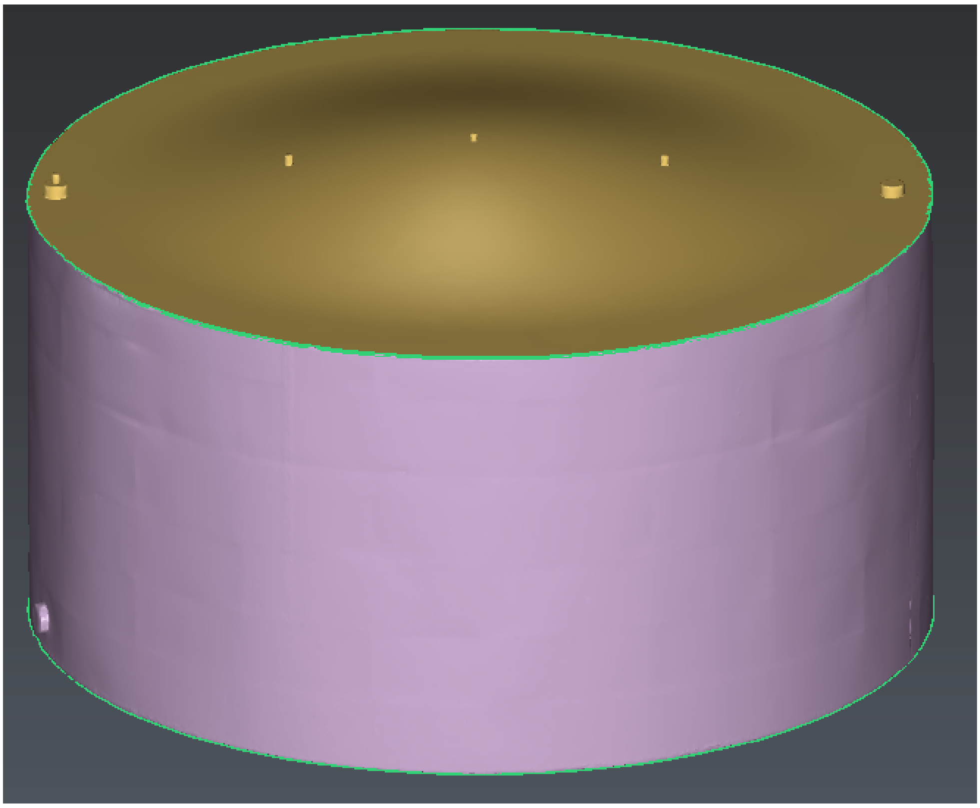
> ⚠ The use of API 653 tolerance is not mandatory. If you want to enter another tolerance, uncheck the corresponding checkbox and enter your own value.

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, the label giving the plumbness value and a Report data.

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 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).



119 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

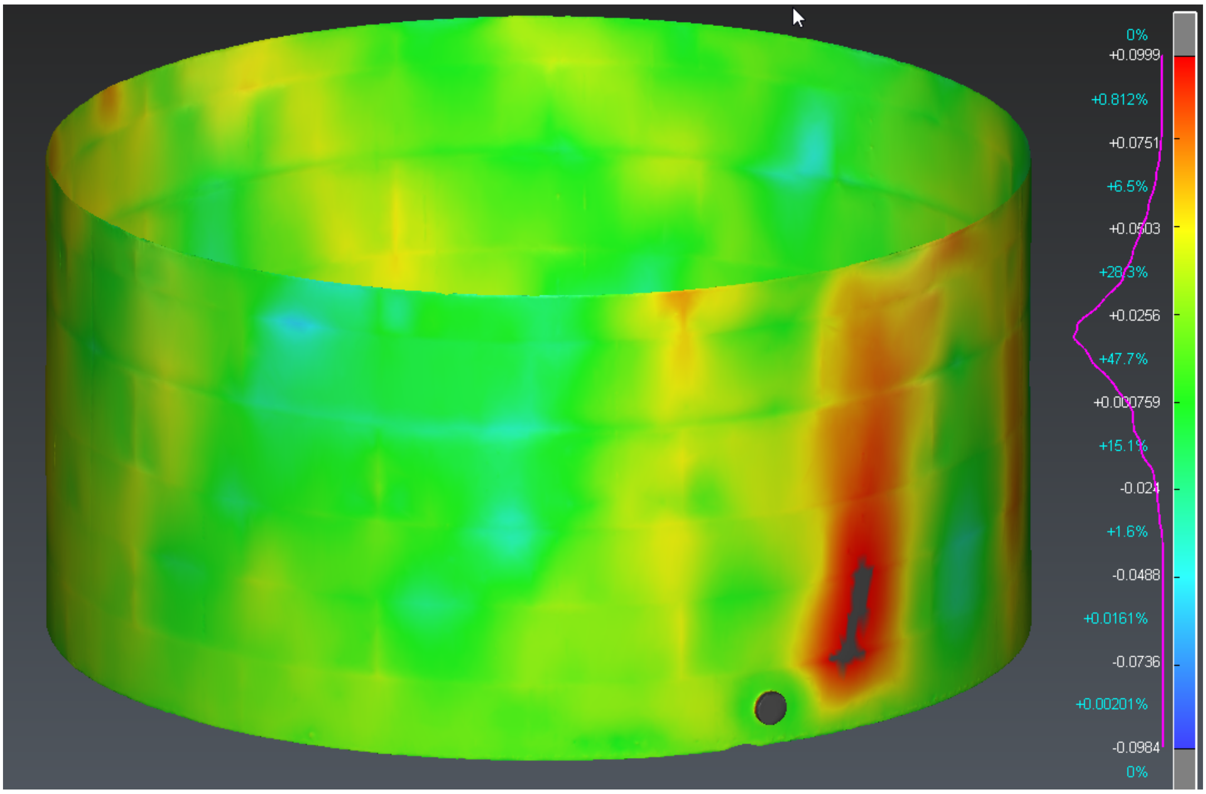
Show only the mesh called “Tank mesh Shell”. Select it and launch Create Color Map.

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.

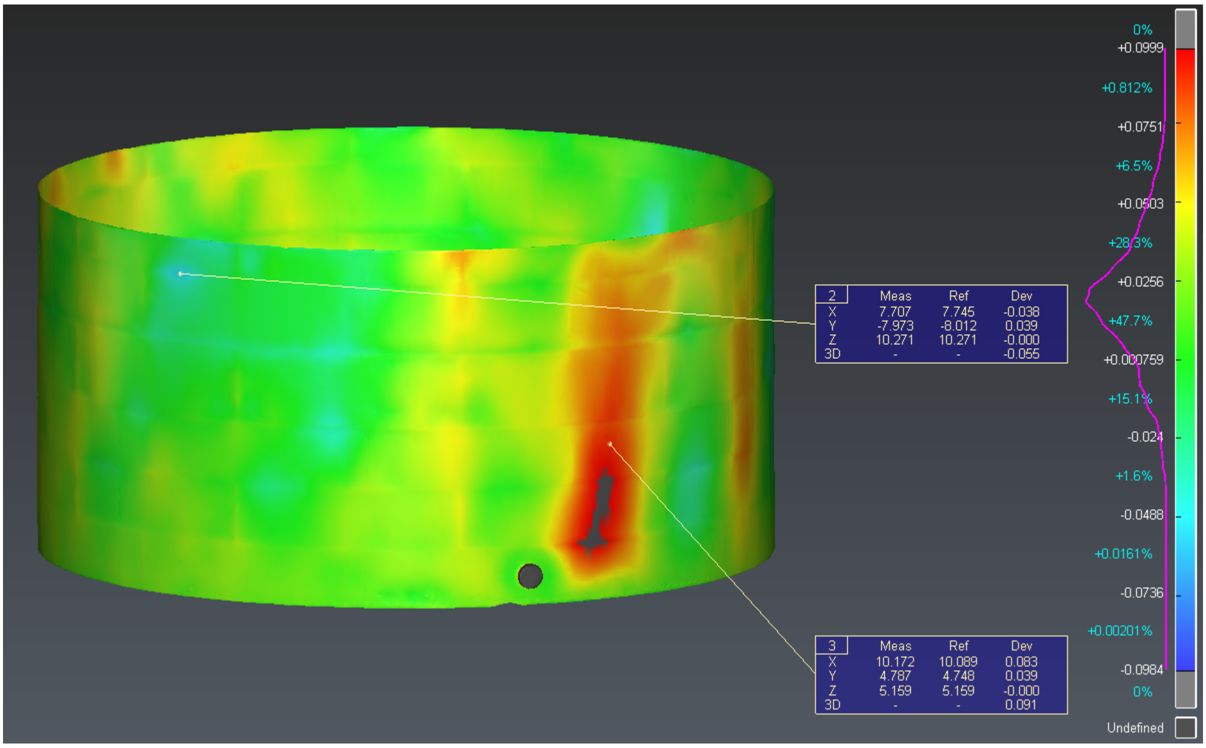
Change this distance to **0.10 m** in order to ignore the manhole and click **Preview** to replay the computation. We now see better where could be some problematic areas on the shell.

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



120 Compute the 3D deviations on the tank shell

Click **OK** to validate the result. Then add labels on specific points on the shell using Measure Deviation. 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.



121 Create labels on specific points
The colored mesh and the labels are created in a new folder “3D Inspection”.

11.3.2 Unroll the color map

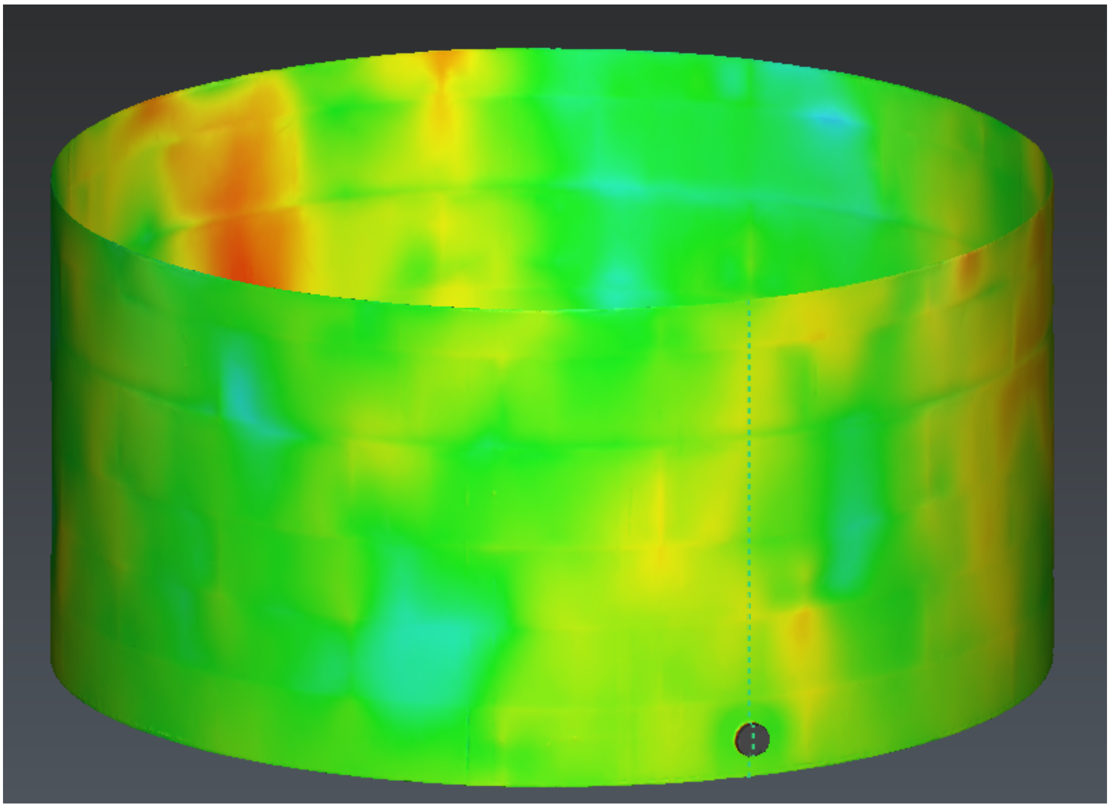
This colored mesh can now be unrolled to show a 2D inspection map of the shell.

Show only the mesh **Compare Tank mesh 1 Best Cylinder / Tank mesh 1 Shell**, select it, and launch 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**.

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.

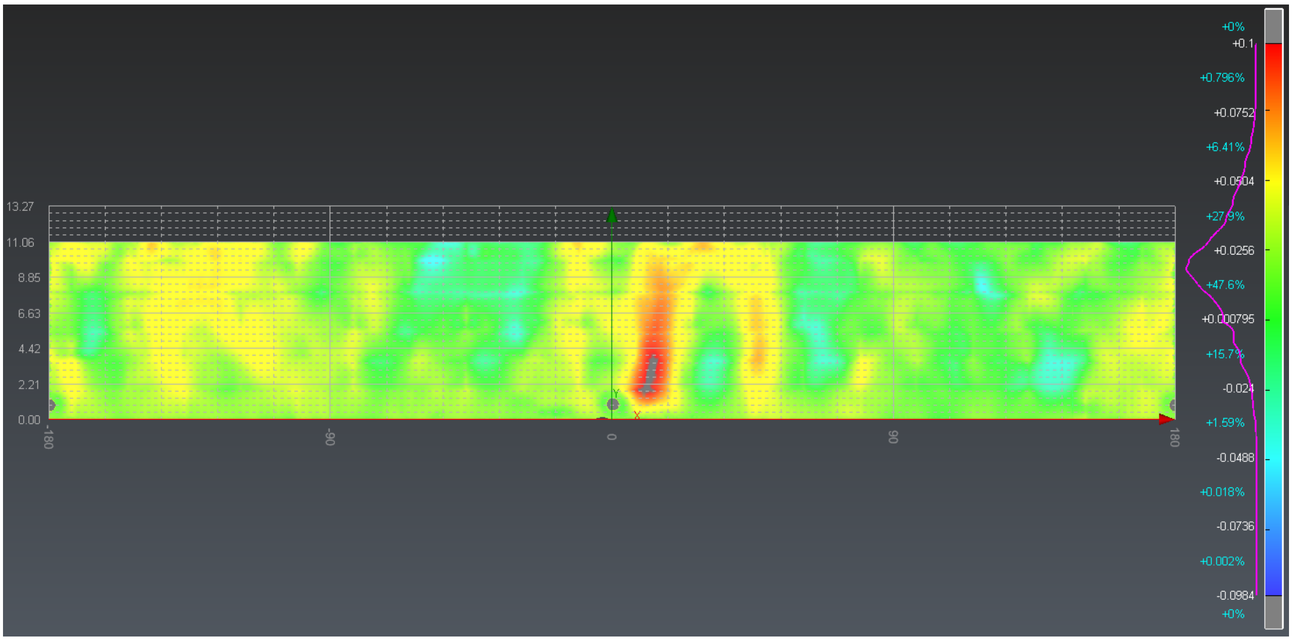
The shell will be cut in its height, where the Orientation Marker has been defined during the project creation.



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

Click **Preview** to compute the result. **Turn the result.**

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



123 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 1**. Select it and launch the function 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. It is also possible to define height reference at the lowest point instead of the elevation marker.

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.

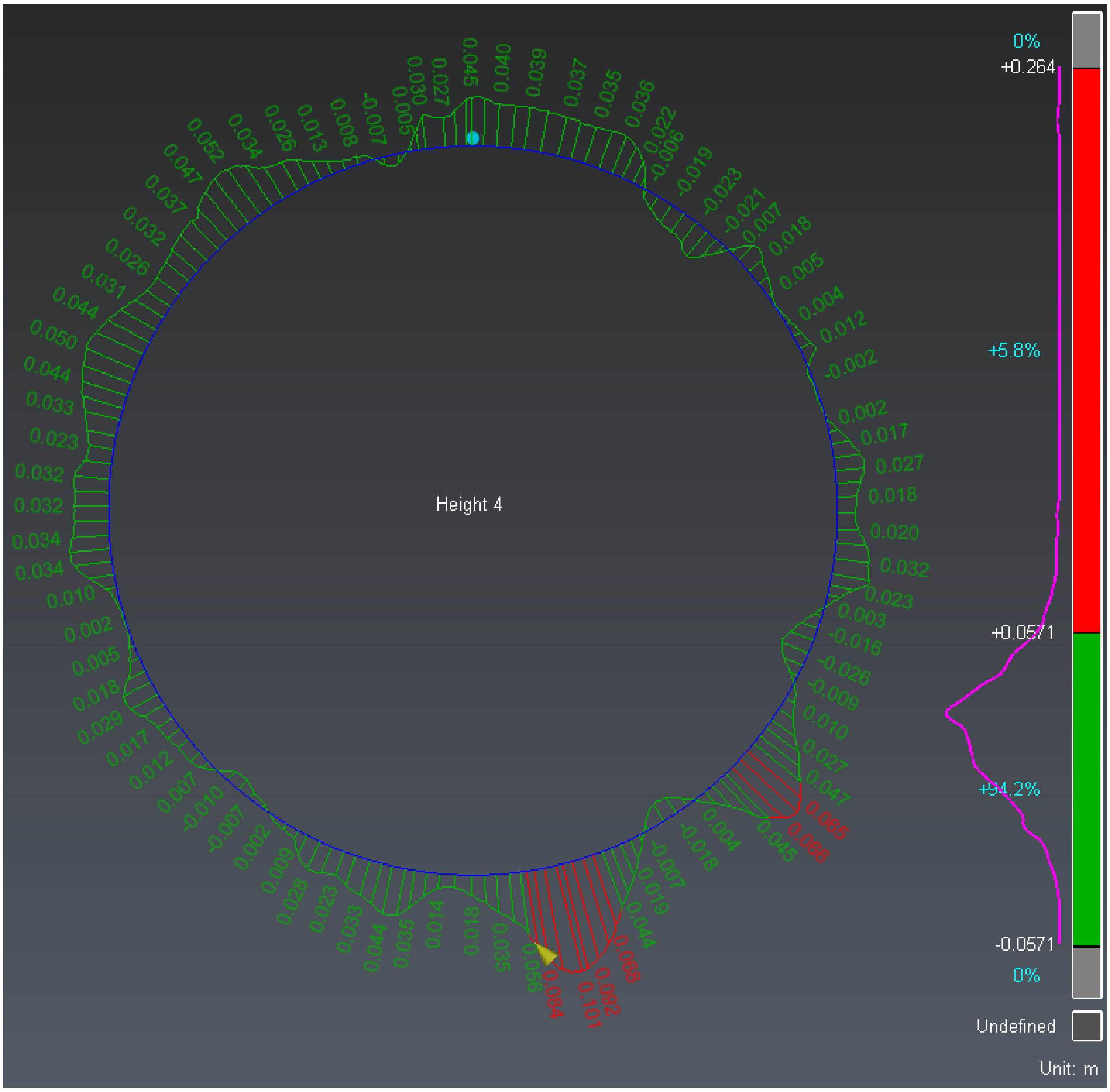
Choose the options **With a regular step** and **All over** and give a step of **1 m**. Click **Preview** to display the resulting sections.

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.

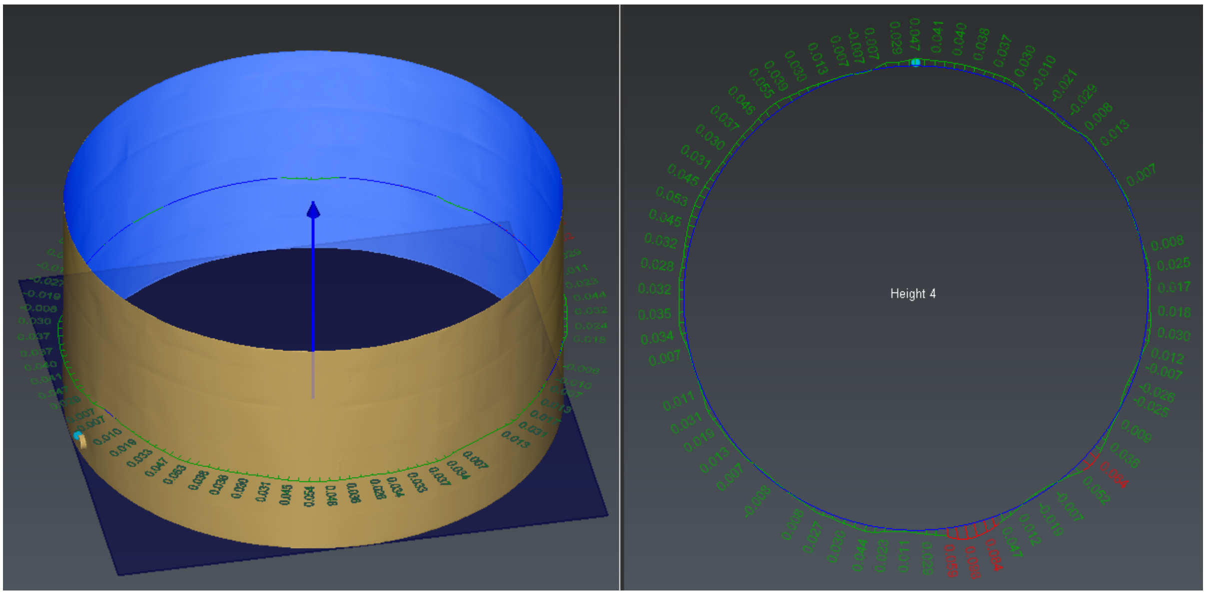
Choose to show only the section at height 4 (4 m) and click **Edit Color**. Here, it is possible to adapt the color scale if needed, and also to magnify the deviations in order to see even the smallest ones. Set the cursor on 32 for example.

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



124 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.



125 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 1”. Select it and launch the function 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 around** and give a step of **10°**. Click **Preview** to display the resulting sections.

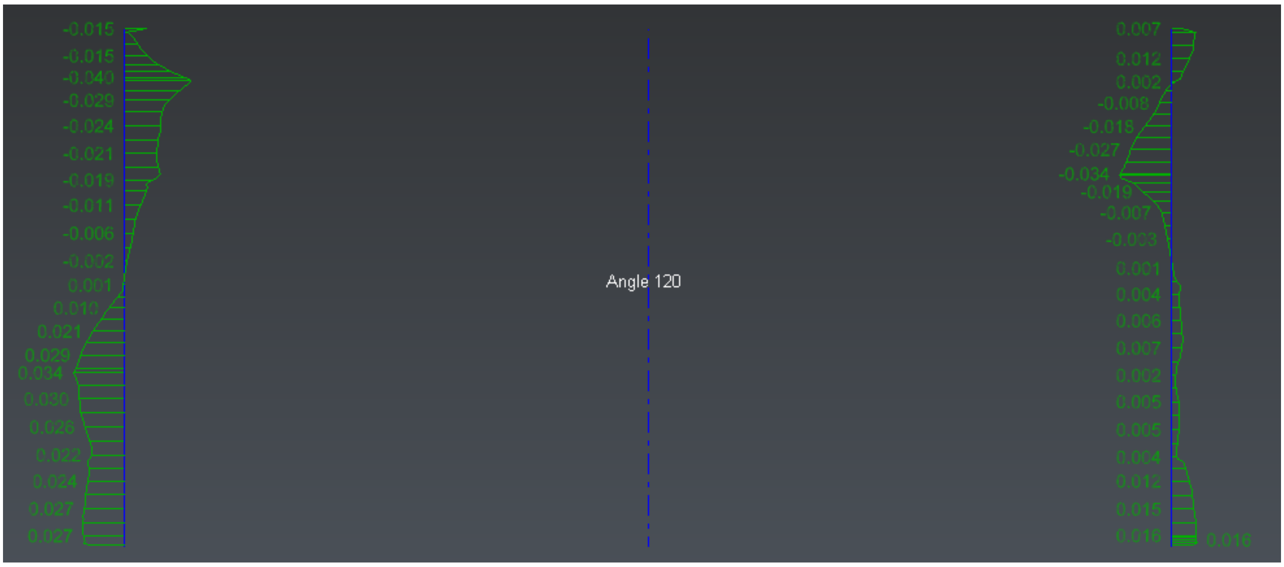
The first section is done on the Orientation Marker defined when creating the project. Compute the sections in counterclockwise direction.

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.



126 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 "**Tank mesh 1**". Select the polyline "**Tank mesh 1 Contour bottom**" in the "**Separate shell**" folder and launch 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.

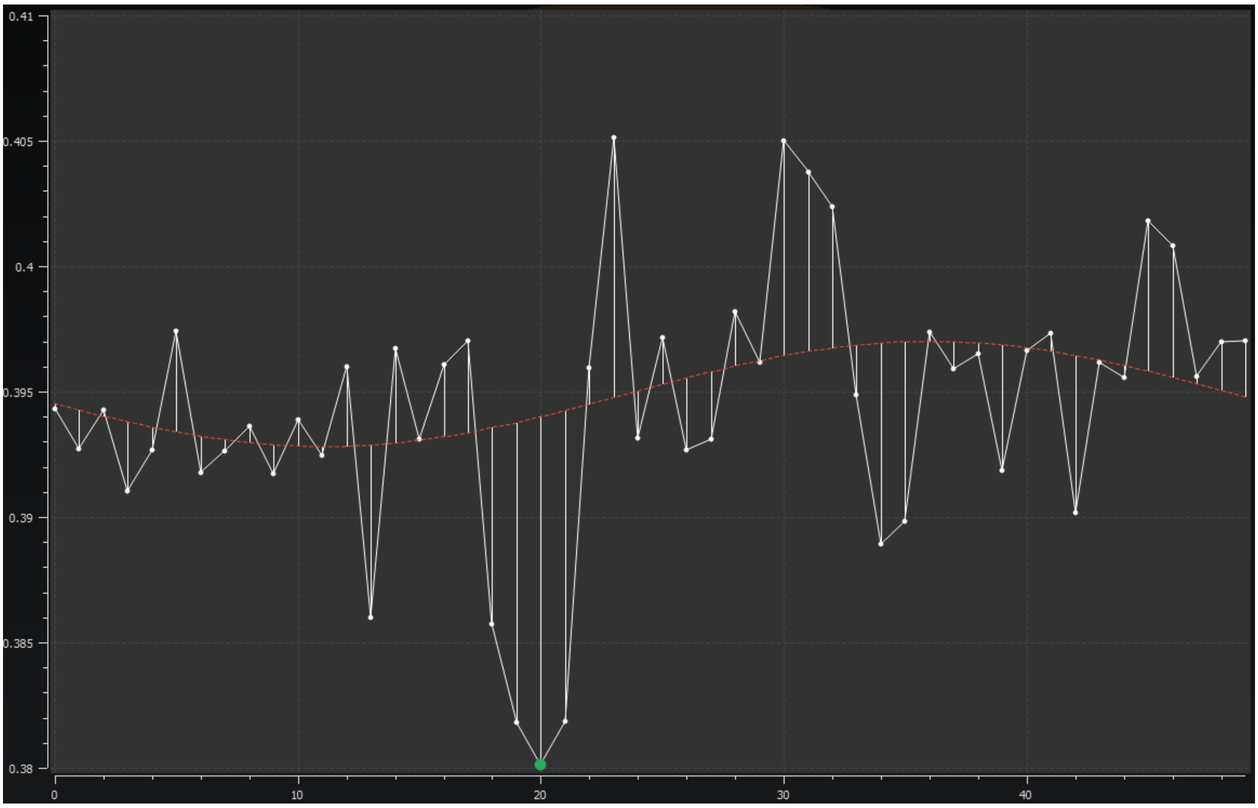
The results are displayed in the dialog box. A table is showing the results in each point:

- **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 the best cosine curve computation in order to improve the coefficient of determination R^2.

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 point when moving the mouse cursor on them.



127 Results from the differential settlement

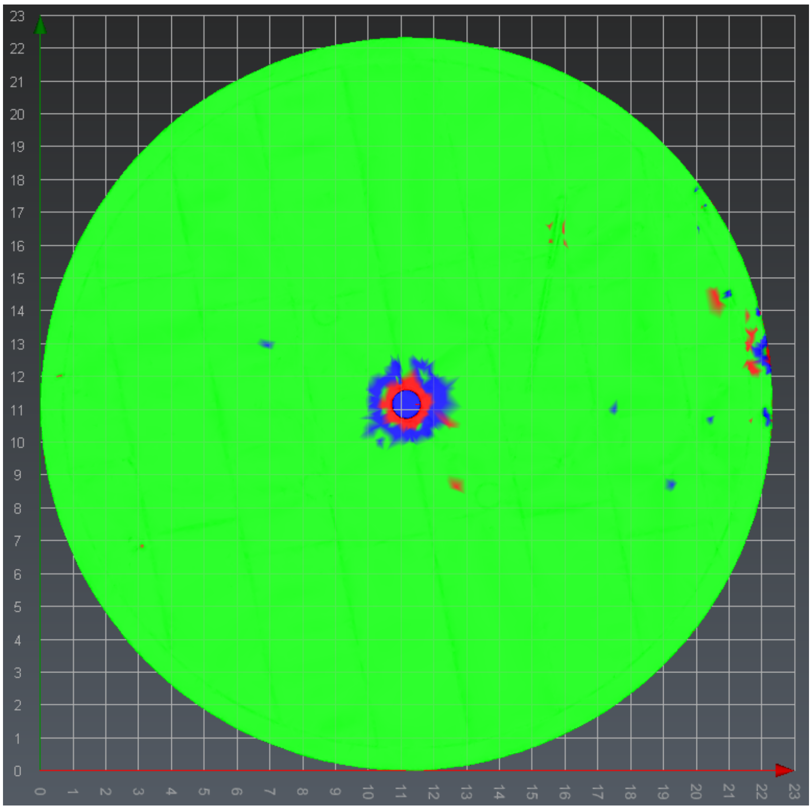
Click **OK** to validate the results. A new folder called "*Differential Settlements*" is created in the tree. It contains the 50 points used for the computation.

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 1 Bottom** from the **Separate Shell** folder. Select it and launch 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.



128 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 Report menu. 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:

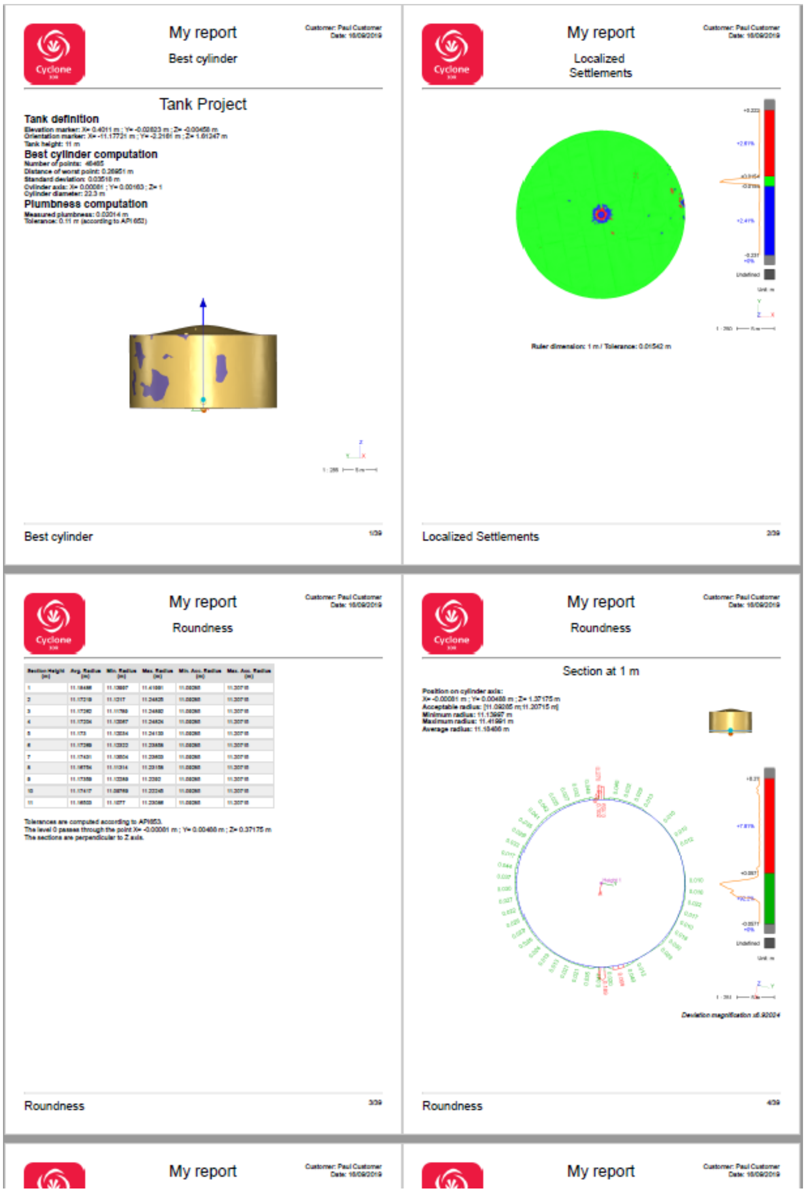
- Give a title to the project and to chapters.
- Enter the name of your customer and of your company.
- Insert the logo of both companies or any other image.
- Add and modify texts. You can drag and drop automatic fields from the Data panel.
- Add a global view as a 2D or a 3D scene.
- Edit all the views (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

129 Figure 1: csv export

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



130 Figure 2: report example

12 Image and Texturing

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.

12.1.1 Exercise: Texture a mesh with reference points

> Open the file “**TextureRefPoint.3dr**”, then import the following perspective images “**TextureRefPoint.jpg**” and “**TextureRefPoint-Distortion.jpg**”.

131 The monument to texture with reference points

Click reference points

Classic picture

Select the image "TextureRefPoint" and go to Estimate Pose. The view will be automatically divided into two parts:

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

Click now 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. The points should also be scattered all over the model. All the couples should be sufficiently distant in order to map correctly the texture. Click the **Remove** button in order to remove a pair or deactivate it when it is doubtful. Once you have entered four couples of points, you can see the position of the camera in the **3D View**.

> **Type of image**
>
> Make sure that the type of the image in the options is defined here in perspective mode.

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 7 couples of points. Note that in most of cases, 4-5 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).

At any time, when you think that your reference point definition is correct, click the **OK** button to validate.

> **Reference list of points**
>
> Save your list of pairs before validating the command if you want to edit it afterwards.

Then select the mesh and the image "TextureRefPoint" in order to launch Standard Texture.

> You also can texture a model with Smart Texture: refer to Exercise: Texture a mesh with spherical images.

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 **Fully visible triangles** 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 **Invisible parts** and click **Preview** again. All triangles will be textured.

132 On the left, only visible triangles are textured; on the right all triangles are textured, but it creates blurring zones.

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 on the corners. You can import the file “TextureRefPoint-Distortion.txt” in order to have a sample.

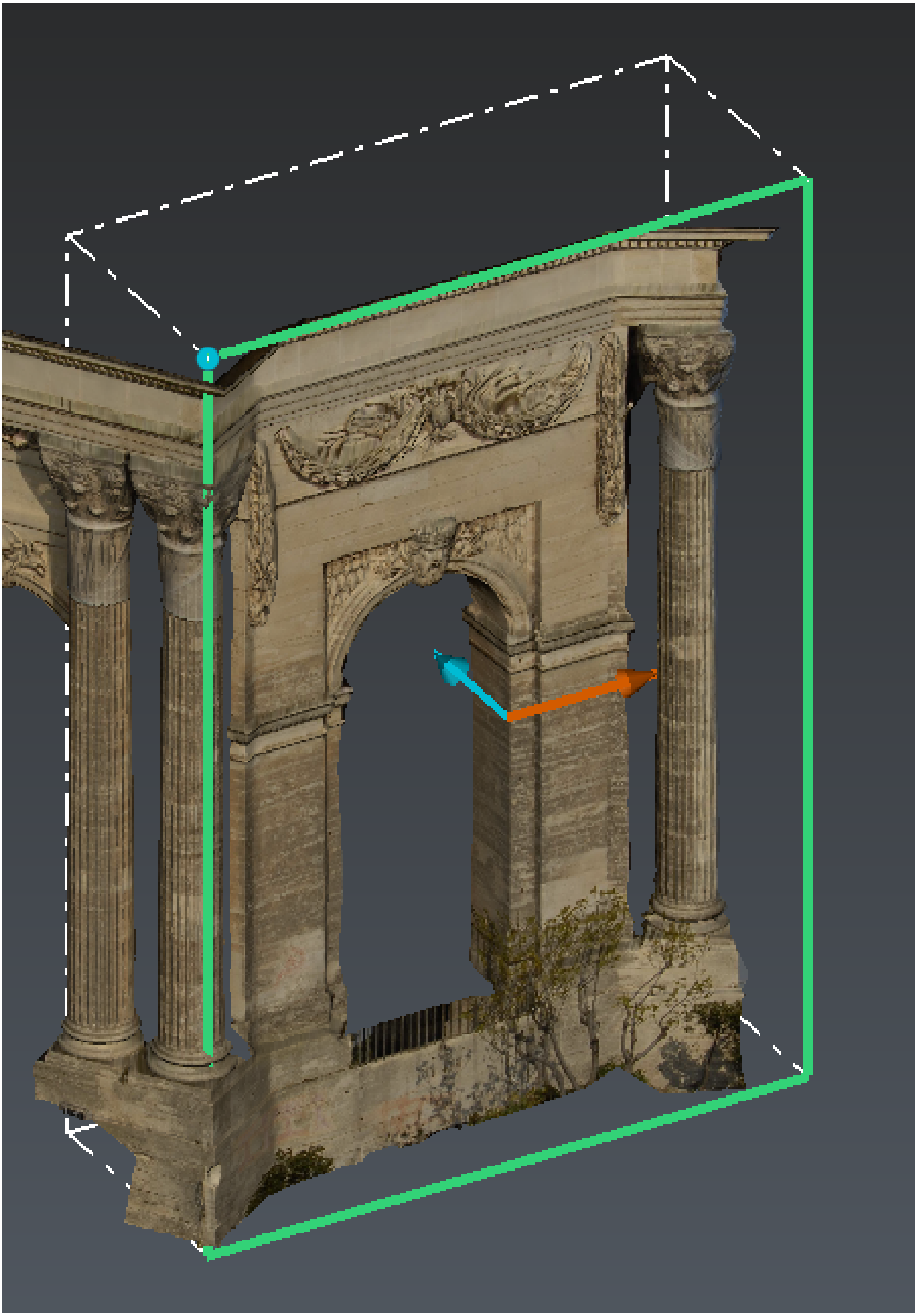
Create an ortho-image

Once the mesh is textured, you can for example create an ortho-image.

Launch Extract OrthoImage and choose **Elevation view** mode.

Click 2 points on the right wall, then set the **Depth** with the 3rd one. Adjust the limit box through the 3D scene (CTRL+SPACE) or through the dialog:

- **Width**: 10 m
- **Height**: 15 m
- **Depth**: 4 m
- **Origin**: 1035.75 5024 120.75



133 Export an ortho-image

Set the **Pixel size** to 0.01 m and the **Background color** to white.

Finally, **Save** the **image**. The created image has a resolution of 1000x1500 pixels.

A .txt or a .tfw or a .jgw 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

Open the file “**TextureParam&CameraPath.3dr**”.

Select the mesh “CliffTextured” and go to Extract Texture.

The images become available in the treeview:

- you can estimate their poses again to correct them.
- you can save the document again: this will create a folder, next to the file, containing these images.

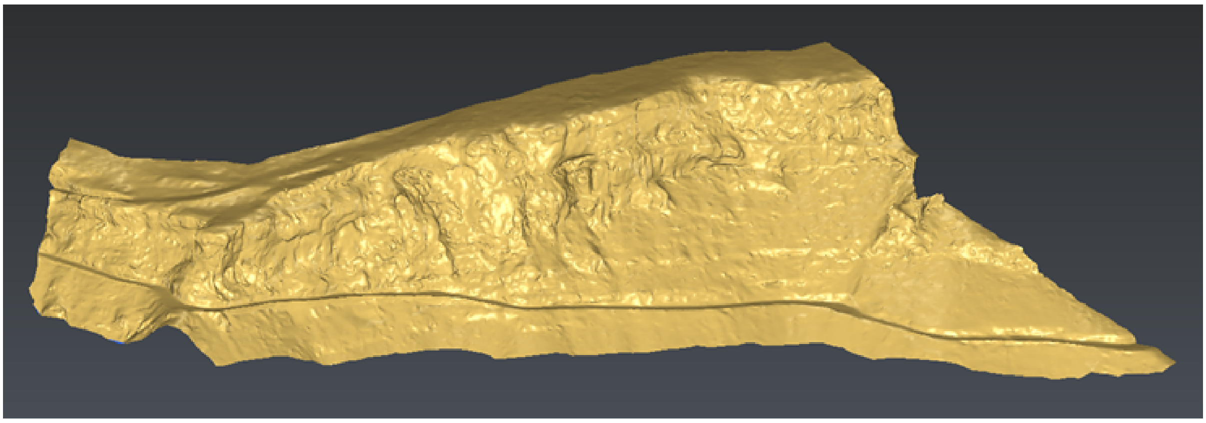
12.1.3 Exercise: Texture a mesh with camera parameters, adjust textures and export

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.3dr**” saved in the previous exercise.



134 The mesh to texture with camera parameters

Check external and internal parameters

Select an image in the treeview, 1029 for instance, and go to the Edit Image: **external and internal parameters are already filled** because these images come from an existing textured mesh. In most cases, you will have to define these parameters. External parameters can be computed thanks to Estimate Pose.

Internal parameters can be computed thanks to Calibrate Camera. The Edit Image dialog box goal is to edit manually these data. Then you can save or load these parameters (excam, incam or xml).

Internal parameters are:

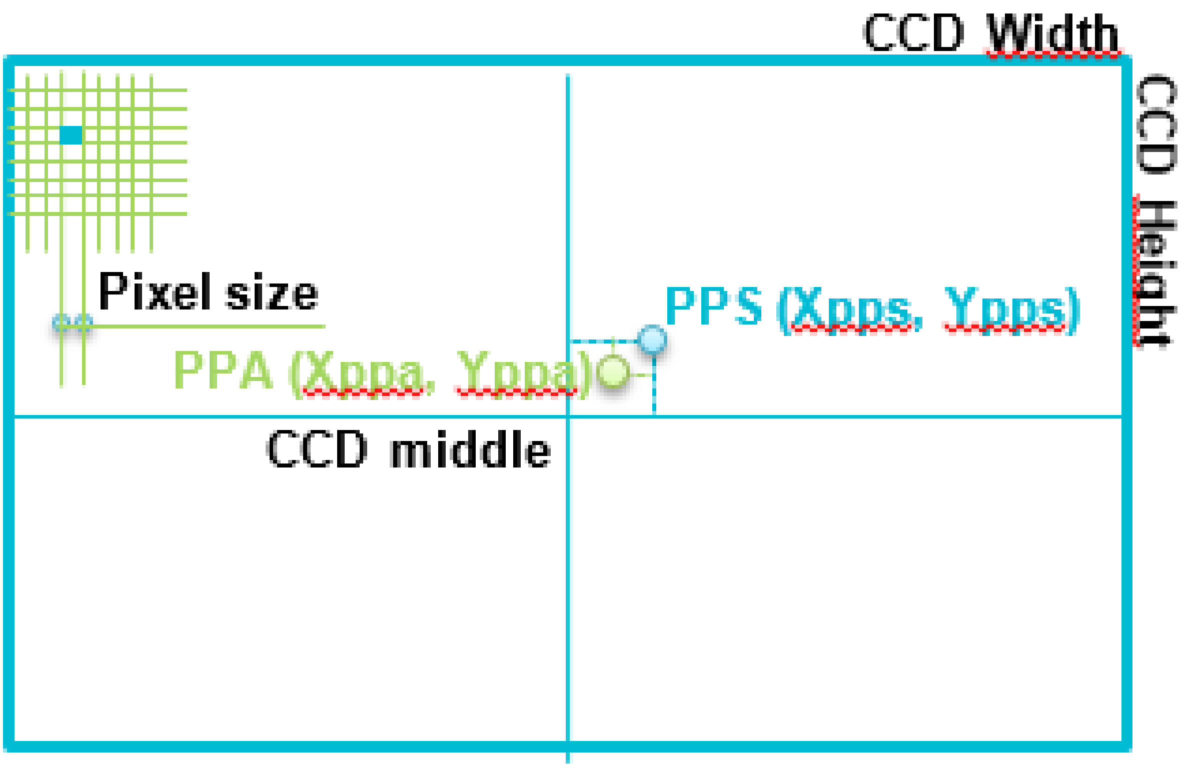
- The sensor size and the pixel size
- The focal length

General tab

- The lens misalignment
    - Principal point of autocollimation - PPA (offset from the PPS)
    - Principal point of symmetry - PPS (offset from the CCD center)

Distortion tab

- The radial distortion (from k0 to k6)
- The tangential distortion (from p1 to p2)



135 Camera internal parameters

External parameters are:

- The camera position
- The camera rotations

Learn more: Edit Image

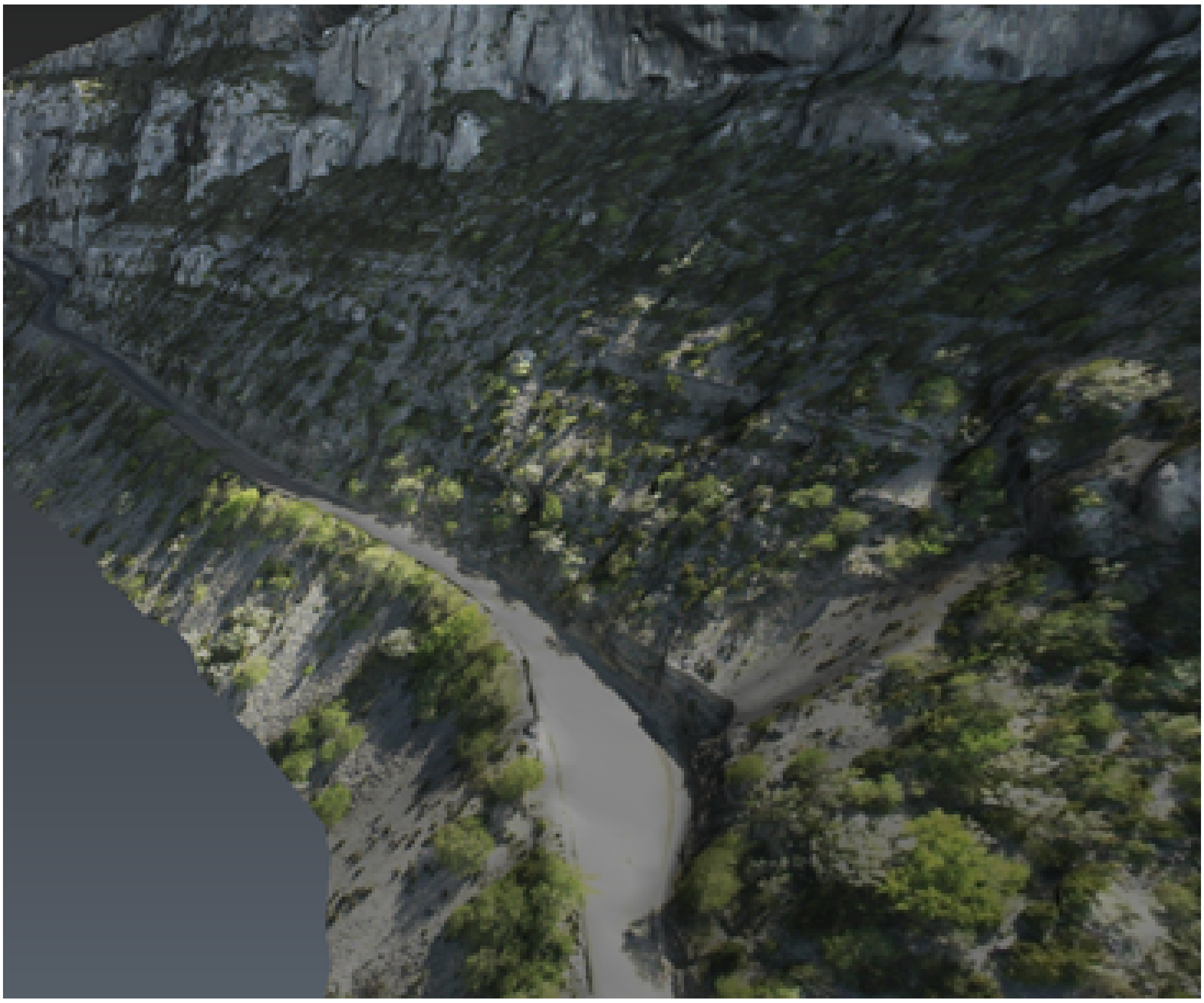
Apply textures

Once all parameters (internal and external) are entered for all textures, show only the mesh **MeshToTexture**, select it and the three images and go to Standard Texture. Choose to texture **Fully visible triangles** with **Best Projection** image choice.

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.

You can try other options (**At least one visible vertex** and **Invisible parts**) to texture blank areas. Click the **Preview** button again to see the difference.



136 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.

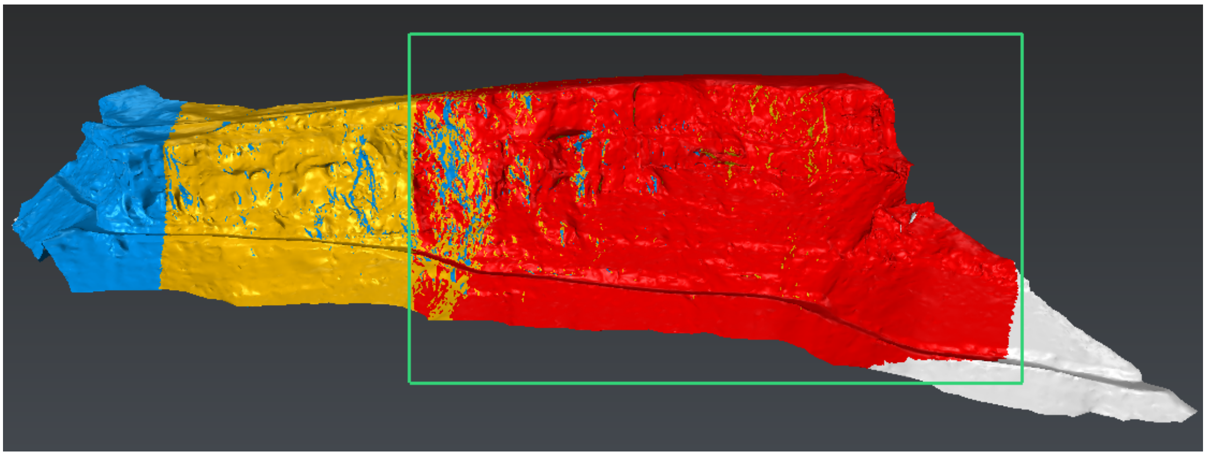
- The angle between the camera orientation and the triangle normal.

However, in some cases, you may want to select manually the texture to apply. To do this, show only the textured mesh, select it and go to Adjust Textures.

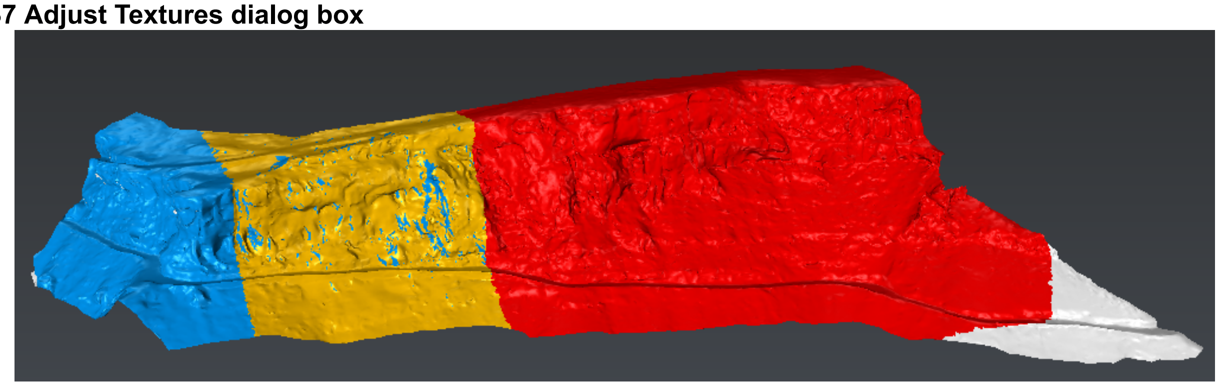
Thanks to the dialog box, **you can represent the textures by unique colors** or visualize them separately to better visualize where each one is mapped on the mesh. You also have tools to display images in the 3D scene.

In the central toolbar, you have **selection tools**, **filters** which are applied to the selection and **actions** which are going to be done on the next selection. For instance, select the texture 1032 (displayed in red in our example):

- Display **Colors** so as to better understand where textures are currently applied,
- **Select through** to select triangles hidden by others from user viewpoint,
- Select **No Filter** to select triangles hidden by others from camera viewpoint,
- Choose **Update** textures to replace previous applied texture,
- Draw a rectangle as shown below



137 Adjust Textures dialog box



138 Result of Adjust Textures

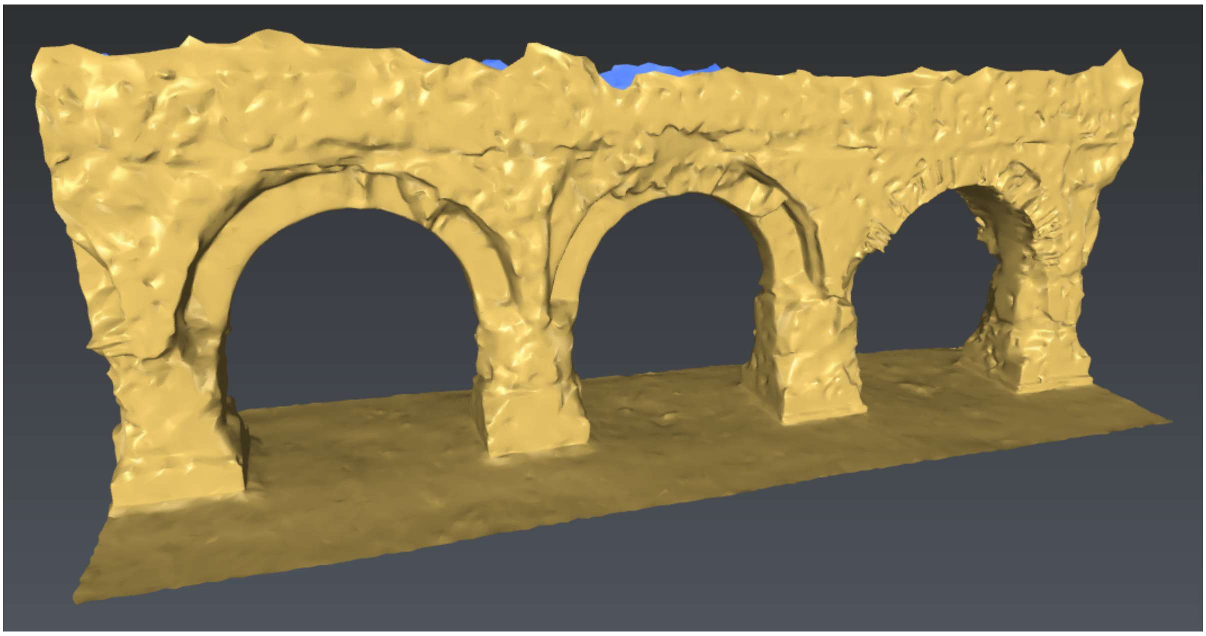
The selected area will take the color of the texture 1032.
Then select options **Remove Current Texture**, and “erase” some triangles on the right. They will become white, means that they will not be textured.
Click **OK** to validate.

Export a textured mesh

Now that the texture is applied and adjusted, we can export the mesh. You can select it, then go to Export. OBJ, GLB and FBX formats allow the 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 to remove them.

12.1.4 Exercise: Texture a mesh with spherical images

Open the file “**Aqueduct.3dr**”.



139 The aqueduct to texture

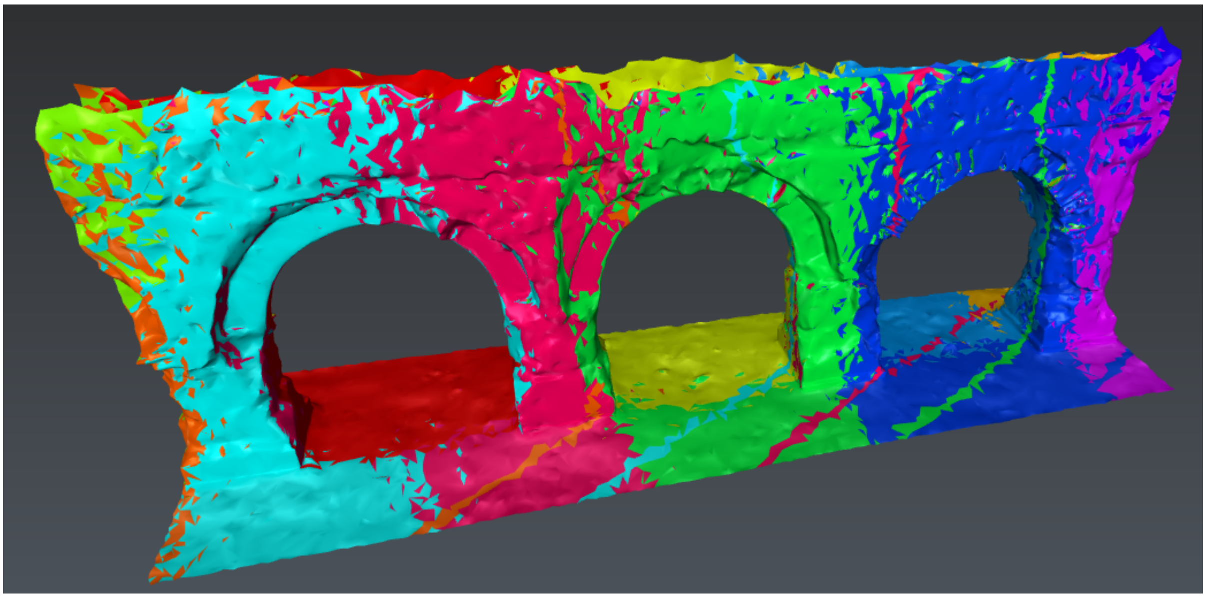
Smart or Standard Texture

In Exercise: Texture a mesh with reference points, the texturing was performed by Standard Texture. Now, we are going to use Smart Texture. These commands are almost similar in their use but technically different.

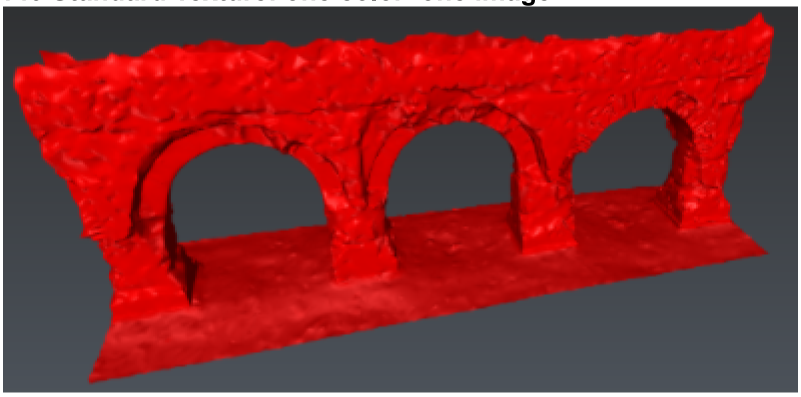
Standard Texture provides a dynamic texture: image areas are selected to be projected onto a mesh. If the mesh is modified, for instance by a smoothing, the areas will continue to be projected onto the mesh. That is why, you can also modify the pairing between images and the mesh using Adjust Textures. Note that the original images are fully stored in the mesh object: you can restore them using Extract Texture.

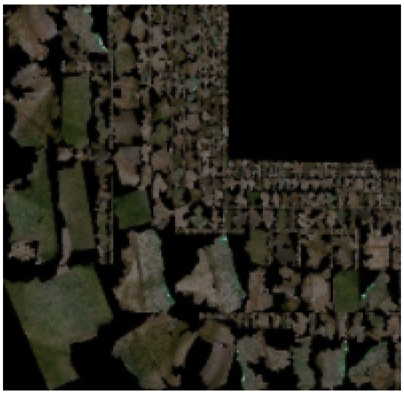
Smart Texture provides a static texture: the texture is painted on each face. Any mesh modification, done later, will affect the texturing. Thus Adjust Textures is only suitable to remove the texture locally, for instance to replace the texture by another one (Texture from Clouds, Texture Material or Standard Texture). Note that the mesh will store a texturing map (not the original images).

Beside these differences, Smart Texture can be used with more images at the same time and supports masks. For instance, when using images with blank areas (spherical images), the corresponding areas will no longer be used.



140 Standard Texture: one color=one image



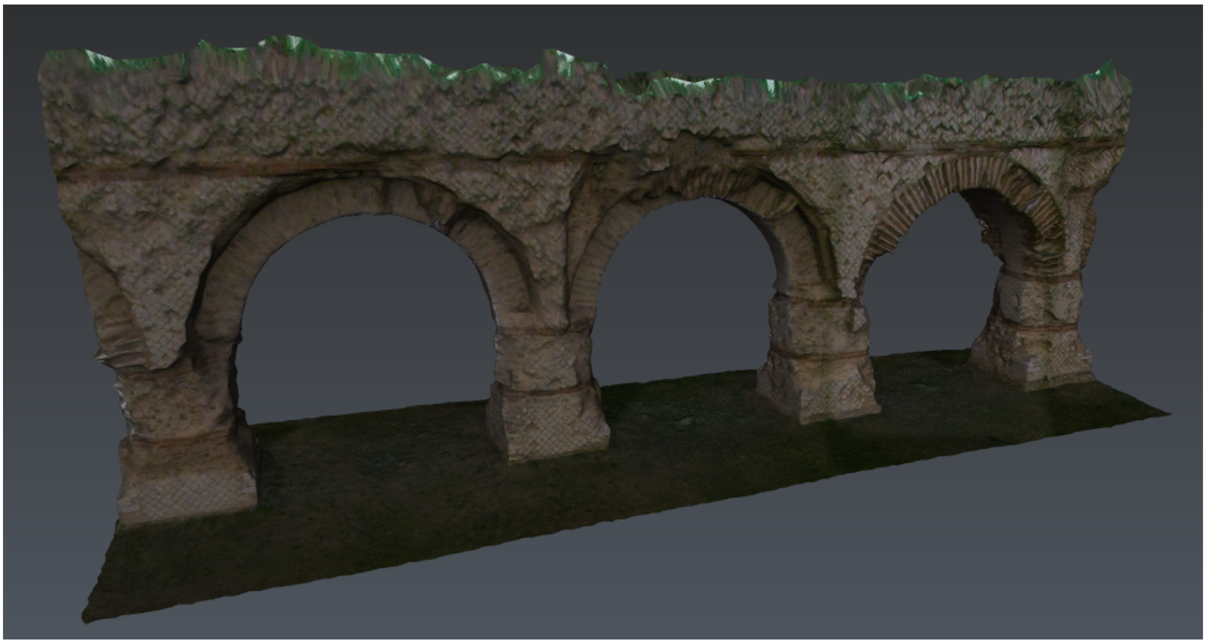


141 Smart Texture: one color=one texturing map

Smart Texture

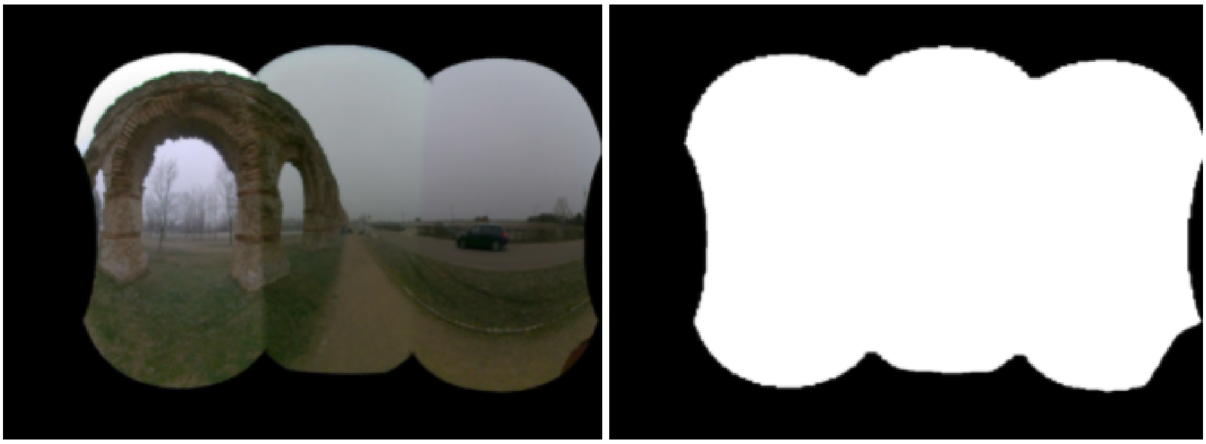
This sample was measured by a Leica BLK 2Go and imported from a LGS file. Then a mesh was created.

Select all images and the mesh, then launch Smart Texture. Set the **Pixel Size** to 5mm and run the computation.



142 Smart Texture result

Note that, as expected, the blank areas are not used. Open the Image Preview of Spherical 1. You can switch the preview between the image and its mask.



143 Image Preview

If the automatic mask is not suitable, you can clear it using Clear Mask.

Adjustments

If some areas are still incorrect or missing, it is possible to remove the texture locally using Adjust Textures. Select the mesh and run Adjust Textures. Choose the **Textures** display mode and the **Remove All** option. Then, remove a small area.

Now, you can complete the texture using another images or the real color of the cloud. Refer to Exercise: Texture a building mesh using several textures.

12.1.5 Exercise: Texture a building mesh using several textures

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

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

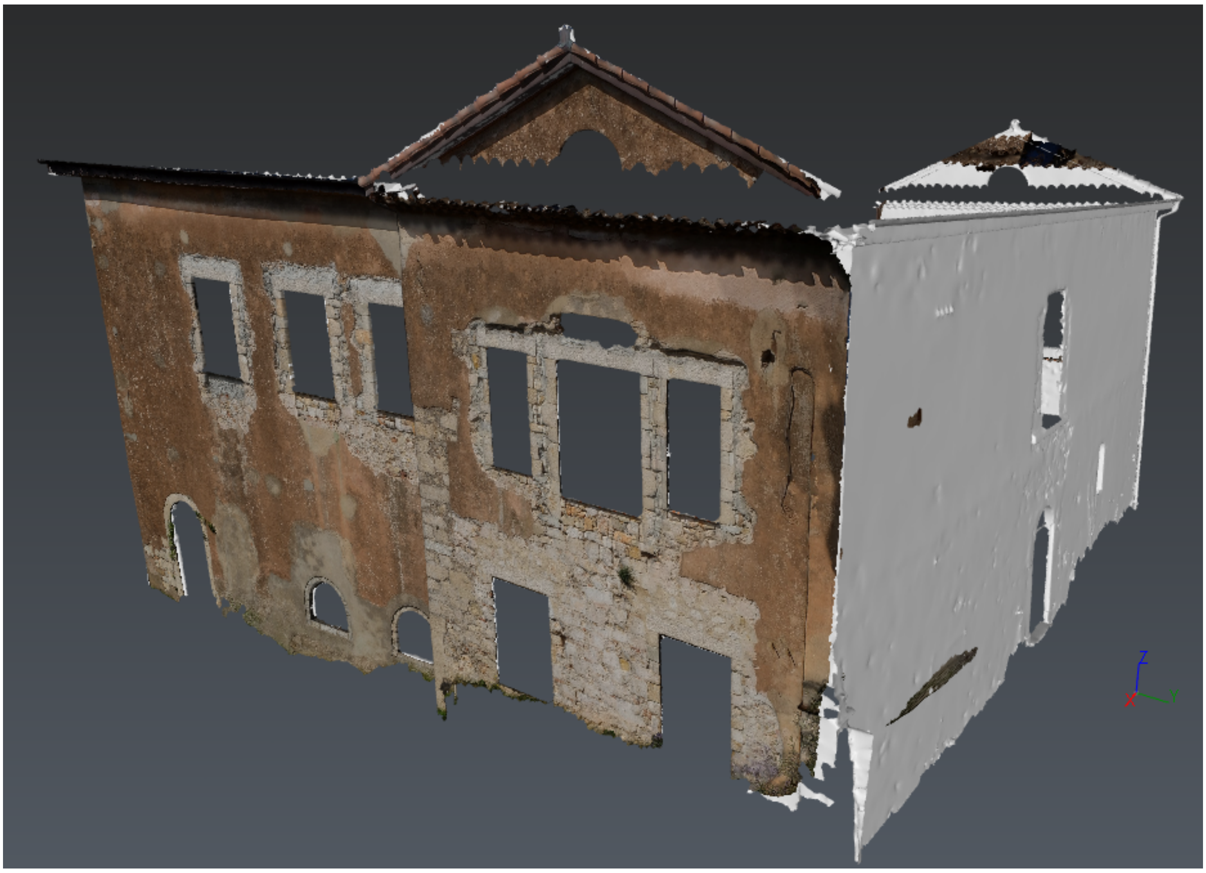
Open the file

Open the file **TexturePhotomodel.3dr**. It contains a mesh, a point cloud with real colors and two images.

Texture the main frontage with an image

Select the mesh and MainFrontageImage, then launch Standard Texture.

Choose the option **At least one visible vertex** and click **OK** to validate.



144 Fig.1 Standard Texture
Now, the first side is well textured. However, the image overlays other walls, which are now partially textured. We are going to correct it in the next step.

Adjust texture on the side wall

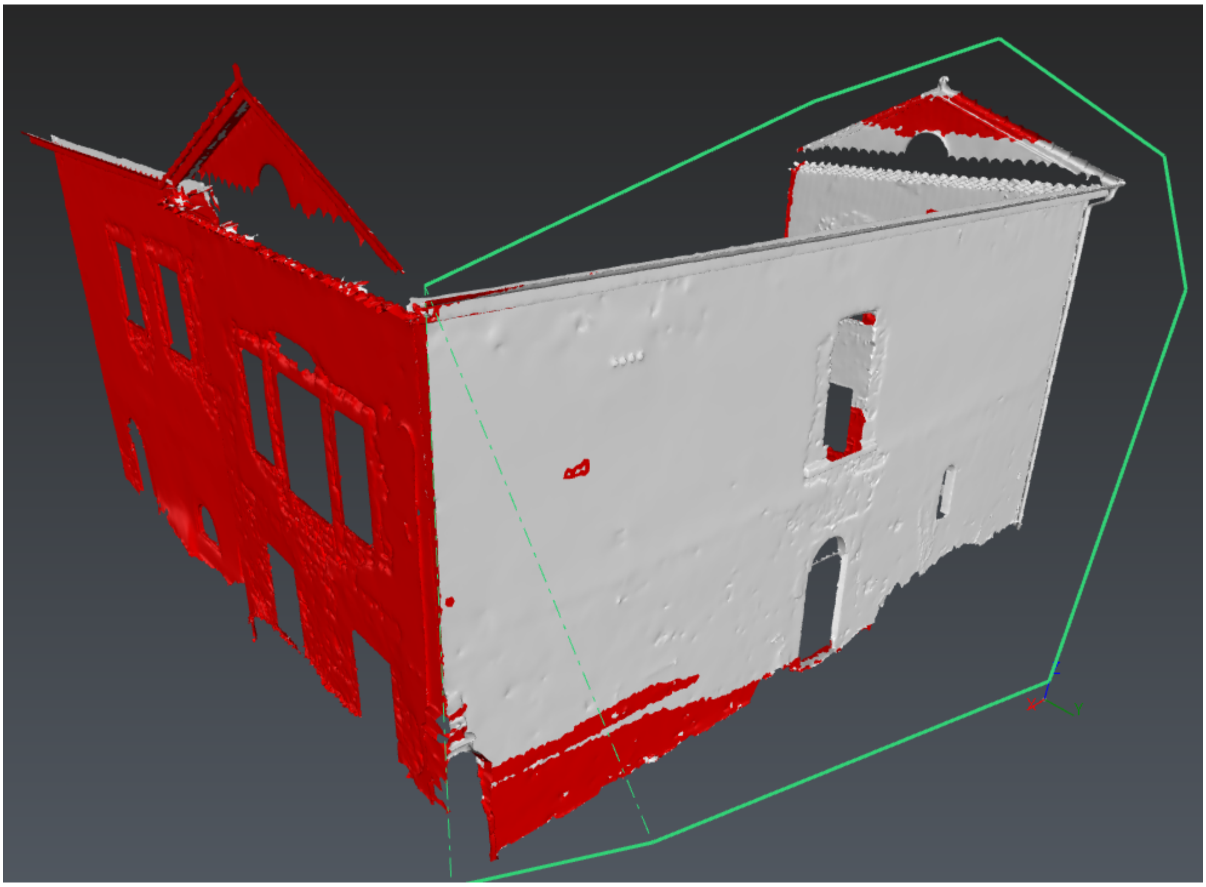
Before texturing other faces of the building, we will first remove the texture coming from the image from walls where it should not have been applied.

Select the mesh and launch Adjust Textures.

From the toolbar, choose **Select Through**, **No Filter** and **Remove Current Texture**.

Rotate the view as below to remove only the texture on the other sides.

Click **OK** to validate the command.



145 Fig.2 Erase a texture

Texture the side wall with a material

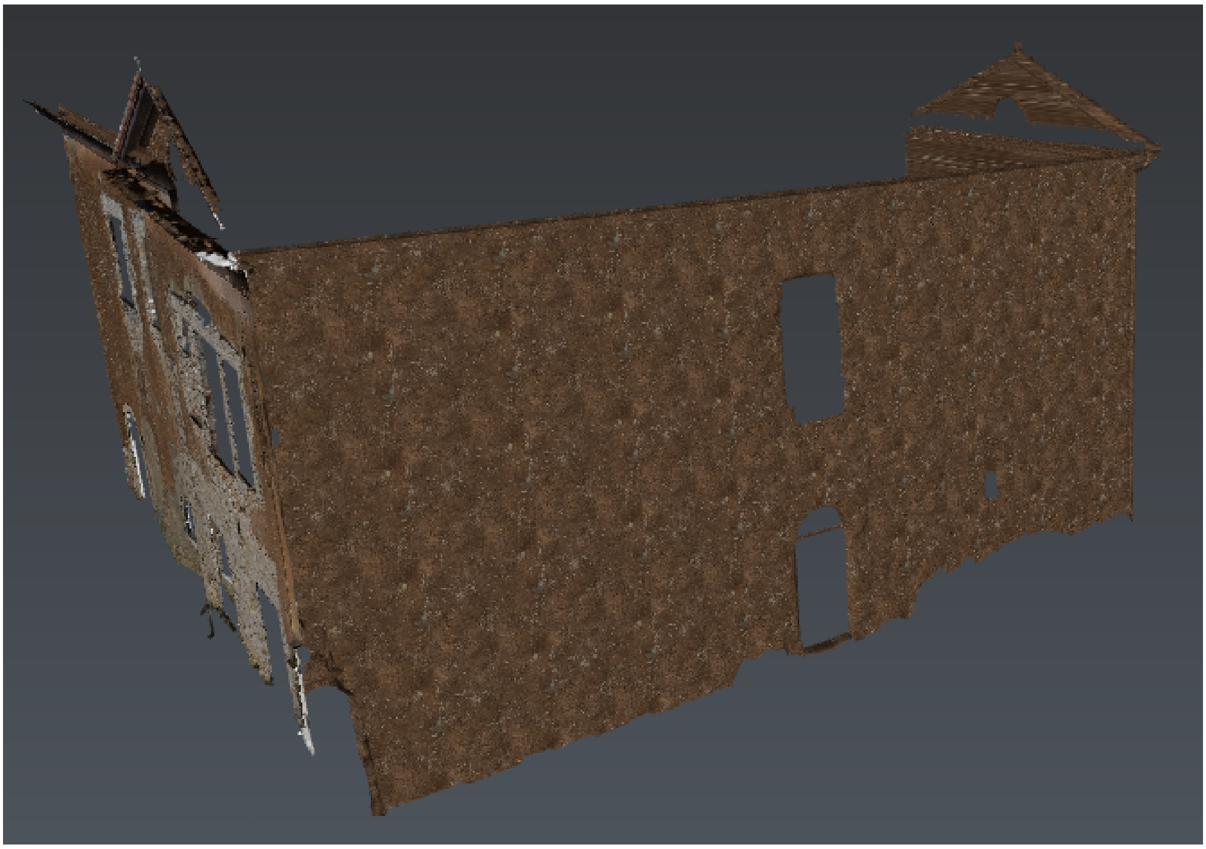
The next step will be to texture the side wall using a material.

Select the mesh and TextureMaterial image, then launch Texture Material.

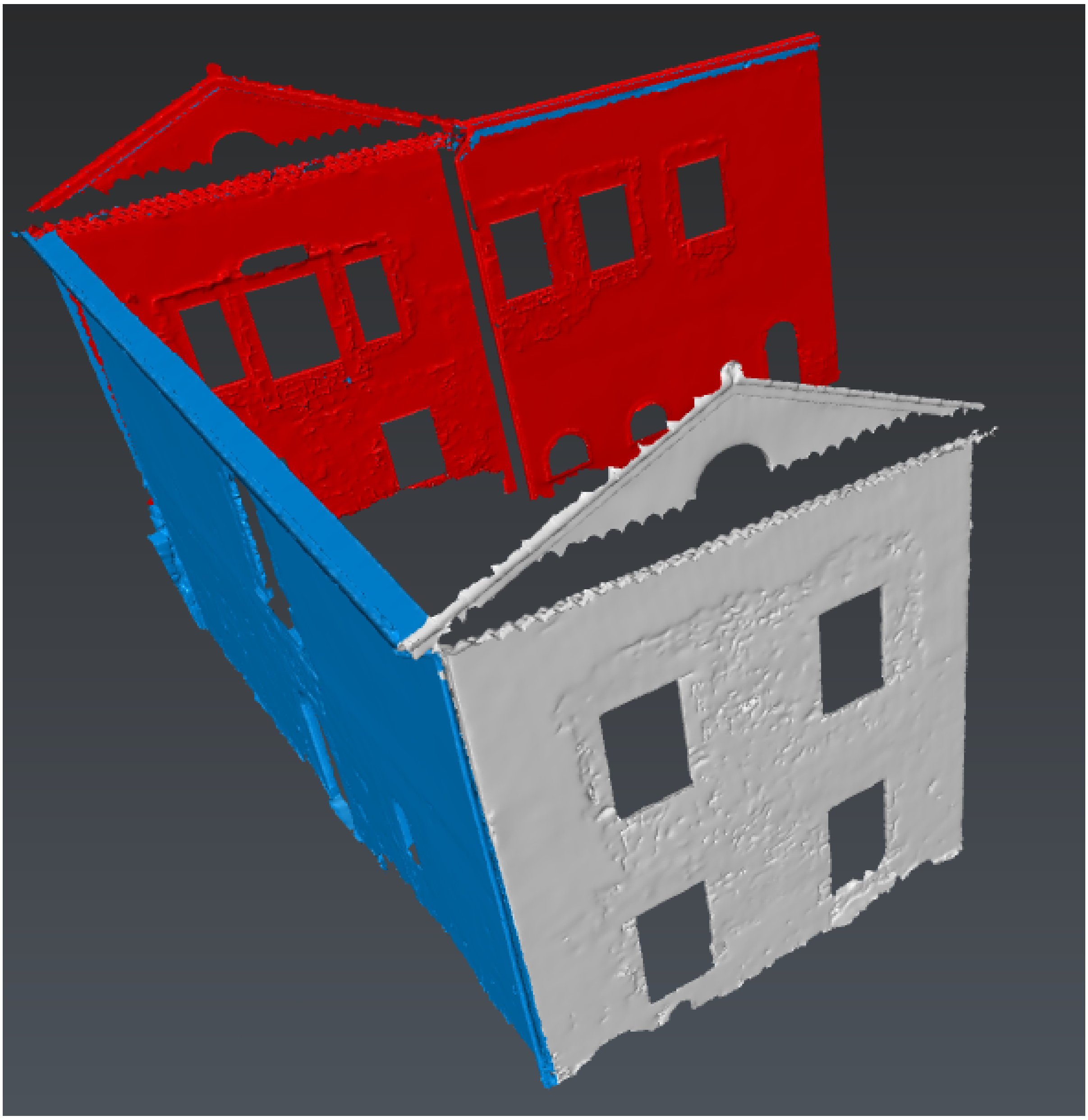
Choose to **Keep proportions** and select **Preserve existing texture**.

Set the **View direction** using the normal of the second frontage (refer to Define normal direction).

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.



146 Fig.3 Complete the texture with a repetitive pattern
Then remove, the repetitive texture on the last side using Adjust Textures.



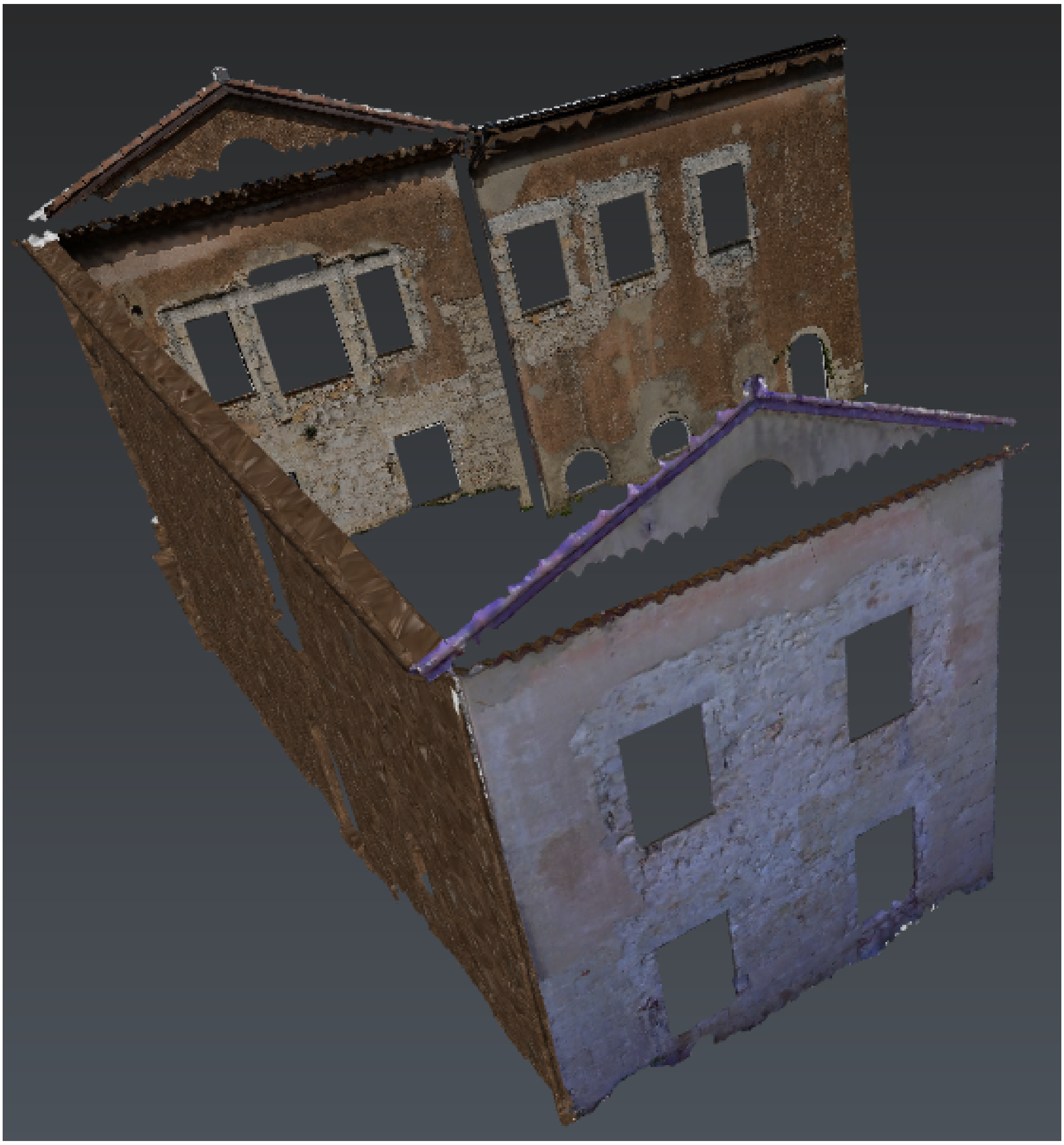
147 Fig.4 Adjust Textures (2)

Color the mesh with a point cloud

We will now create a texture on the last side using the colors coming from the original point cloud.

Select the mesh and the point cloud and launch Texture from Clouds.

Set the **Maximum distance** to 5 cm, the **Pixel size** to 1 cm, the **Default color** to white and select the option **Preserve existing texturing** in order to keep the previous result.



148 Fig.5 Texture using the cloud
Now, you can enjoy your photomodel.

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 **FrontageOrthoImage.3dr**.

Launch the command Extract OrthoImage and select Elevation view for the **OrthoImage mode**.

Give the 3 points as described in the picture below to define the frontage to map (ensure XYZ mode is enabled in the toolbar). The 3 points can be clicked accurately in orbit orthographic camera mode or given by coordinates.

149 Frontage to map

Now, you can adjust the defined box to capture exactly what you need. This is possible either through the scene by editing the limit box (in 3D frame view) or though the dialog box by editing the **size**, the **depth**, and the **top left corner**. To keep horizontal lines, horizontal in the orthoimage, it is advised not to modify **Horizontal** parameter.

You can activate the **Preview** to visualize the changes in real time.

150 Image preview

Set the image properties parameters as below:

- **Pixel size**: 0.01 m,
- **Point/Line magnification:** 1,
- **Background color**: white.

Then **Save image** as a picture file. 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

Warning

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

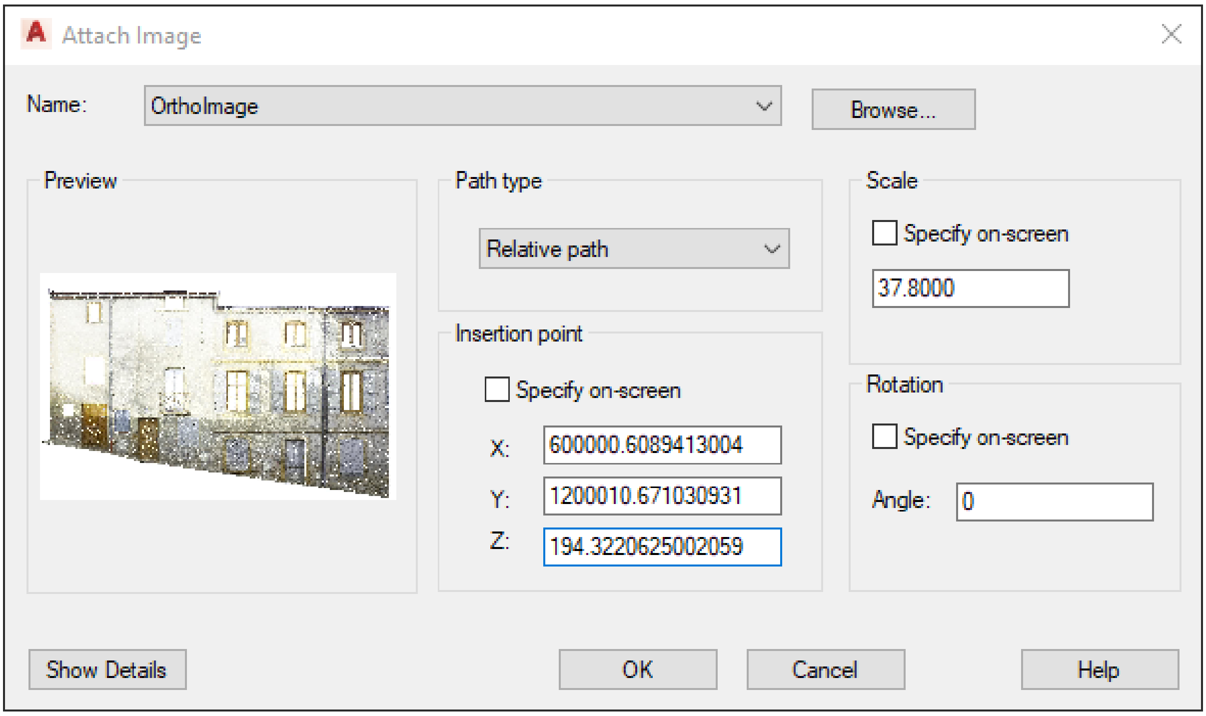
Open **orthoimage.txt** with a text editor like Notepad and an empty file with AutoCAD.

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 from the ribbon.

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



151 Attach Image

Rotate the image in AutoCAD

Now, it is necessary to rotate the image.

> **Warning**
>
> If more than 1 rotation is needed, remember that a first rotation along an axis will modify the 2 others axis.

If rotation Z is not 0,

1. enter **ROTATE3D** in the command line,
2. select the image,
3. select the Z axis direction,
4. select bottom left corner in image object and copy-paste **rotation Z.**

Then if rotation X is not 0,

1. enter **ROTATE3D** in the command line,
2. select the image,
3. use **the bottom left and bottom right points of the image object** to define the rotation axis (X'axis)
4. copy-paste **rotation X.**

Finally, if rotation Y is not 0,

1. enter **ROTATE3D** in the command line,

2. select the image,
3. use **[the bottom left and top left points of the image object]** to define the rotation axis (Y"axis)
4. copy-paste **rotation Y.**

Command line prompt

```
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]: 179.2574996998036

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]: 90
```

Now the OrthoImage is inserted in 3d in Autocad scene.

12.2.2 Exercise: send an ortho-image to AutoCAD

Open the file **TextureParam&CameraPath.3dr** and a new empty file with AutoCAD.

Warning

The plugin for AutoCAD is needed to do this exercise.

Launch the command Extract OrthoImage and select the **Top view** mode. Set the parameters (for example):

- **Width**: 600 m,
- **Height**: 220 m,
- Uncheck **Depth,**
- **Top left corner position**: X=1741210; Y=2298885; Z=1000,
- **Horizontal direction** X=0.5; Y=-1; Z=0,
- **Image pixel size**: 0.25,
- **Background color**: white.

You also can define the frame by editing the limitbox.

Select the **Send To** button.

Send to AutoCAD

Make sure that AutoCAD is open and in relation with to the good version of Cyclone 3DR. Enter in AutoCAD the command line 3DRCHOOSE VERSION to select the good Cyclone 3DR version.

The ortho-image has been inserted in your DWG drawing, in the active layer.

> The ortho-image file is added in 3DR Temporary folder (General settings).

12.3 Virtual visits

In the software, you can create and export a video of a virtual visit in the 3D scene. You can either import, edit or 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.3dr**

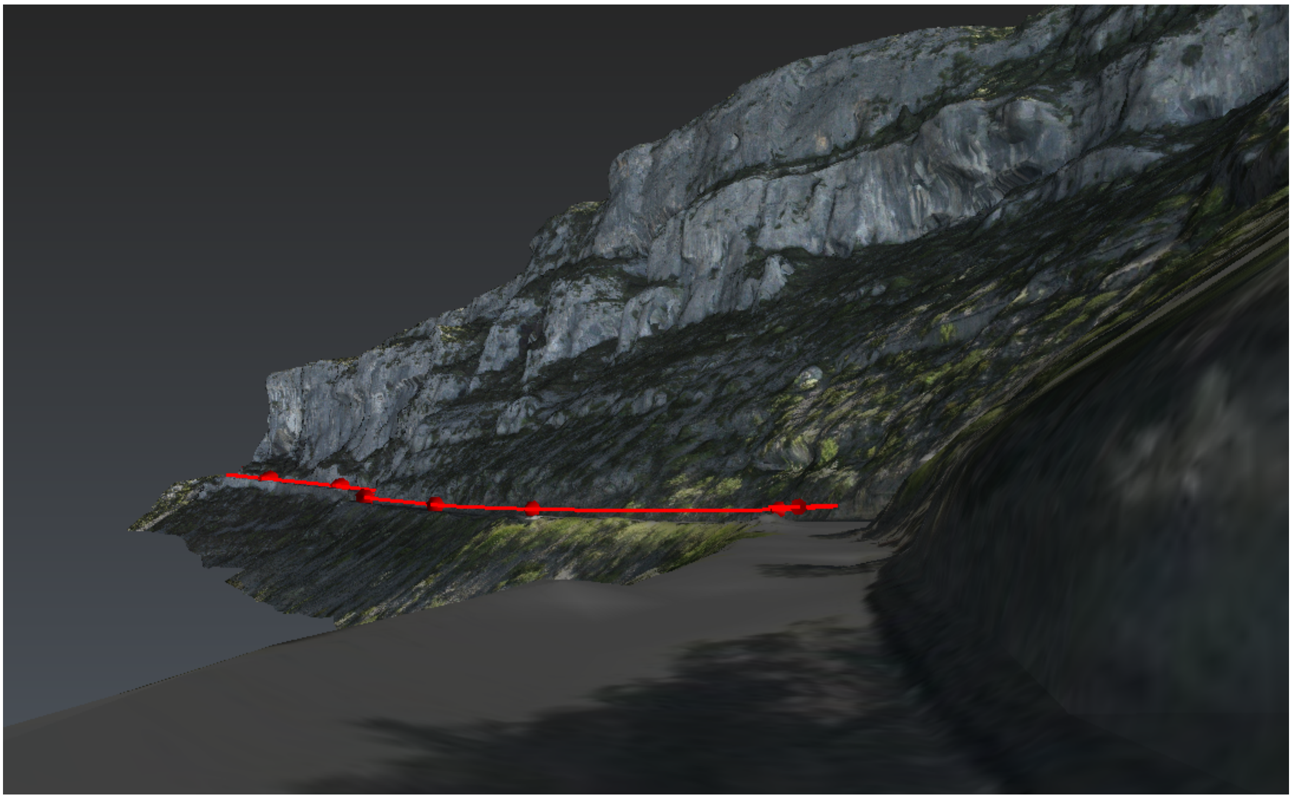
This file contains a textured mesh of a cliff, also used for the texturing exercises.

First, it is important to check the direction of the polyline because, by default, the camera orientation will follow the camera path selected. You can right click on the polyline and check **Reverse** to reverse its direction if needed. Note that polylines are represented with arrows thanks to a parameter available through the colored disk in the tree view. Then, use a perspective mode, select the polyline **Camera path** and launch the command 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 (back and forth if the polyline is open).

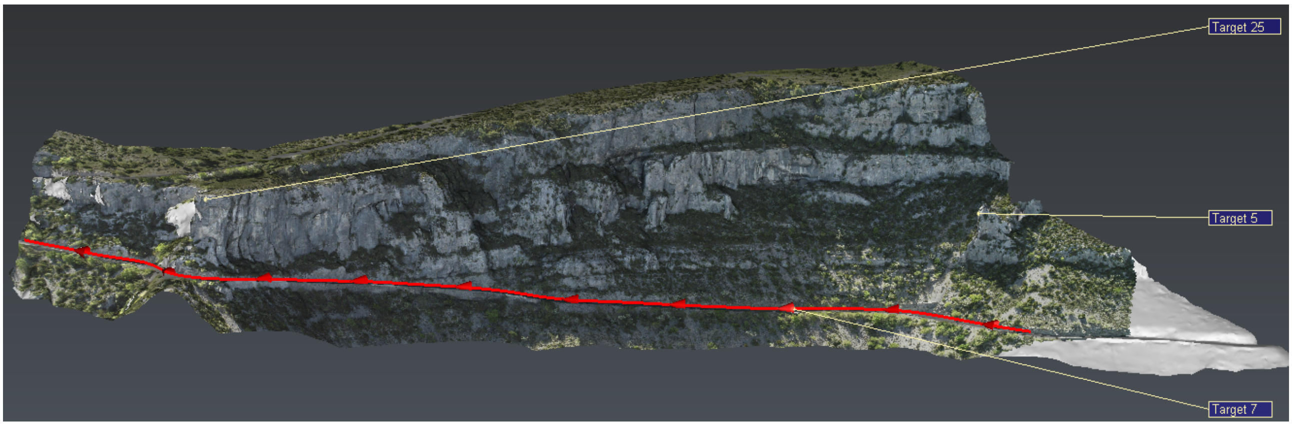
Optionally, you can set the up vector of the camera by setting the view in the good orientation and clicking on **Use current**. For example, to set the up vector as the Z axis, click in the scene with your mouse and press the key **X** or the key **Y** to have the Z axis perfectly vertical, and then click **Use current** to set the up vector of the camera.

By default, Z is always the Up vector so you should not have to change this option, except in specific cases.



152 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**.



153 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 or MP4 file.

> See the Help files of the Camera Path 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.3dr**

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

Go to Camera Scenario. Set the 3D scene in the desired view and click on **Add 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 **Position** value 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 setting the right number with **Position** and clicking **Delete Position**.
If you would like to turn around an object, you could check the option **Close Path**.

When you click **OK**, a camera path is created so as to continue with Camera Path (as explained in previous exercise).

13 CAD and Reverse Engineering

13.1 Generalities about Reverse Engineering

13.1.1 Introduction

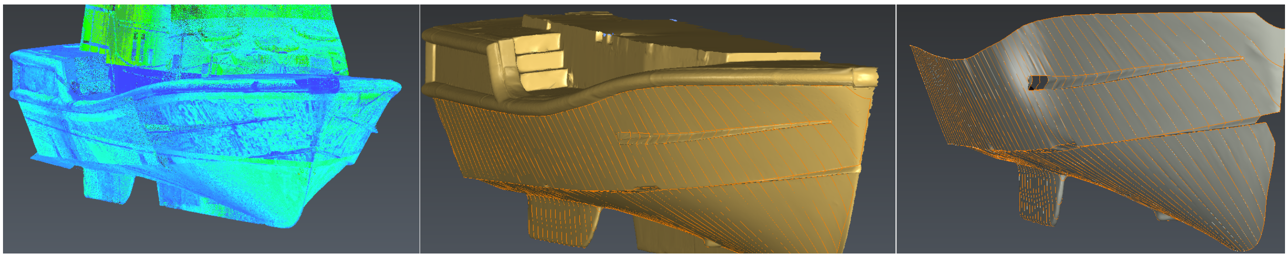
The polygonal mesh modeling created by the software generates models, made up with hundreds or thousands of non-continuous triangles. These 3D meshed models are ready for rapid prototyping, tool path generation, simulation, analysis, etc.

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. The CAD Surfaces generated are 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.
- Finally, we create NURBS/BSpline surfaces using previous BSpline curves. These surfaces are fitted on the mesh.



154 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.

This process can be manual or automatic.

Automatic or quick network extraction

1. You can use the command Create Network which creates a grid network of polylines on the selected mesh(es).

2. This grid can be edited Edit Network in order to improve the delineation locally (select the mesh and the network).
3. Generate the CAD surfaces from a mesh and the polyline network Generate Patches. Select the mesh and a polyline network.

Manual network

To make a manual delimitation you must follow these rules as much as you can:

- Make borders on zones having same curvature characteristics: lines along a small radius, lines along sharp edges, etc.
- Create polylines that intersect so that the software can easily determine the accurate intersection. To do so, **you can use all available polyline creation tools** (freehand sections, planar sections, single break line, etc.), and all polyline editing tools (in particular Edit Network or Stretch Polyline command).
- The lines that are created must lie “on” the mesh (projected); otherwise some surface reconstruction may fail.
- Make contours with 4 sides (wherever possible).

Make smooth polylines along curvature discontinuity

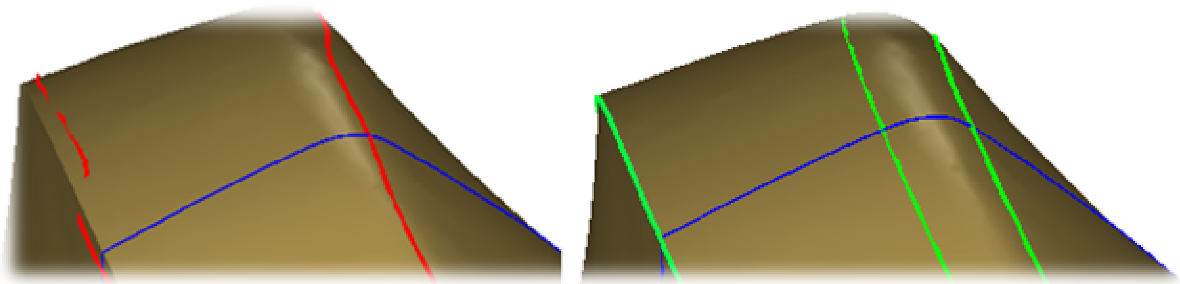
The goal of the surface reconstruction is to divide the complete surface of the model into elementary surfaces called “patch” or “face”. The most interesting property of the NURBS / BSpline mathematical definition is that the surface is continuous. “Continuous” means that the shape changes smoothly from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



155 Polylines along curvature discontinuity; lines should be as on the right, they are smooth and correctly positioned on the discontinuities.

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:

Border with 4 sides	This is the ideal situation.
Border with less than 4 sides	The algorithm will create a **degenerated face**, which means that the mathematical definition will keep 4 borders but some border(s) will have a null length.
Border with more than 4 sides	The algorithm will create a **trimmed face** from the input contour. First a bigger patch is generated, covering the entire contour. Then this patch is fitted on the mesh and finally a restriction is applied using the input contour. This case is the most complex.

13.2 Generate patch on a mesh

The main command to create surface patches is Generate Patches. This command is very powerful because it drives you directly from a set of polylines to the set of patches.

You can launch the command by selecting a network (polylines, set of polylines, BSpline, etc.) and a mesh.

Open the file **Reverse-MetalSection.3dr**.
This file contains a mesh and some polylines created with the following commands:

- Single Breaking Line
- Resample
- Create Network

Select the mesh and all the polylines from the folder **Contour group**, then launch Generate Patches. At startup, the command will automatically process a set of operations to give a first result:

- Lines of the network are approximated by BSpline curves with an automatic tolerance (by default the software tries to find the best compromise between accuracy and smoothness).
- Patches are created from the curve network and fitted on the mesh. Hide the input mesh.
- Each time the computation has finished, information regarding the standard and maximum error deviations will be outputted to give an idea about the overall accuracy of the CAD model:
  - Deviation between the input lines and the corresponding CAD curves
  - Deviation between the mesh and CAD faces

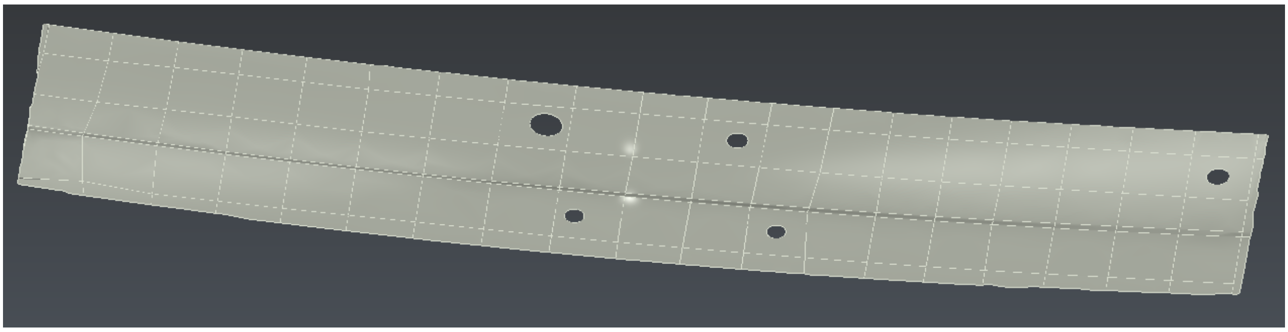
If you want to locally adjust curve approximations, you can select a single curve and edit the slider value (deviation values change in the dialog box). The new curve will be automatically recomputed and displayed in the 3D scene. Then the next computation will only occur on CAD faces whose borders have been modified.

Feel free to reproduce the network using the commands listed above.

When some patches are missing, it is possible to reconstruct them individually using a CAD wire and Fill Surface.

Sometimes it is interesting to make holes directly on the patches: this is useful if you want a smaller number of patches or if you want to avoid tangency problem between surfaces. To do this, the best way is to have a patch larger than the hole.

- Extract the mesh holes with the command Click Holes and Borders to get a polyline around each hole.
- Select the corresponding polylines and launch the command From Polyline to transform them into NURBS curves. Choose **Auto** mode.
- Ungroup the CAD surface using Ungroup CAD in order to select the patches individually.
- Select each patch and its hole and launch Hole. In a similar way you could change the patch external contour using Surface Restriction.
- If necessary, reverse the patch using **I** shortcut key.
- Finally, select the visible patches and create a shell with Group CAD



156 CAD result

When you have reconstructed your CAD curves and surfaces, you may want to use them with other CAD-CAM software. Select objects that must be exported and go to the Export and choose IGES or STEP format.

14 BIM Inspection (Touch Mode)

14.1 Introduction

In this exercise, you will learn how to do an inspection between a scan and a BIM model using the Touch Mode interface.

For time-saving purpose, the BIM Inspection workflow relies on a specific file called Preparation file.

> **Learn more about the BIM Inspection workflow**
>
> More infos here: BIM Inspect

The exercise will first help you to make a correct preparation file then how to use this file for the inspection.

14.2 Content

- Create the preparation file
- Inspect the scanned data with the BIM object

14.3 Create the preparation file

> Start the software in **Desktop Mode**.

The preparation file is mandatory to start a BIM Inspect workflow.
It must include separately data to register your point cloud and data which will be inspected.

The goal of this step is to prepare a file containing three BIM objects from the content of the IFC file (a building model):

1. **site:** the main model, used for the registration with the scan.
2. **columns:** the specific elements of the BIM project we want to inspect.
3. **other:** other element that may serve for other purpose.

14.3.1 Import BIM data from an IFC file

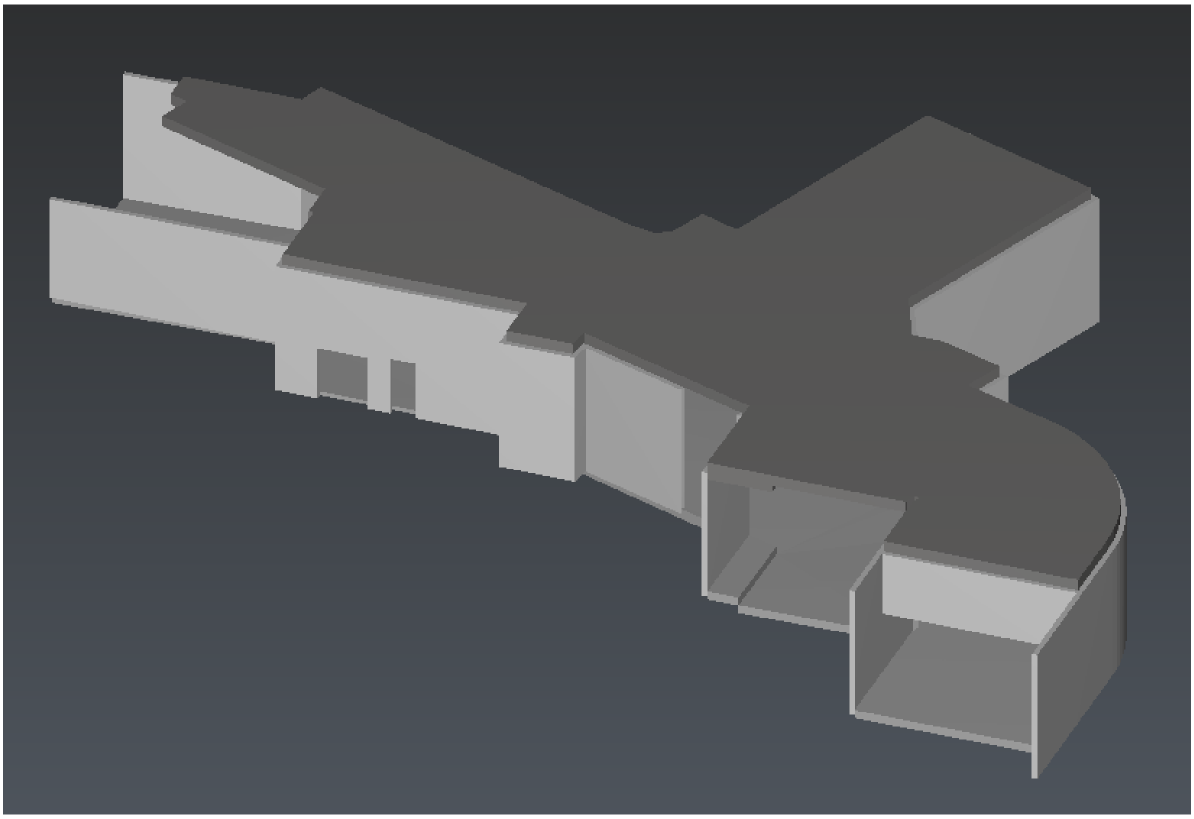
> Import **site.ifc** trough File > Import.

Unfold **GROUND**, then load only **Wall**, **Slab** and **Covering**. This selection filters which parts are going to be used among all parts available from an IFC or RVT import.

Validate with **OK** and hide it.

Ensure the output object is named **"site"**.

14.3.2



157 Imported model (site)

14.3.3 Create the BIM objects relevant for inspection

Copy-paste the object named "**site"** and rename it **"columns"**. Now, double-click on this copy to edit it and modify the selection.

Unfold **GROUND,** then right-click on the **Column** folder and select **Load only selection.**

Validate with **OK**. The object remains hidden, but its content has been updated.

Repeat the previous step with loading only **Flow distribution** and **Door** groups**.** Name this object **"other"**.

> **Tip & Tricks**
>
> It is also possible to filter the selection by keywords (component name, IFC type, etc.).

14.3.4 Validate the preparation file

Save this file as **BIMInspectionPreparationfile.3dr**, close it.

Once the preparation file is done, you can use it for all on-field analyses through the BIM Inspection workflow in Touch Mode.

14.4 Inspect the scanned data with the BIM object

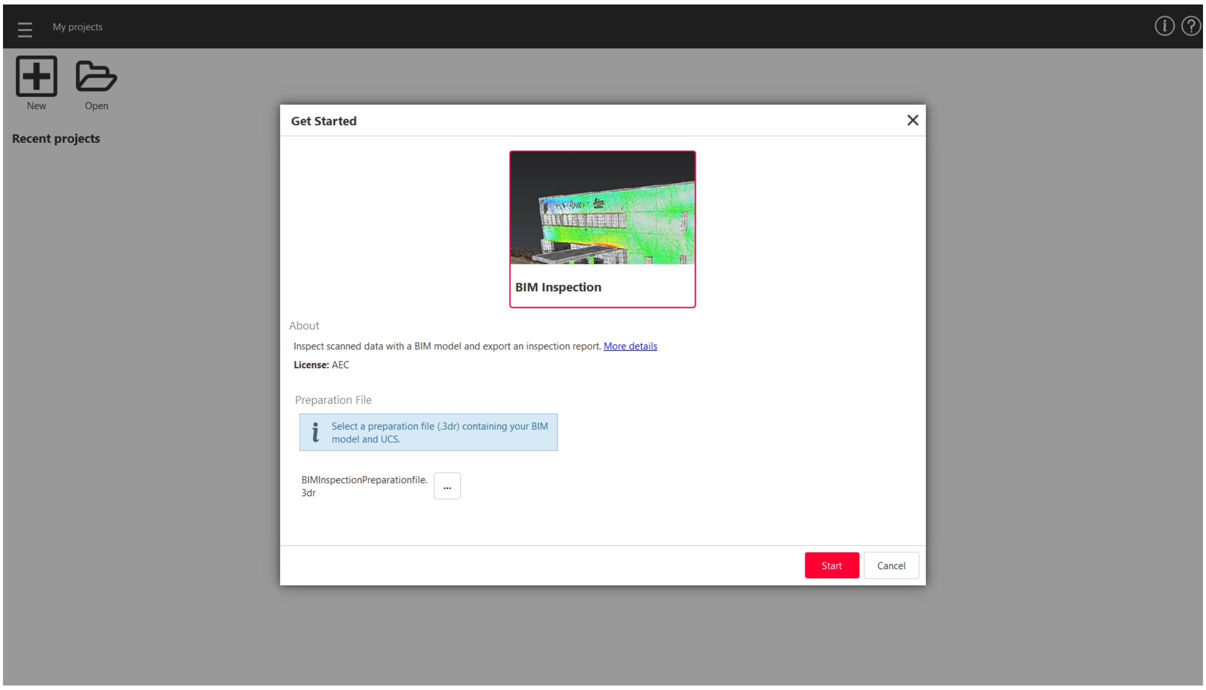
> Start the software in **Touch Mode**.

⚠ Ideally, you would follow those steps using a touch device.

14.4.1 Start a new workflow (LOAD)

Start a new BIM Inspection workflow.

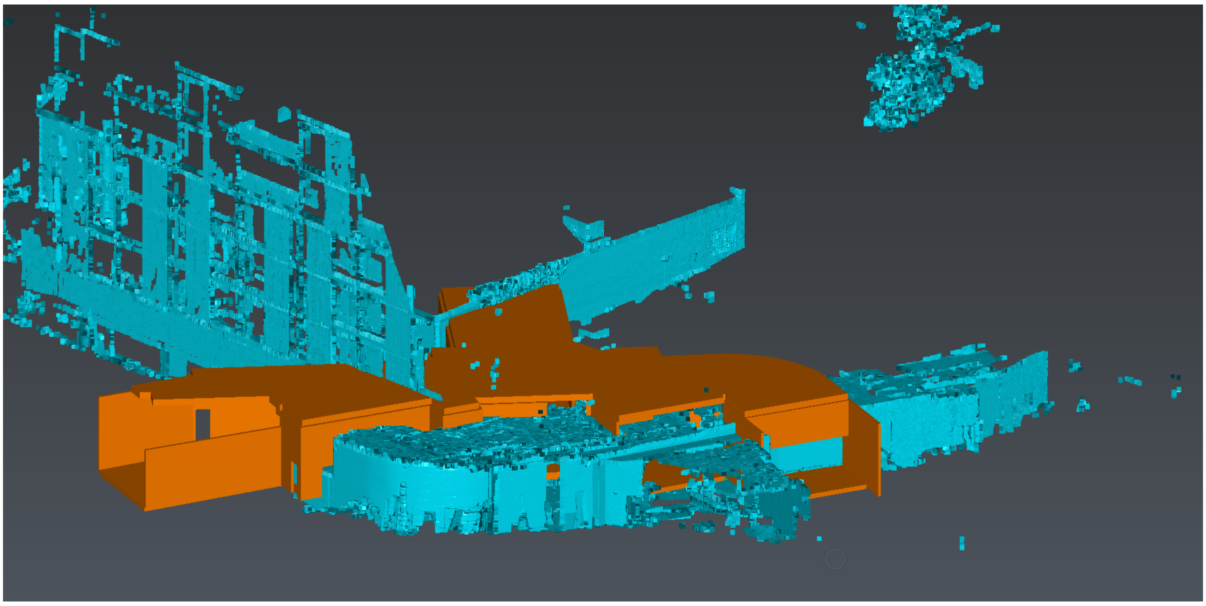
✔ Select **BIMInspectionPreparationfile.3dr** as the preparation file



158 Get Started
Follow the workflow instructions and load the **site** object, corresponding to site BIM model and import the scan from the file **site.e57** then click on **Next.**

Align model and scanned data (PREPARE)

Both objects are not aligned together. For instance, choose to move the cloud toward the BIM model (select **Scan to BIM Model** at the startup of the alignment step).

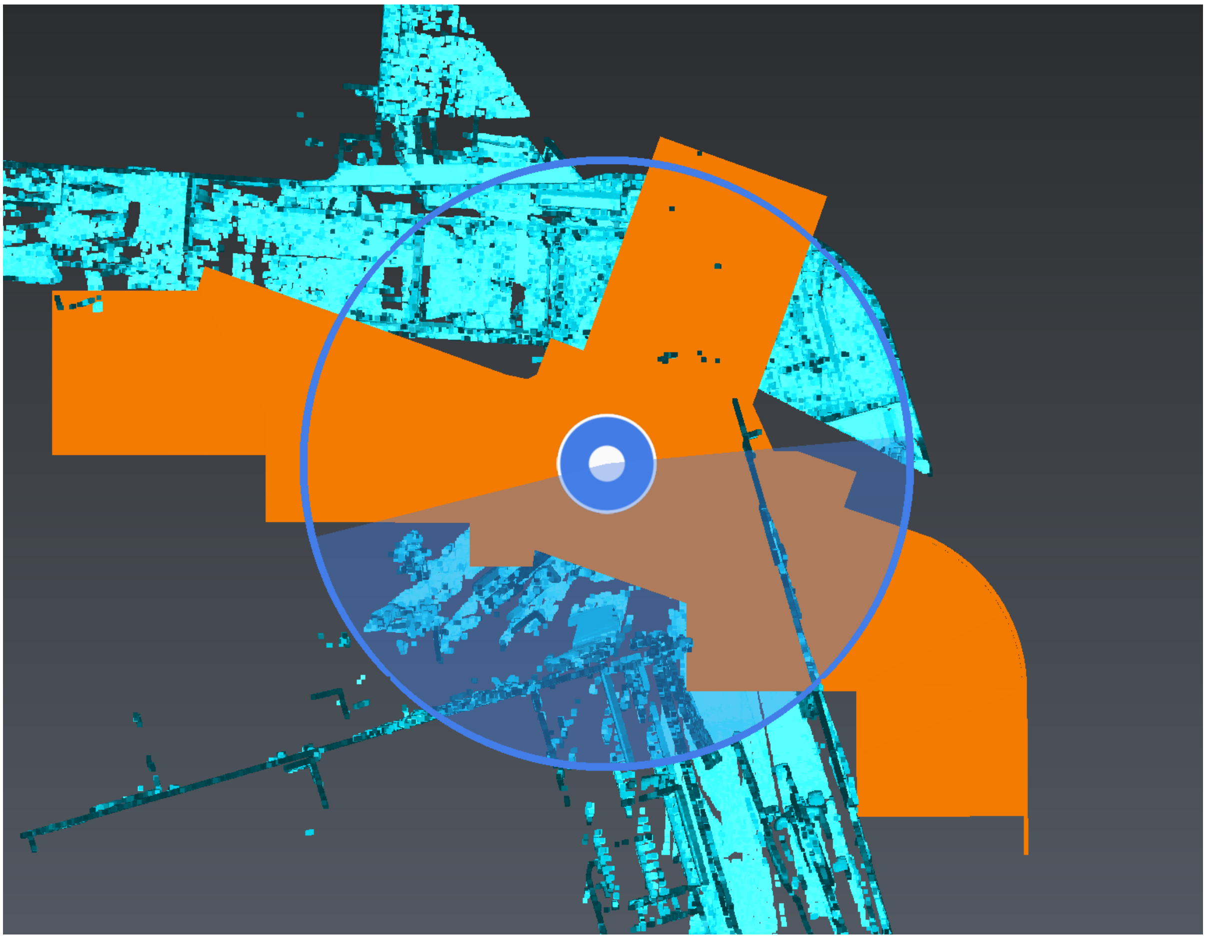


159 Both objects are not well aligned

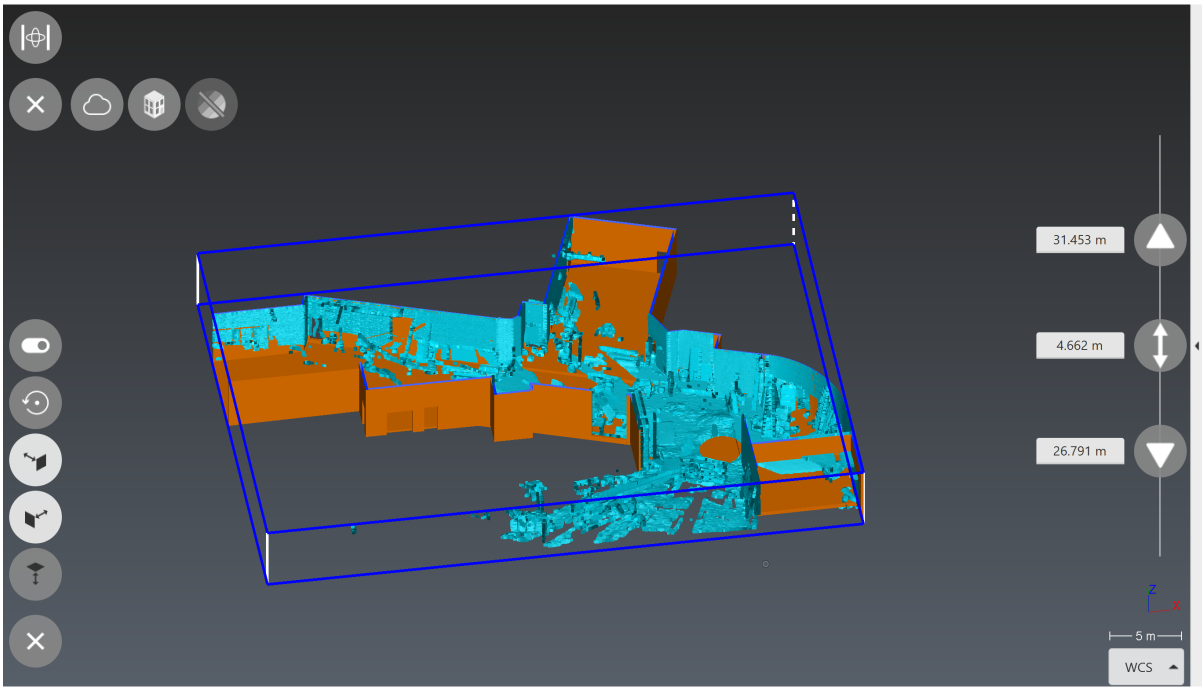
Then, you can align manually the cloud more accurately.

1. Start using the **Top Ortho** view: either translate or rotate the cloud in 2D.
2. Activate a **limit box** centered around the BIM to ease camera movements:
    a. Hide the scan
    b. Display the **Quick Access** toolbar and activate the limit box, then close the limit box edition
    c. Show again the scan
3. Switch to **Front Ortho** view: either translate or rotate the cloud in 2D.
4. Switch to **Orbit Ortho**: check that both objects are well pre-aligned. If not, repeat steps 1 & 3.

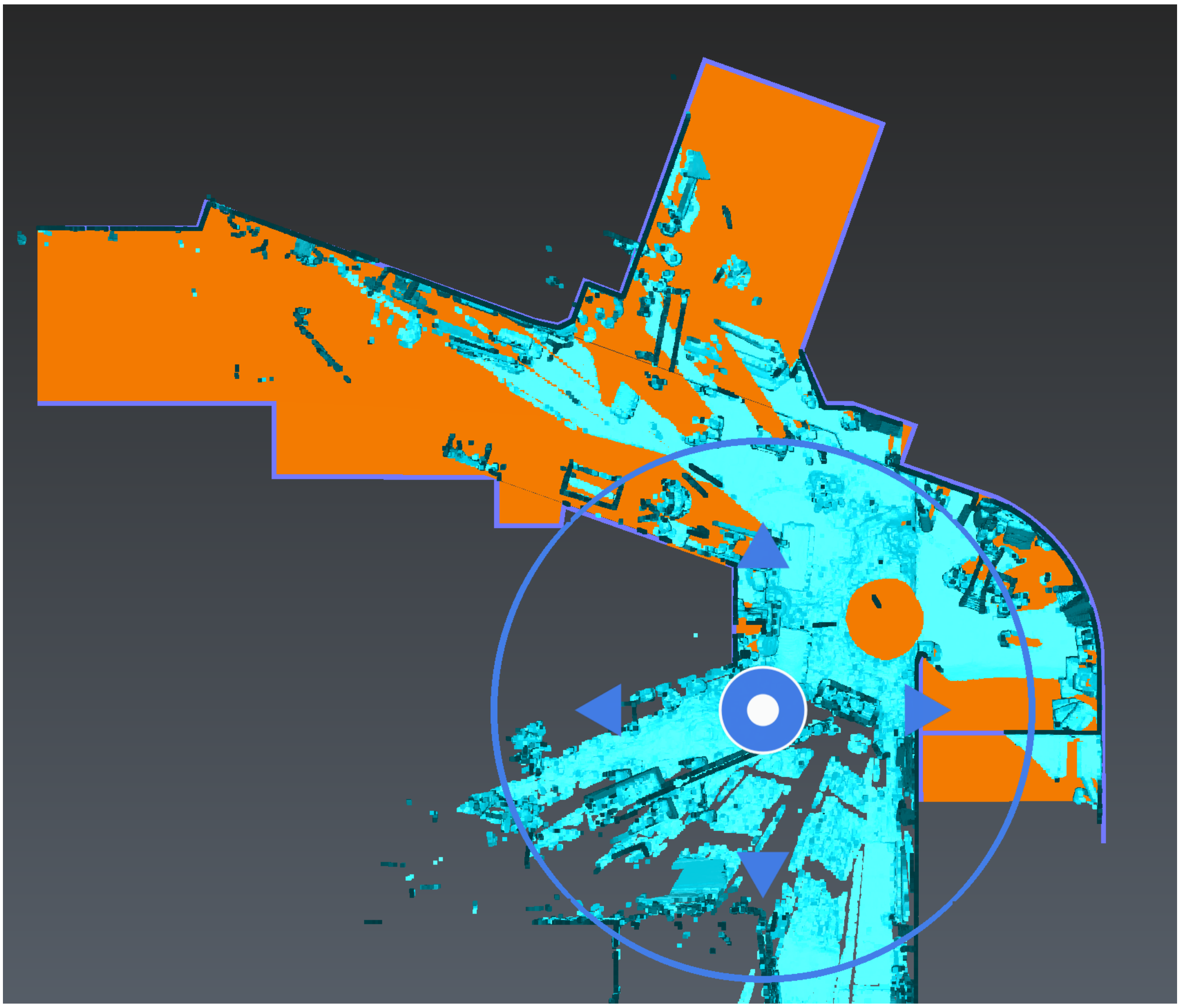
> ⚠ If the manipulator is not centered enough, you can relocate it.
> The manipulator is displayed depending on zoom factor.



160 Move the cloud manually (a)
Tap on the screen to display the quick access toolbar, then **tap** the limit box icon to create and adjust it. Limit boxes are initialized thanks to displayed objects bounding box and UCS orientation.



161 Adjusting the limit box



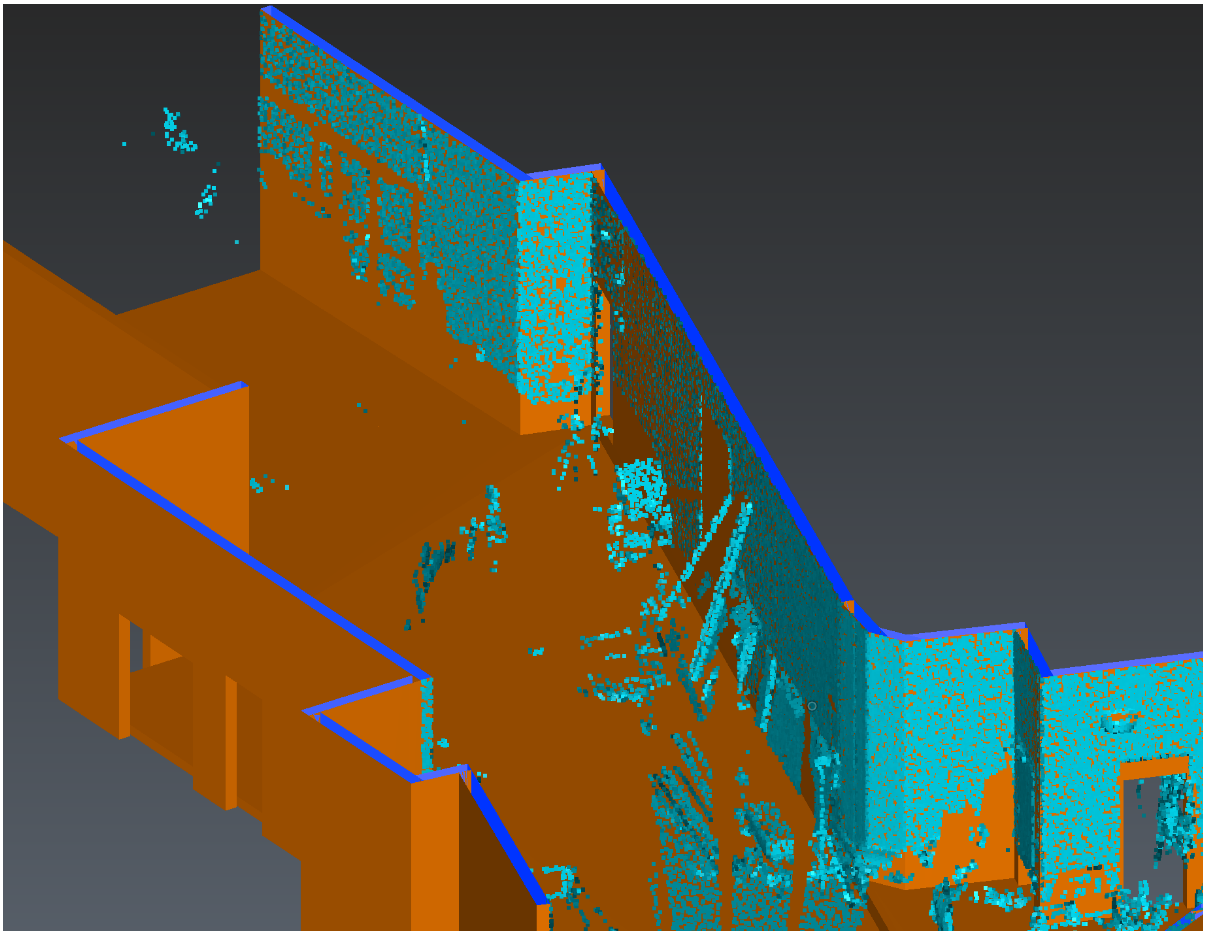
162 Move the cloud manually (b)

Set the best-fit settings:

- **Preserve Z axis orientation**: checked.
- **Best Fit Attraction**: 5 cm. Both objects have to be closer than this threshold before computing the best fit.

Finally, perform the best fit, check the mean value error and validate this step.

The **Best Fit tolerance** can be set through specific settings (BIM Inspect Specific Settings, **BIM inspection** tab).



163 Both objects are now well aligned

14.4.2 Clean the point cloud (PREPARE)

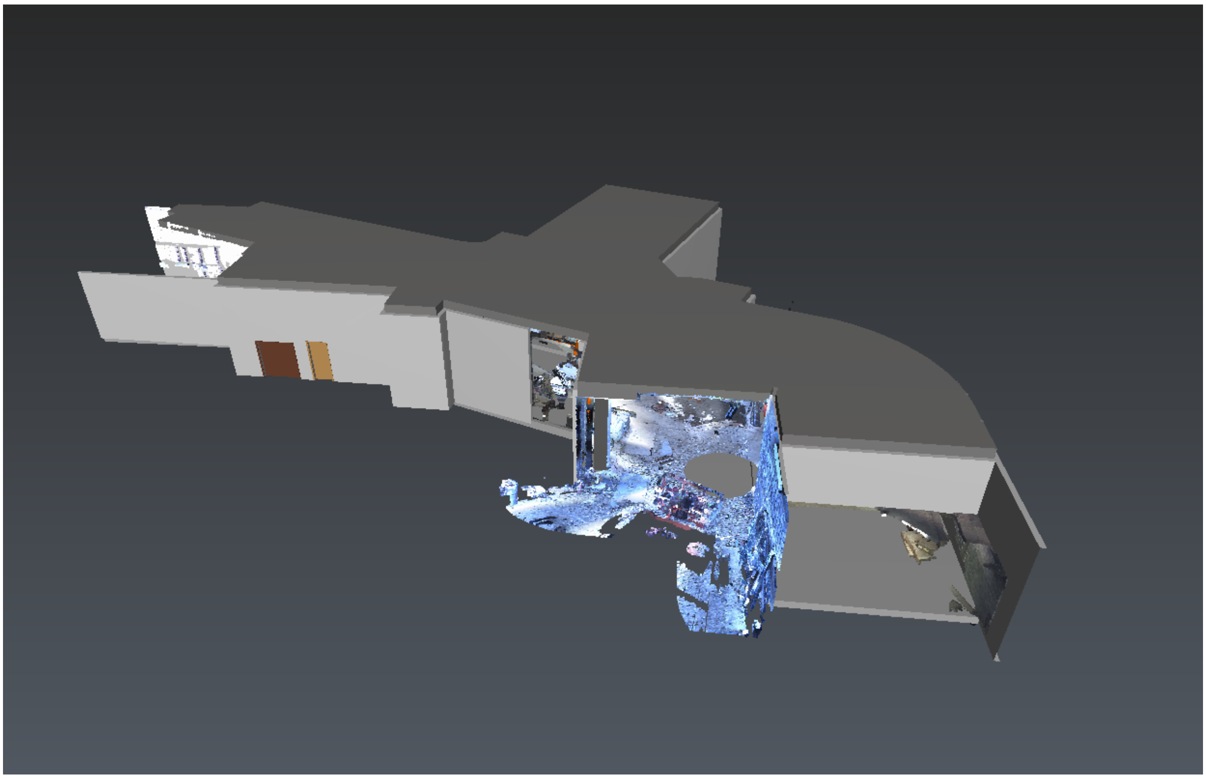
Usually and especially with terrestrial laser scanners measuring full domes, you will acquire areas you are not focusing on. In order to deal with tablet specifications, which could be very low compare to your everyday workstation, it is recommended to remove these areas.

> Note that another setting, which limits the amount of imported points, can be managed thanks to BIM Inspection settings.

You can remove points:

- far from the selected BIM model
- clipped by the active limit box

Here, remove clipped points farther than 3 m.



164 Cleaned scan

14.4.3 INSPECT

In this exercise we will focus on inspecting **columns**.

Select the parts to inspect, that is to say the **columns**, then compute the inspection excluding points farther than 0.2 m and keep the 3D deviation option.

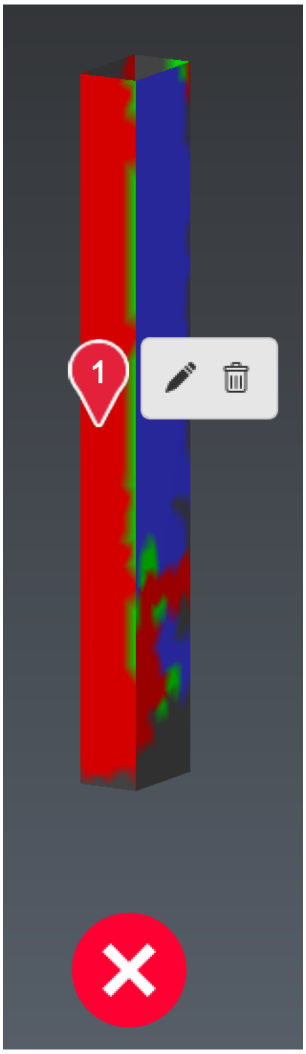
Finally, customize the color mapping as you wish (for instance: tolerance +/- 0.02 m and origin 0).

165 Inspection

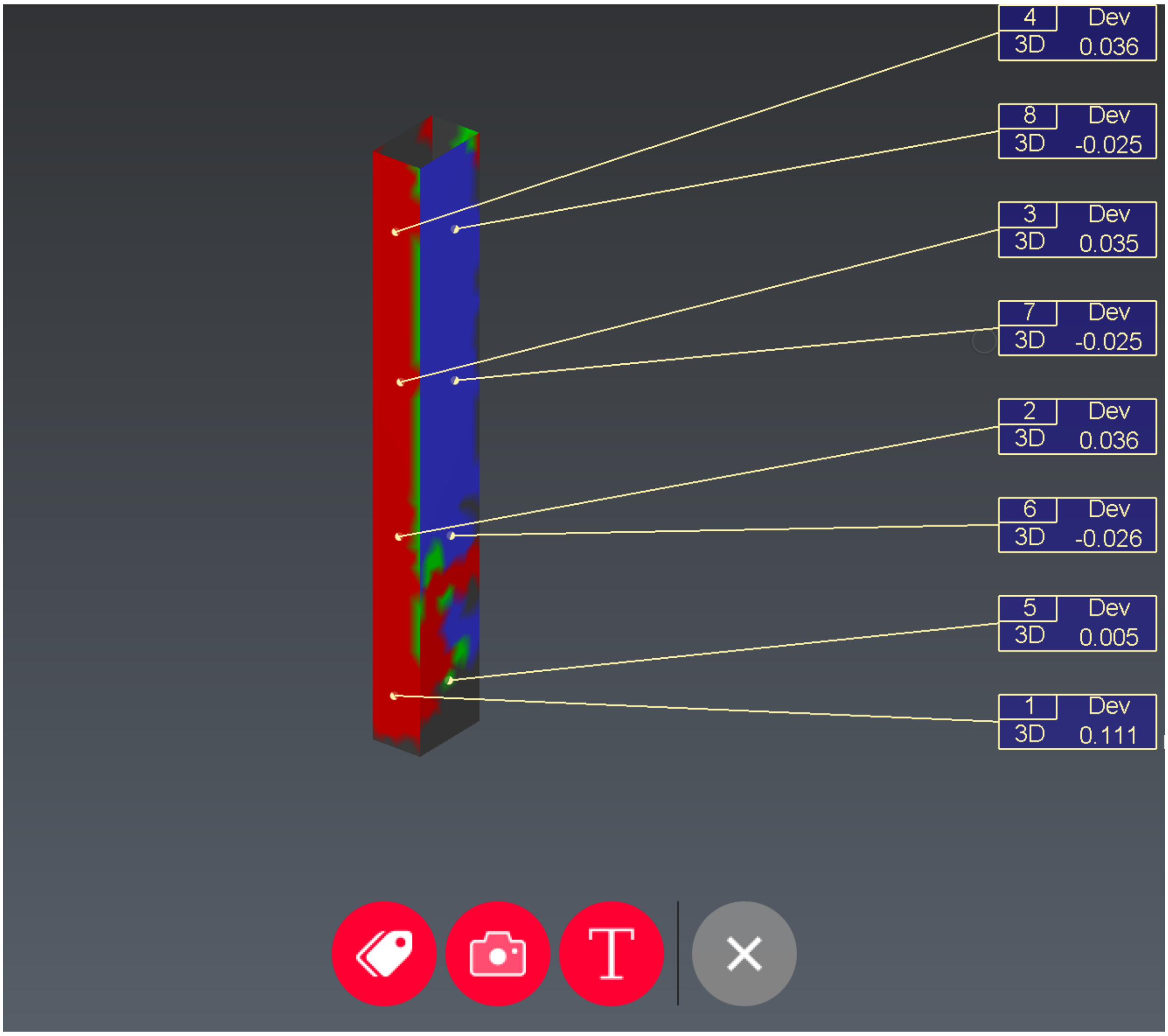
14.4.4 DELIVER

In order to provide a report, add notes for each identified object with defection.

In this sample, only one column has issues.



166 Add a note
Each note can gather inspection views with deviation labels, photos and comments.



167 Edit a note

Finally, generate a .pdf report and .csv deviation tables.