



HEXAGON

3DRESHAPER®

The 3D scanner software

3DReshaper Version 2016MR1 – Beginner's Guide



LEGAL NOTICE

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INTRODUCTION

Technodigit is pleased to introduce you to **3DReshaper**, the 3D Scanner Software.

Your Beginner's Guide...

This Beginner's guide will walk you through some typical process using 3DReshaper 2016 MR1. All practices used in this guide are present in the public document directory, by default:

C:\Users\Public\Documents\3DReshaper 2016MR1 for the 32 bits version

C:\Users\Public\Documents\3DReshaper 2016MR1 (x64) for the 64 bits version

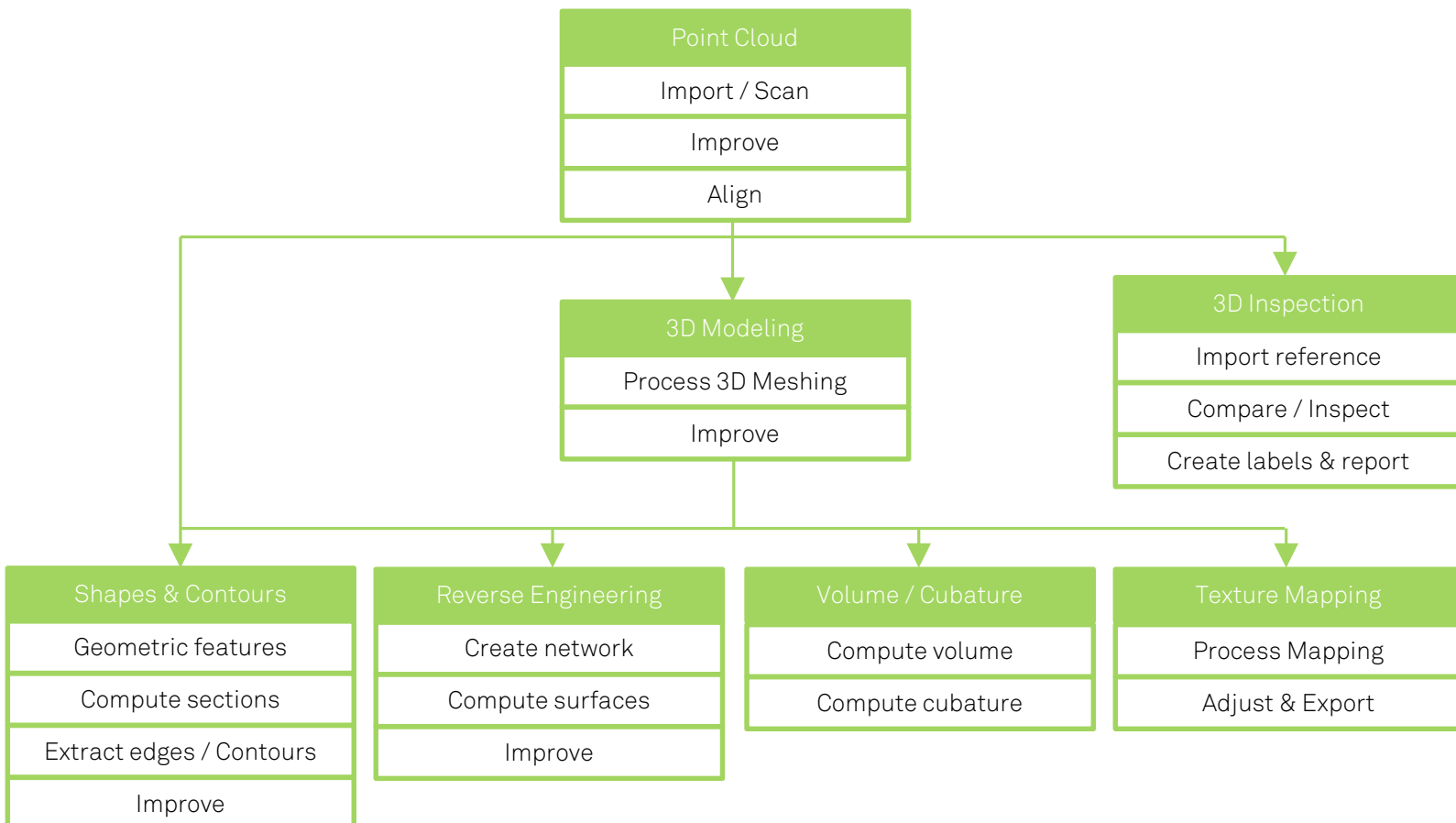
A shortcut to this folder (/Samples) is also present in *My Documents* folder and you can also access to the practices through the Windows start menu, in the 3DReshaper folder.

To go further...

Some practical exercises are available on our [website](#), and feel free to refer to the 3DReshaper help files (press F1 in 3DReshaper) to find specific information about all the features. You can also contact your local reseller in order to have a complete training.

3DReshaper typical workflows

Here are the typical 3DReshaper workflows:





3DRESHAPER BASICS

In this section, we will learn the general knowledge about 3DReshaper: how to customize the software, how to handle objects, how to click a point, etc.

The User Interface

General

3DReshaper interface is divided into three parts:

- The **ribbon** on the top with the main button, menus and some shortcuts.
- The **tree explorer** on the left with all the objects contained in the current project (clouds, meshes, polylines ...).
- The **3D scene** where the objects are displayed.

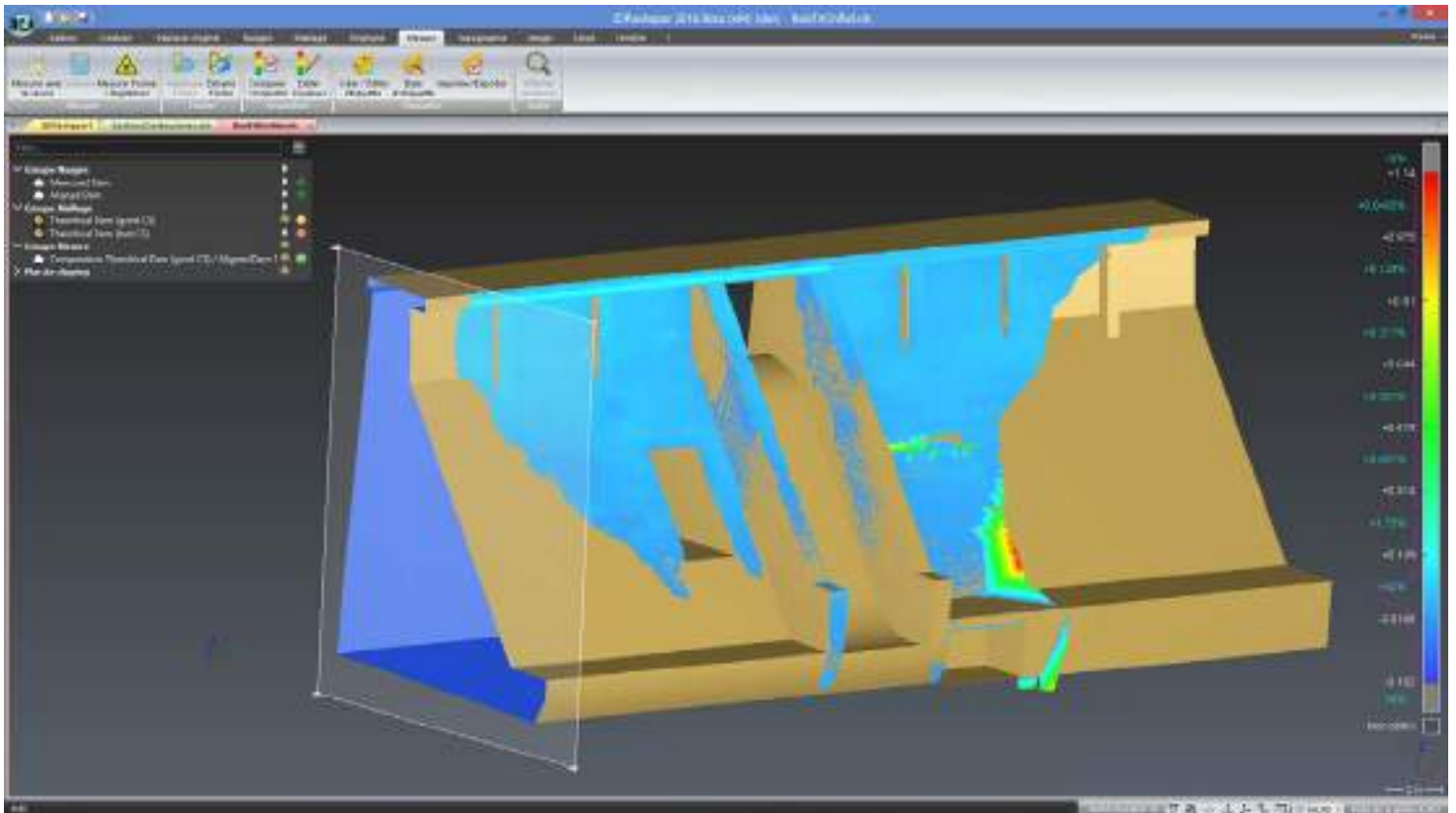


Figure 1: The 3DReshaper Interface

The **selection** of one or more objects can be performed either in the tree or in the 3D scene using standard Windows keys (Shift + Mouse Click - CTRL + Mouse click). In the scene, you can also define a selection rectangle with the left mouse button down, while moving the mouse (see section *General Instructions / Selecting objects* in the *help* file).

A 3DReshaper project is an .RSH file. This file contains all the objects: point clouds, meshes, polylines, geometric features, surfaces, etc.

An .RSH file contains also a screenshot in order to see what there is in the file in the preview pane of Windows Explorer.

Open / Import files

To open or import files you can:

- Use the command [Main button\Open](#) to open .RSH files.
- Use the command [Main button\Import](#) to import all compatible formats (.STL / .IGES / .STEP / .OBJ / .PTS ...).
- Drag & Drop files directly in 3DReshaper.
- Double-click the file in the Windows Explorer for the files associated with 3DReshaper during the installation (option at the end of the installation process).

Note: Click [here](#) to consult the complete list of supported formats.

Units

In 3DReshaper, by default, the measurement unit is not specified. In fact, **the measurement unit used by 3DReshaper is the unit of the imported data file** which is not known by 3DReshaper, or **the unit of the scanner if you are scanning points**.

Thus, if you read a point file or a STL file in meter, then, the “current” unit of all the values expected by the diverse dialog boxes will also be the meter:

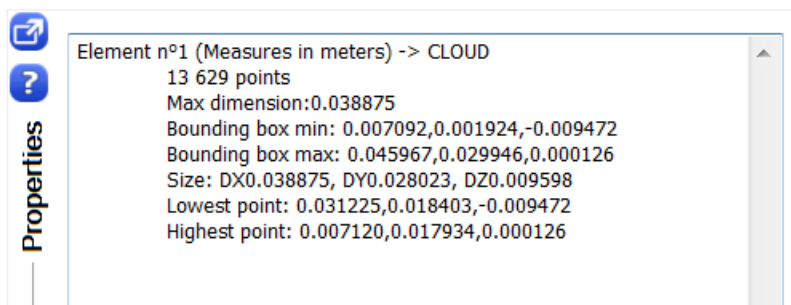
- Deviation error
- Hole size
- Triangle length
- Accuracy to merge meshes
- Etc.

If you import files with different units, you can use the command [Move-Align\Scale](#) to unify them.

Activating the units is possible by using the command [Edit \ License and Settings](#).

Exercise: Convert a cloud from meters to millimeters

Open the file “CompareInspect.rsh”. This file contains two clouds, one in meters and one in millimeters, and a CAD reference in millimeters. For example, select both clouds, do a right click and select “Properties”. Have a look at the size of each cloud: one is 1000 times bigger than the other.



To do the conversion, select the cloud in meters and then go to [Move-Align\Scale](#) and choose the Scale tab (if not already selected). Define the center (0, 0, 0) – see *Entering or clicking points* - and then select the option “Same scale for X, Y and Z” and enter “1000” (converting meters to millimeters requires a multiplication by 1000). Press the “OK” button.

Note: In the case you have chosen to activate the units in 3DReshaper, an additional tab is available which allows you to directly scale from one unit to another (mm to m, inches to m...).

Have a look again at the properties; the size is now exactly the same.

Undo

3DReshaper is designed according to the Microsoft Windows standard, in particular:

- You can cancel the last actions by making [Edit\Undo](#) or Ctrl-Z. You can make undo until the beginning of your work session.
- You can restore the last actions by making [Edit\Redo](#) or Ctrl-Y.

The main problem with traditional applications using the undo is that you cannot restore an object deleted 10 commands ago without making an undo of the 9 previous commands especially if you need to keep them! To get over this problem, 3DReshaper gives you an access to the undo stack, which is named “Recycle bin” and where deleted objects are stored.

When you modify or delete an object, the original is stored in the Recycle bin. If you make “Undo”, this original is restored in your work session.

At any time, you can decide to restore an object from the recycle bin. For that, you just need to:

- Develop the recycle bin in the tree explorer.
- Select the component(s) to restore (the clicked object temporarily appears in the graphic zone while you press the left button so that you can see it).
- Click with the right button to open the contextual menu.
- Choose the “Restore” command.

If you handle very big objects, the memory of your computer could be saturated if you store too many objects in the recycle bin. You can then decide to delete the objects from the trash:

- Develop the recycle bin in the tree explorer.
- Select the component(s) to delete.
- Press the key “Del” or use the contextual menu “Delete” command or go to [Edit\Delete](#).
- A warning message appears to recall you that no restore or undo operation will be possible.
- Click “OK” to permanently delete the selection and corresponding undo operations.

You can also delete all the objects of the recycle bin using the command [Edit\Empty recycle bin](#).

Finally, if your computer does not have enough memory, you can also disable the full undo feature of 3DReshaper with the command [Edit\Full Undo](#). In this case, the recycle bin is cleared every time you enter a new command. This provides an undo level of one action only.

The 3D Scene

Move

In the graphic window, it is possible to modify the angle of view. You can:

- Do a **Zoom** with the mouse wheel or by pressing the right button and the shift key while moving the mouse.
- Do a **Zoom All**; means adapt the zoom in order to see all the displayed objects, by pressing “A” on the keyboard or [Window\Zoom all](#).
- Do a **Zoom Window**; means adapt the zoom to a drawn rectangle, with the command [Window\Zoom Window](#).
- Do a **camera rotation** by pressing the right button and moving the mouse (maintaining the mouse button pressed); press ALT-right button in order to slow down the rotation (the center of the rotation is the clicked point ; If you click the first point on the border of the screen, the rotation axis will be the screen normal).

- Do a **camera translation** by pressing simultaneously right and left button and move the mouse. If your mouse has 3 buttons, you can also use the middle button.
- Press X, Y, Z, SHIFT-X, SHIFT-Y, SHIFT-Z on the keyboard to respectively set the view along the X, Y, Z, -X, -Y, or -Z axis.

Select

The left mouse button is used to select objects inside the 3D scene:

- The left click onto an object, in the graphic zone or the object explorer, selects the object.
- If other objects were previously selected, they will be automatically deselected and the previous selection will be replaced by the new one.
- You can also select many objects together using a rectangle: if you click and move the mouse before releasing the button the software will draw a selection rectangle:
 - If you move the mouse to the right, you will select the objects **strictly** inside the rectangle
 - If you move the mouse to the left, you will select the objects **cutting or inside** the rectangle
- If you press the “Ctrl” key of your keyboard, the selected elements will be added into your selection. Inside the object explorer, it is possible to select or deselect objects that are not visible in the graphic zone.
- If you click on an object of the object explorer, this object is temporarily highlighted while the button is pressed, even if the object is in the trash.
- You can use the “Shift” key in the object explorer to select objects “from / to” in the tree explorer.
- Press “ESC” in order to unselect all the objects.
- You can also press “CTRL-A” to select all visible objects.

Note: It is possible to select and work with objects even if they are not displayed.

Note: The command [Edit\Selection filter](#) adds some parameters to the selection with a rectangle. For example, it allows you to select only visible objects (and not objects hidden behind another one) or to remove some objects from the current selection. This option is mainly used to handle CAD files made of many patches incorrectly oriented after import.

View





Orthographic / Perspective

Two view types are available in 3DReshaper: Orthographic view and Perspective view. You can identify that a view is orthographic or perspective by looking at the bottom right corner of the 3D scene.

The scale is only drawn in orthographic as the pixel distance is the same for every depth and area on the screen in this mode. In perspective, this is not true: close objects will appear bigger than far objects.

The other important difference between Orthographic and Perspective view is that in perspective, the camera is not defined at an infinite distance to the object but is located within the 3D environment. With this rendering, it is therefore possible to enter inside a tunnel or any closed object.

An additional menu, located at the bottom right of the screen also helps changing the view:

- The icon  switches from a perspective to an orthographic view.
- The icon  allows going through the surface located behind the mouse or to zoom on this surface but never go through. This command is not activated in orthographic mode.
- The icon  enables to use a reference plane (usually the Z plane). When activated, the vertical axis of the screen will always be perpendicular to the reference plane.
- The icon  defines the current plane as the reference plane.

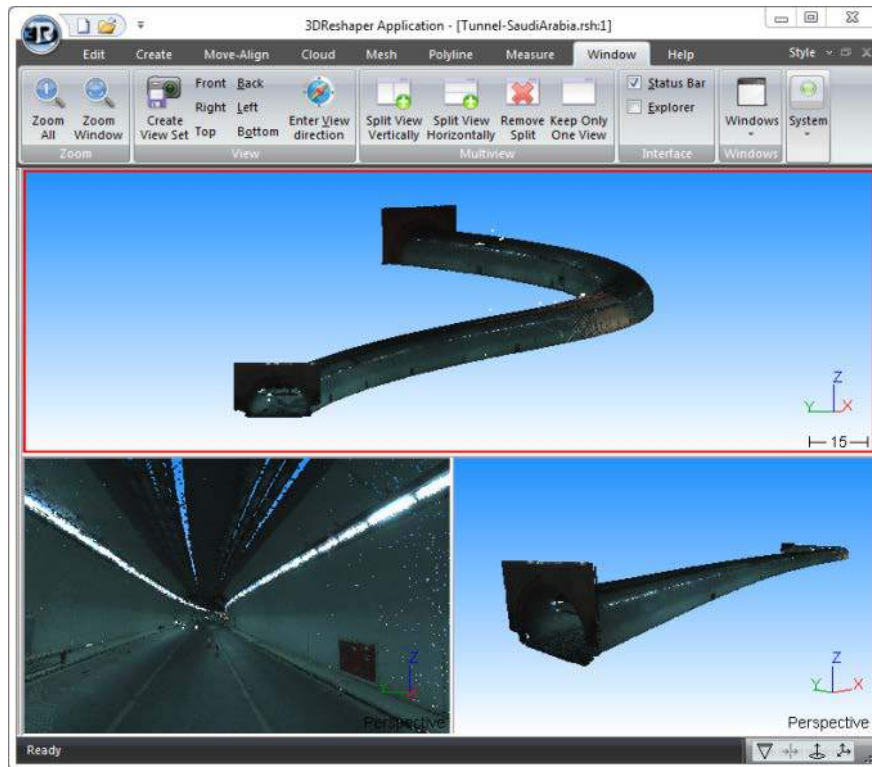


Figure 2: Difference between perspective and orthographic view

Show / Hide objects

To show or hide objects in the 3D scene, you just have to select them (in the scene or in the explorer tree), then do a right click and select:

- “Hide” to remove the selection from the 3D scene
- “Show” to display the selection in the 3D scene
- “Show only” to make everything invisible and then show the selection
- “Zoom on” to make a progressive zoom in order to see where the object is located

Multiview

In 3DReshaper you can create up to 8 views with the commands [Window\Split view vertically](#) and [Window\Split view horizontally](#). These commands will split the current view into two parts. The current view is the one with the highlighted border (the last one where you have clicked). To delete the current view, use the command [Window\Remove Split](#). To keep only the current view, use the command [Window\Keep only one view](#).

When you have several views, you have more visibility options:

- “Show in all views” to make the selection visible in all the views
- “Show only in all views” to make everything invisible in all the views and then make the selection visible in all the views
- “Hide in all views” to make the selection invisible in all the views



Figure 3: Multiview sample - 3 views along each main axis

Customize 3DReshaper

Customize Interface

Language

You can change the software language with the command [Edit\Language](#). You will have to restart 3DReshaper to apply the setting.

Shortcuts

You can create keyboard or icon shortcuts in order to access more rapidly to your favorite commands. Click the small arrow on the top of the window (next to the main button) and then select “More commands...” (see Figure 4).



Figure 4: Button to add icons or keyboard shortcuts

General Interface

In the command [Edit\Colors & Aspect](#) you can change:

- The background (color, graduation, texture)
- The selection color (color of objects when they are selected)
- The character height (for all texts written in the 3D scene)
- The default behavior for dialog boxes
- ...

[Go back to default settings](#)

If you want to restore all the default settings, just go to [Edit\Clear settings](#).

Representation, colors and aspect of objects

Several representations are available for each object. Select an object, then do a right click and go to “Representation”. Several options are available:

- Smooth: to display a shading on a cloud or triangles with smoothed colors on a mesh
- Flat: display a shading on a cloud or triangles with uniform colors on a mesh
- Wire: no shading on a cloud, only triangles contours on a mesh
- Smooth + Wire: on meshes, triangles contours are drawn over the smooth representation
- Flat + Wire: on meshes, triangles contours are drawn over the flat representation
- Smooth + Rotation Wire: smooth display by default and wire display during rotation with the mouse
- Flat + Rotation Wire: flat display by default and wire display during rotation with the mouse
- Texture: to see real colors from a point cloud or textures applied on a mesh
- Inspection: to see colors coming from inspection, intensity, curvature analysis, etc.

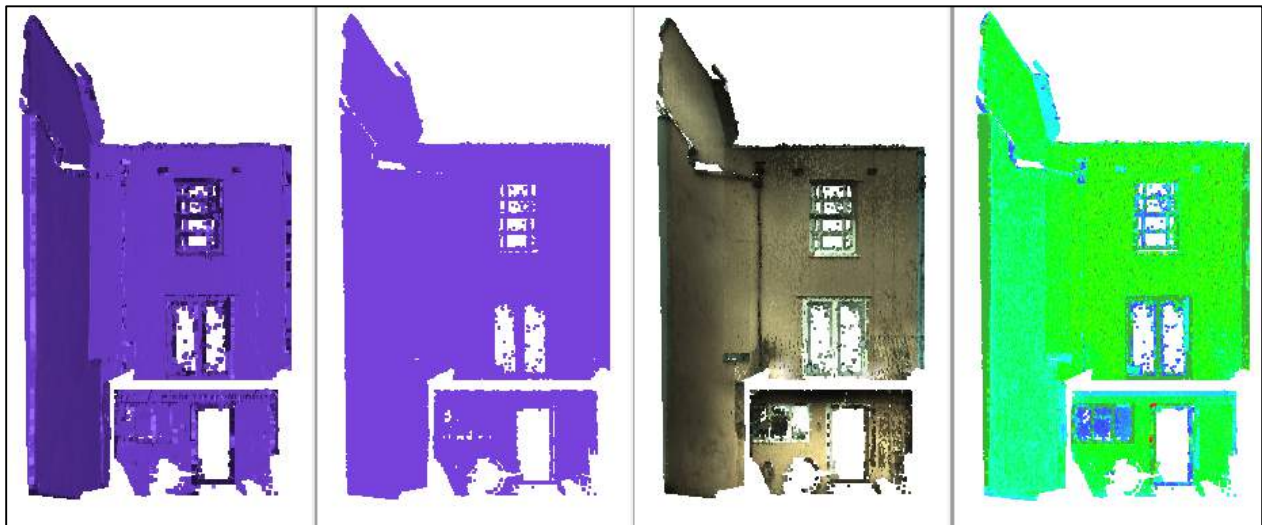


Figure 5: Representation mode on a point cloud (from left to right: smooth, wire, texture, inspection)



Figure 6: Representation mode on a mesh (from left to right: smooth, flat, wire, texture, inspection)

To change the color, select your object, then do a right click and select “Color”.

To change the transparency, select your object, then do a right click and select “Transparency” (not available on clouds and polylines).

More parameters are available in the command [Edit\Colors and Aspect](#). This command allows you to change more display parameters for each type of object (clouds, meshes, polylines ...). You can change default parameters (for all objects you will create or import), or current parameters (for the selected objects).

Normal orientations in 3DReshaper

Generally speaking, the orientation of meshes or surfaces is very important (for example during inspection). In 3DReshaper, Meshes and Surfaces have a color for the “inside” and another one for the “outside”, so it is very easy to detect when the normal is not correctly oriented. The “outside” color is also shinier than the “inside” color.

To swap the orientation of an object, select it, do a right click and select “Reverse” or push the key “i”.

Exercise: Understand meshes orientation

Open the file “FillHoles.rsh”.

In 3DReshaper, the default color for meshes is gold. Select and show only the mesh “4-Closed Mesh”. Select it again, do a right click and then select “Properties”. Have a look at the volume, it is positive: everything is OK.

Now select the mesh, do a 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, do right click and go to the “Color” menu). The mesh will not take the selected color (it will be an opposite color).

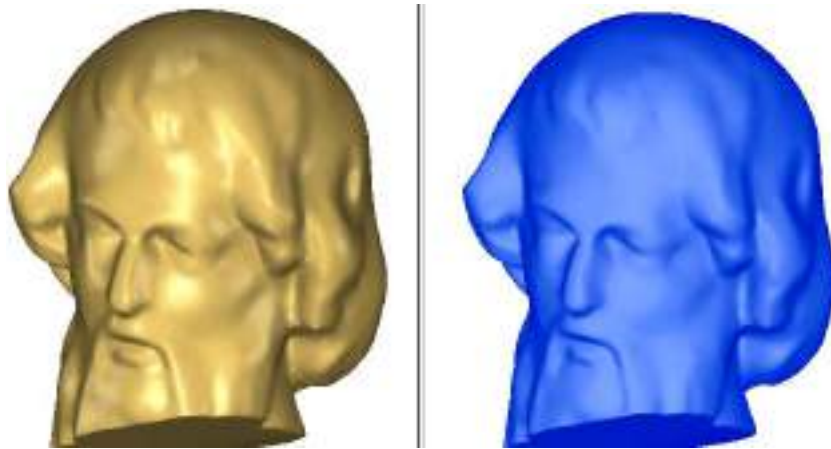


Figure 7: On the left the normals are OK; on the right the normals are inverted.

Entering or clicking points

In 3DReshaper, each time you will have to click a point, a ribbon like in the Figure 8 will appear.



Figure 8: The ribbon with options to enter a point

This ribbon will allow you to choose between 7 options to click a point. You can select several options at the same time, but some of them are not compatible (for example if you select “Point on Selection”, “Vertex / End” will be automatically unselected).

Exercise: Learn all the different options to click a point

Open the file “EnterPoints.rsh” in order to practice a little bit. Make sure all the objects from the tree explorer are visible and open the command [Create\Point](#).

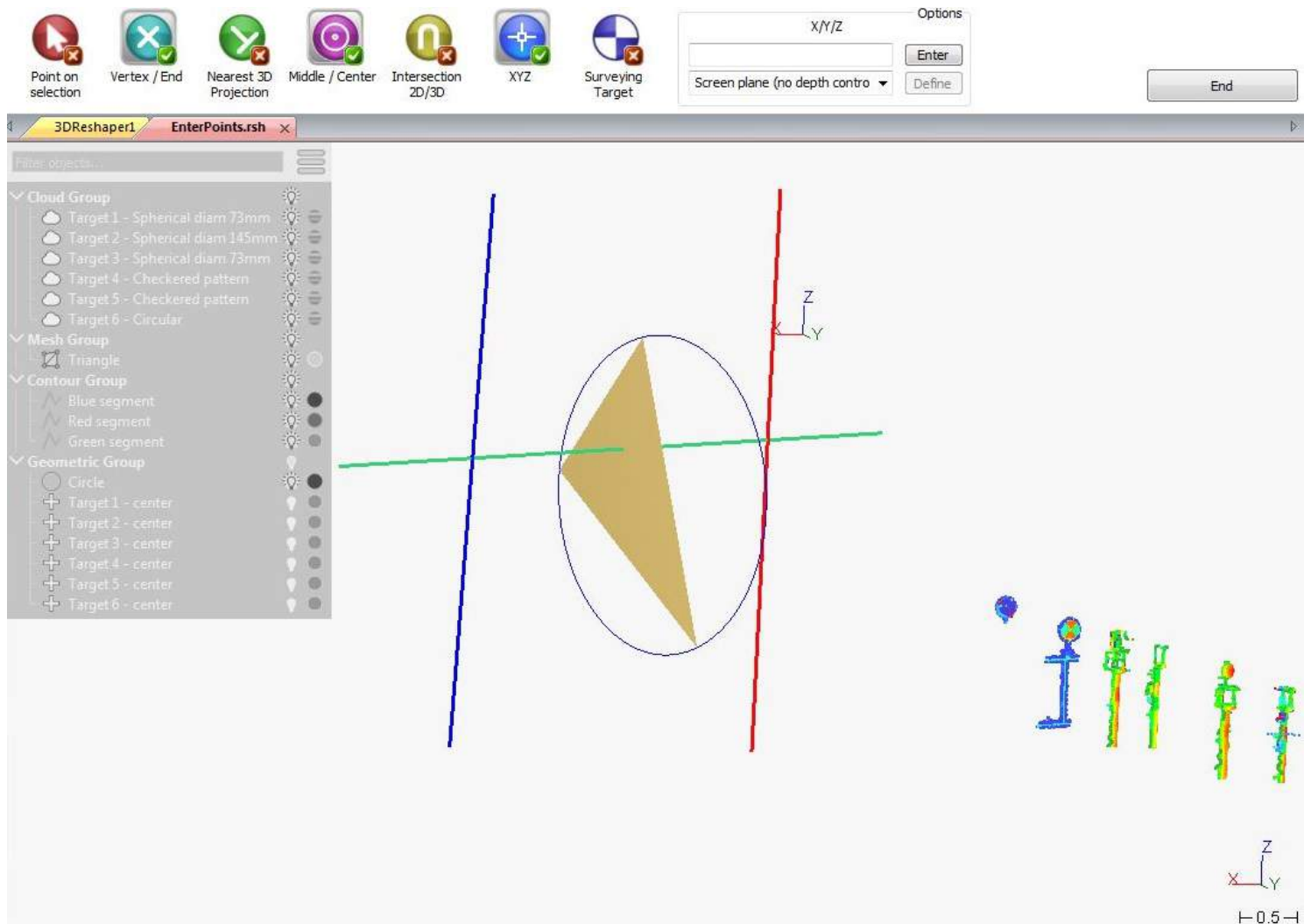


Figure 9: When 3DReshaper is waiting a point input, a preview is made in real time in order to see where the point will be clicked.

So when 3DReshaper is waiting a point, you have on the bottom of the 3D scene the coordinates of the point currently selected and an icon in order to know what kind of point it is. If the point is not just behind the cursor, there is also a small ball in order to locate the point. For example, in the Figure 9, the current point is (0.52, -3.06, -0.45) and is located on one vertex of the triangle.

Point on selection

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

Vertex / End

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

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 model.

Middle / Center

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

Intersection 2D/3D

This option means that the point clicked is the intersection between two lines (in 2D or 3D). The intersection will be computed according to the zoom factor. This means that if there is not a real intersection between two lines and if you make a zoom to focus on the intersection area to see the small gap between lines, the system will not accept to make the intersection. But 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 sounds like a bell to tell you that there is no real 3D intersection between the lines.

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

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

XYZ

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

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

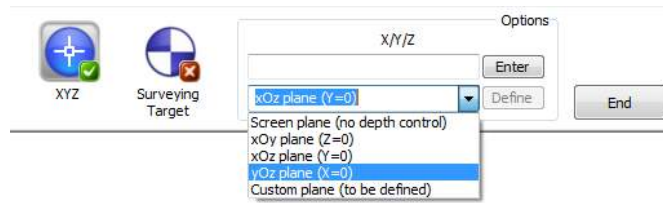


Figure 10: Options to define the plane where the points are created with the XYZ option

But you can also click points directly in the 3D scene, even if there is no object behind the mouse. By default, the point will be created in the screen plane, but you can define the plane where points are created (Figure 10):

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

Surveying target

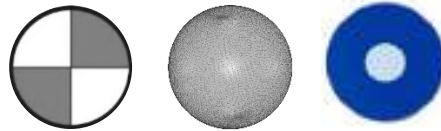


Figure 11: Target compatible with 3DReshaper

If your point cloud contains some surveying targets (black and white or spherical or blue and white) like one on the Figure 11, you can click the center directly in 3DReshaper. For example, select the option “Surveying target” and “With checkered pattern”, enter 0.15 as the diameter target and then click 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).

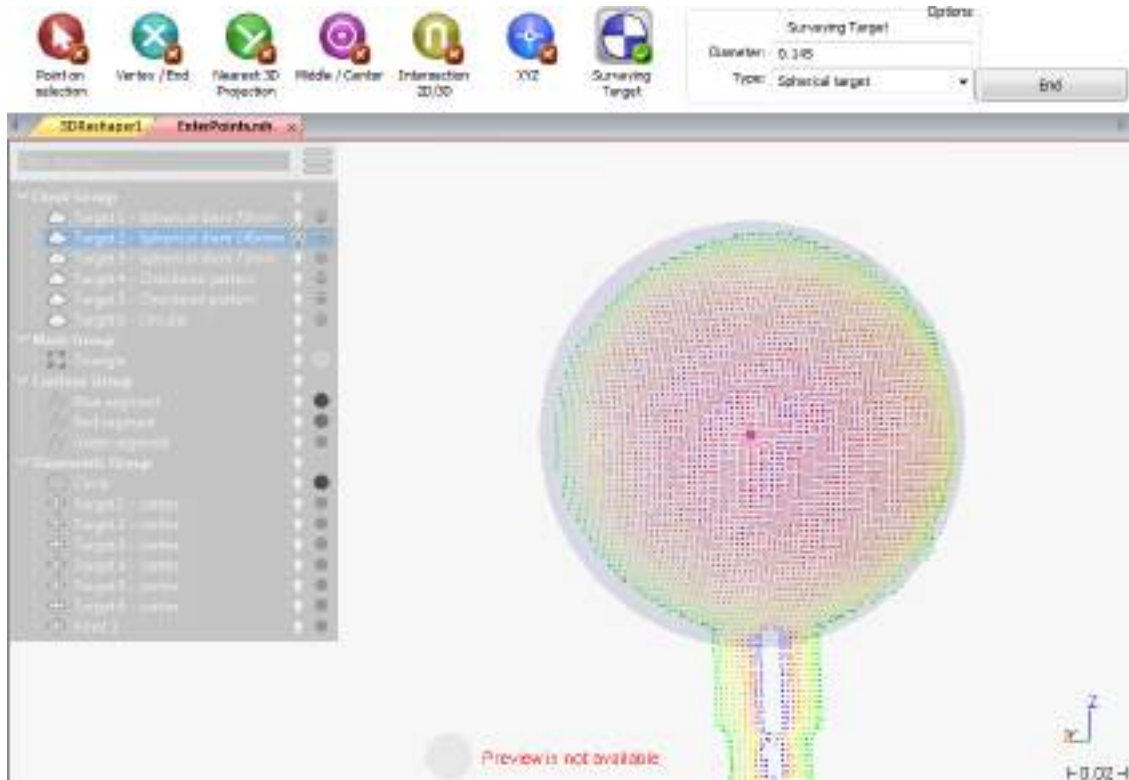
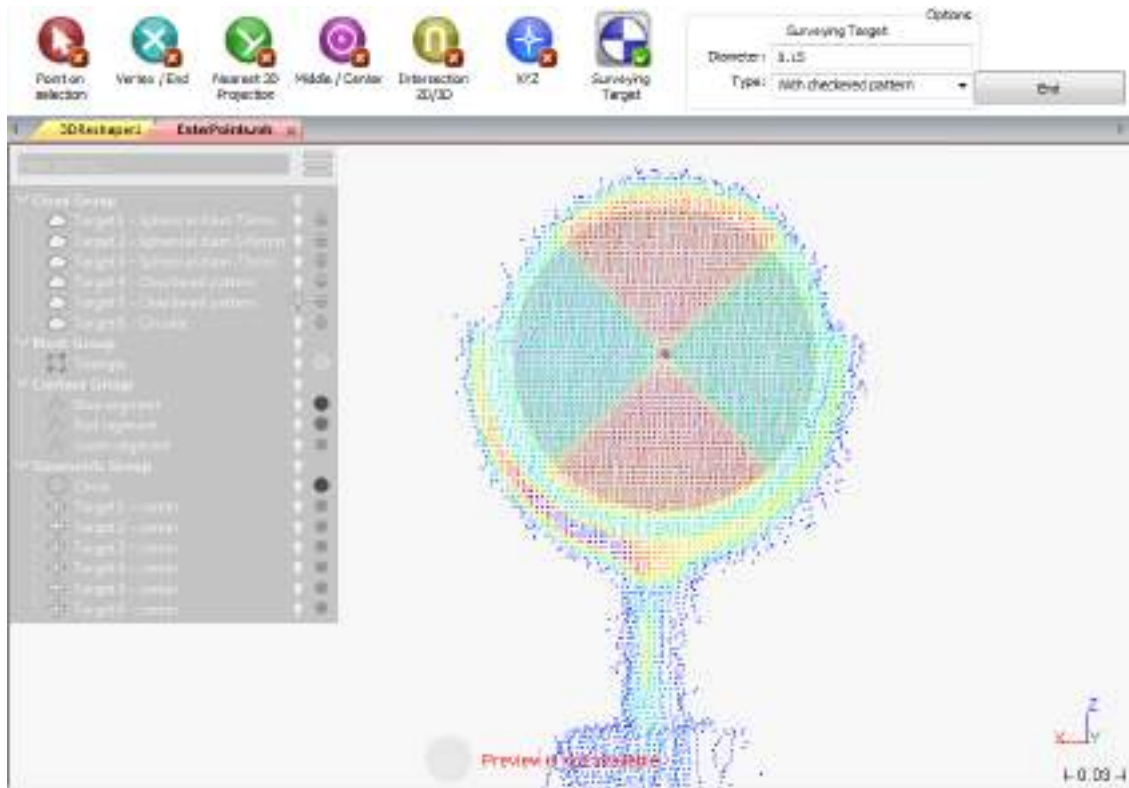


Figure 12: With the "Surveying target" option, you can click automatically and precisely the target center (with checked pattern or spherical target)

POINT CLOUD PROCESSING

In this section, you will see how to import or create point clouds in 3DReshaper, and you will learn how to work on a cloud to improve it before using it for further processing.

Import point clouds

With 3DReshaper, you can import one or several point clouds at a time with the [Main button \Import cloud\(s\)](#). 3DReshaper creates one cloud per file imported. Each cloud has a different color and is named according to the corresponding file.

3DReshaper offers the possibility to reduce the number of points in a cloud before importing it, by giving the maximal number of points allowed for each cloud. You could need to use this option to import very big point clouds, depending on your computer performance. In this case, points will be removed in areas where there is a high density of points in order to preserve the details. To give you a rough idea, a 64 bit mode computer with a 2 GHz Dual Quad Core i7 or higher (i5 minimum) processor, a NVidia – GeForce 1GHz minimum graphic card enables you to process and manipulate about 200 million points. Uncheck if you want to import all the points of your file.

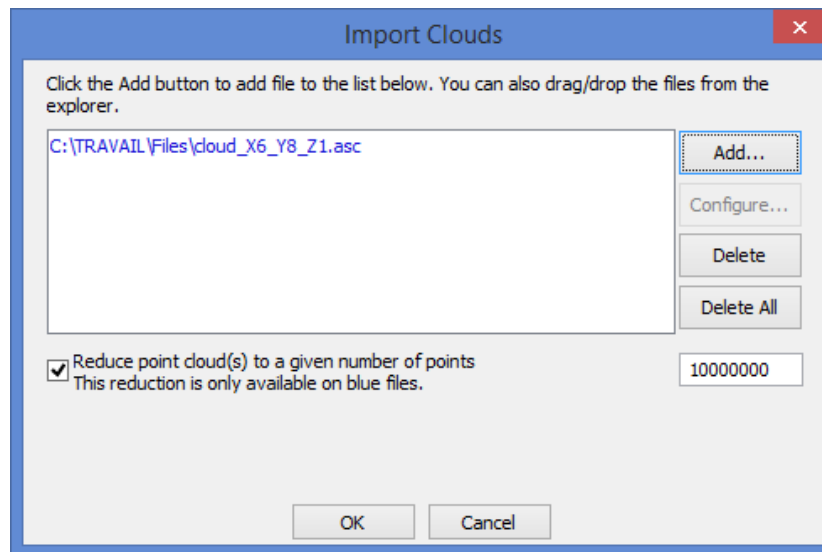


Figure 13: Import clouds

A very large variety of file formats can be imported, such as ASCII files and also formats used by Leica Geosystems (PTS for example). You can see below the list of the main formats. Please check the Help files for more details (press F1 in 3DReshaper) or in our [web site](#).

Ascii File *.asc, *.xyz, *.csv
 Ascii PTS file *.pts
 ShapeGrabber without transformation *.3pi
 ShapeGrabber with transformation *.3pi
 STL File as cloud (*.stl)
 PLY File as cloud (*.ply)
 Binary file (*.nsd)
 Tool path (*.iso)
 AutoDesk DXF file (*.dxf)

Ascii RAW file (*.raw)
 DMIS file (*.dms, *.rnr)
 GSCAN File (*.gsn)
 Confocal Scanning Laser Microscope (*.csv)
 Nikon NEXIV video measuring system (*.csv)
 Konica-Minolta-Vivid file (*.cdm)
 Perceptron SWB/SWL file (*.cdm)
 Konica Minolta CDK files (*.cdk)
 TScan files (*.ac)

Steinbichler files (*.ac)
 LAS files (*.las)
 Faro FLS/FWS files (*.fls, *.fws)
 Polyworks cloud files (*.psl)
 ASCII File (*.*)
 ESRI files (*.asc)
 Ascii File *.yxz (yxz)

Ascii formats are first previewed in a tab and you can configure how to read them before adding them to the list. It is also possible to display again the content of each file by selecting them in the list and clicking on “Configure...”

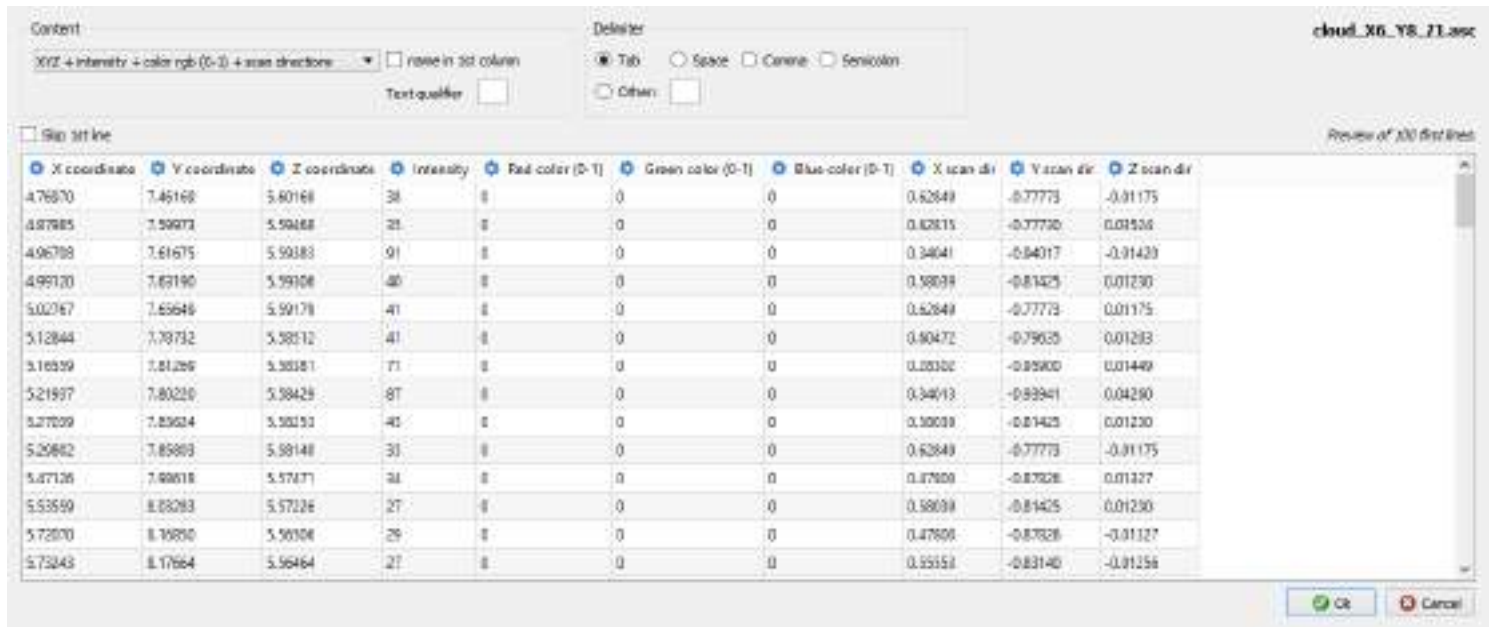


Figure 14: Import ascii point clouds

Note: Point clouds can also be extracted from a polyline and a mesh, using [Cloud\Extract cloud\(s\)](#). You can either extract only the vertices of the mesh, or compute new points using a sampling distance. For the polyline, the segments points are extracted.

Exercise: Import several point clouds at a time and merge them

Launch 3DReshaper and go to [File Menu\Import\Import cloud\(s\)](#). Click on “Add” and look for the clouds in the Samples folder, normally located on C:\Users\Public\Documents\3DReshaper 2016 MR1 (x64)\Samples. Select all the files from “ImportCloud-1.asc” to “ImportCloud-6.asc” by pressing the “Shift” key and click on “Open”. It is also possible to drag and drop the files into the Import window. In this case, all points will be imported.

Six independent clouds are created in the Cloud Group. They can be merged into one unique cloud. Select the six clouds and go to [Cloud\Merge Clouds](#). Choose to keep the colors of the clouds or not and validate the result with “OK”. You obtain one cloud called “Fusion Cloud”.

Scan point clouds

If 3DReshaper is directly connected to some manual measurement devices, you can go to [Measure\Measure shape / Digitize](#).

Note: If several scanner interfaces have been installed, you have to select the one you want to use.

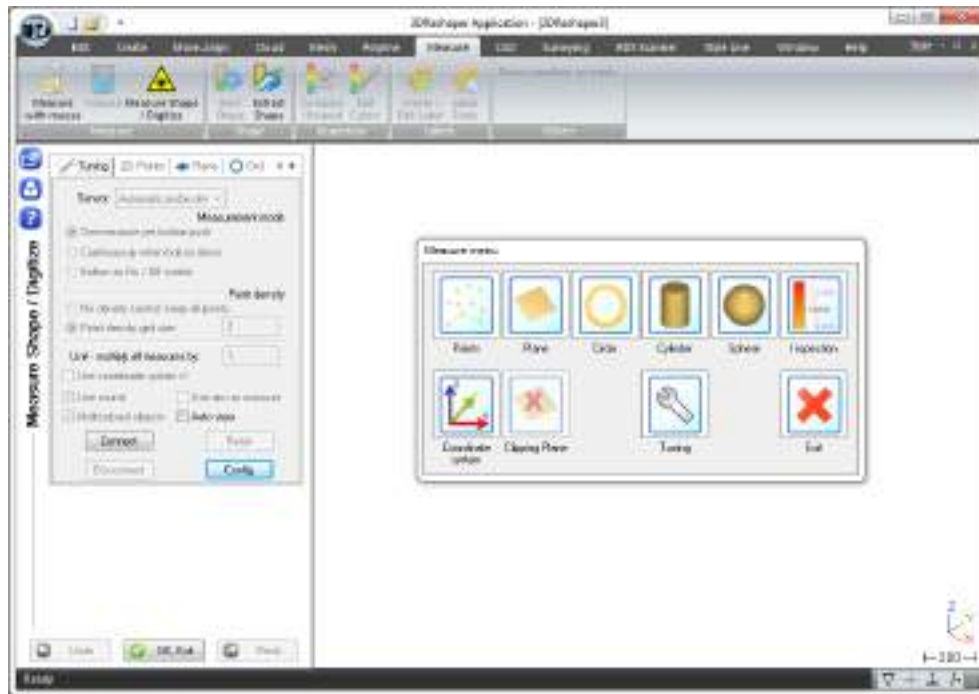


Figure 15: The menu with all the commands to measure directly with your arm in 3DReshaper

Main parameters are:

- Sensor: to select the device you want to use (mechanical probe, scanner, etc.).
- Measurement mode:
 - “One measure per button push” is useful when you measure with a mechanical probe.
 - “Continuously when button down” or “Button as On/Off switch” are more adapted to the scanner.

- Density control:
 - “No density control” will keep all the points coming from the scanner.
 - “Point density grid size” will make a uniform point density, independent of your measurement speed. It will also remove all bad or aberrant points and provide a shading representation. The value you enter in the Point density field is the size D of a square grid, which is projected on the shape during your measurement. During the measurement process 3DReshaper will keep only 10 points in each grid cell (see Figure 15).

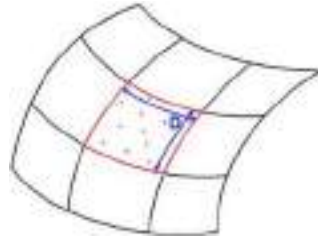


Figure 16: Density control representation

To launch the scan, you just have to click the “Points” button in the main menu. Then use the dedicated buttons on your arm to manage the measure (start / stop / validate / etc.).

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 3DReshaper, by using manual or automatic functions.

Exercise: Remove or separate a part of the cloud

Open the file “CleanWithObject.rsh”. The file contains the point cloud of a tunnel. To make an inspection of the tunnel, we need to keep only the points from the vault. We can use two different tools to split a cloud in several parts and separate the vault from the rest of the points.

Drawing a polygon

Select the point cloud “Tunnel” and go to [Cloud\Clean / Separate cloud\(s\)](#).

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 button “Remove last point” or 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. Press “SHIFT” while moving the ball in order to create a selection box around the points. **Choose the option “Delete INSIDE points” and press “OK, Next” to delete the points inside the selection box. Repeat the action on all the other big areas and validate with “OK”.**

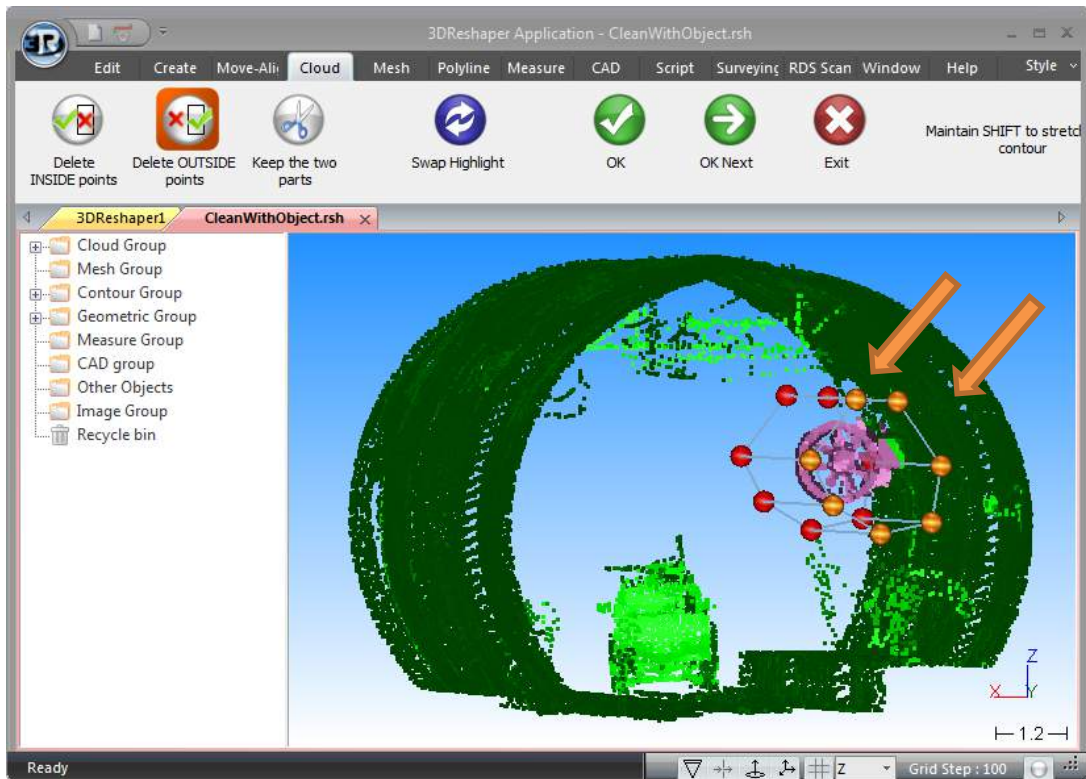


Figure 17: Remove a part of the cloud by drawing a polygon

Using an object

Right click on the cloud “Copy Tunnel” and tick “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 point cloud “Copy Tunnel” and the cylinder and go to [Cloud\Separate with Object](#).

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

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



Figure 18: Separate a cloud with an object

Using 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 [Cloud\Color along a direction](#). Choose "direction of a component" (📏) and click on the central axis. An arrow is displayed in the scene to represent the direction.

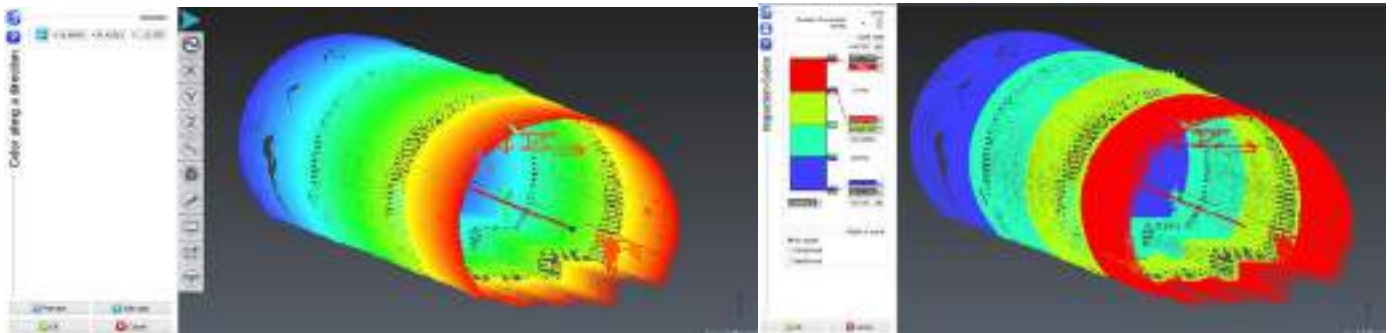


Figure 19: Color a point cloud along a direction

When you click on "Preview", the point cloud is colored along this direction with a color gradient. Click on "Edit color" to modify the colors. In the upper part of the dialog box, set the number of color levels to 4 and click "OK" to validate the color map. Click "OK" again to validate the colored cloud.

Note: You will see all the options of this second dialog box in the chapter Measurement, Inspection and Reporting.

Now we can explode the point cloud according to the colors we have just set. Select the point cloud and go to [Cloud\Filter / Explode cloud\(s\)](#). Select the option “Explode with color” and validate with “OK”. The four sub clouds are added into the Cloud Group and you can work on each one separately.

Exercise: Clean a cloud using automatic filters

3DReshaper provides several automatic filters in order to clean up a point cloud and remove noisy points.

Noise reduction

Open the file “Victory.rsh”. This file contains two point clouds, one with noise, and one already filtered. Show only and select the cloud named “Victory + noise” and go to [Cloud\Filter / Explode cloud\(s\)](#).

Choose the filter “Noise reduction”. The goal is to delete sparse points inside the cloud. The points concerned 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 slider to 80 in order to remove points which are in fact measurement errors (blue points in the Figure 19). A text appears in the dialog box to notify you how many points will be deleted.

Note: in order to work correctly, this filter requires a point cloud with a regular density.

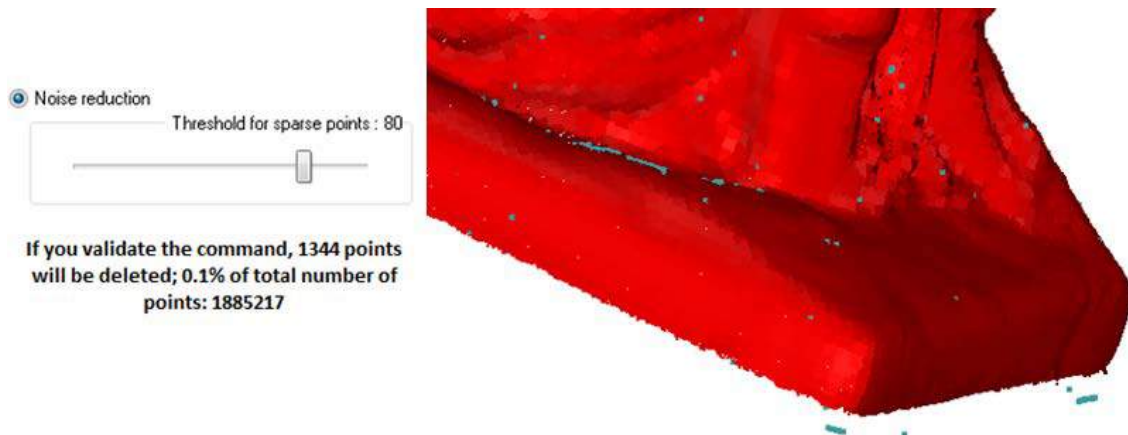


Figure 20: Remove noisy points in a cloud

Explode

Open the file “CleanPointCloud.rsh”. This file contains the point cloud of a dam having noisy and undesired parts. Select the cloud named “DamRock” and go to [Cloud\Filter / Explode cloud\(s\)](#).

Use the filter “Explode with distance” 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 that is entered. This distance also corresponds to the minimum distance between sub clouds. You can compute a first value by clicking on “Default value” and preview the result. Then you can change this value to fit your needs.

See the pictures below to have a better understanding of this parameter. Exploding the initial point cloud with the parameter set to 10 gives us four independent sub clouds.

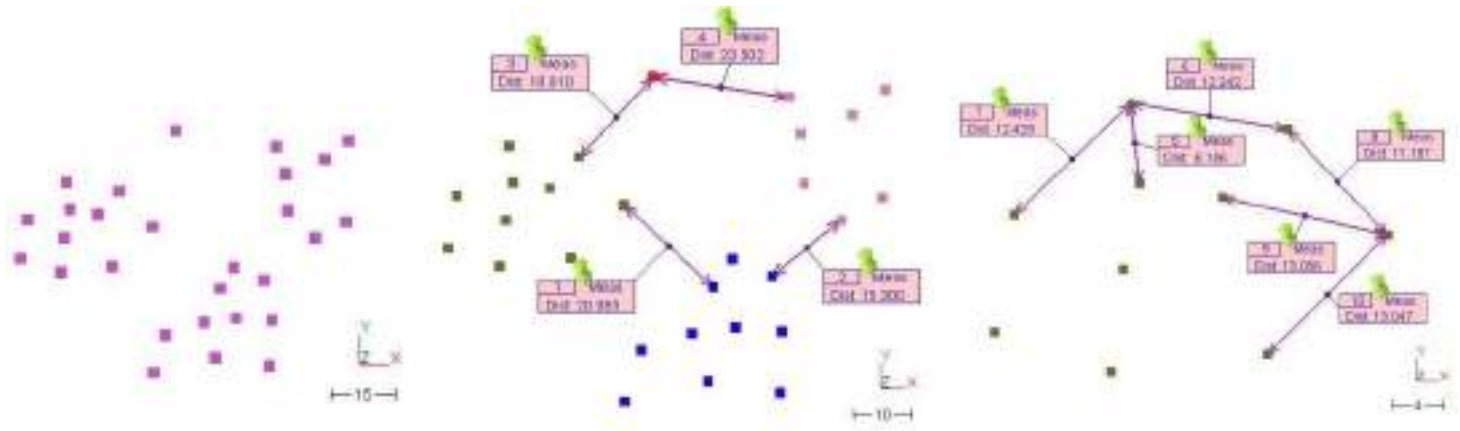


Figure 21: From left to right: Initial point cloud to explode; Point cloud exploded into four parts; One sub cloud

In the left picture we can see that 15 is the minimum distance between sub clouds. In the right picture we can see that 15 is the maximum distance between points in a sub cloud.

In our example, we can put 0.4 meter as the maximum distance allowed between points in a same sub cloud. You can delete automatically the small clouds containing less than 10 points. Click on “Preview” to preview the results and “OK” to validate them.

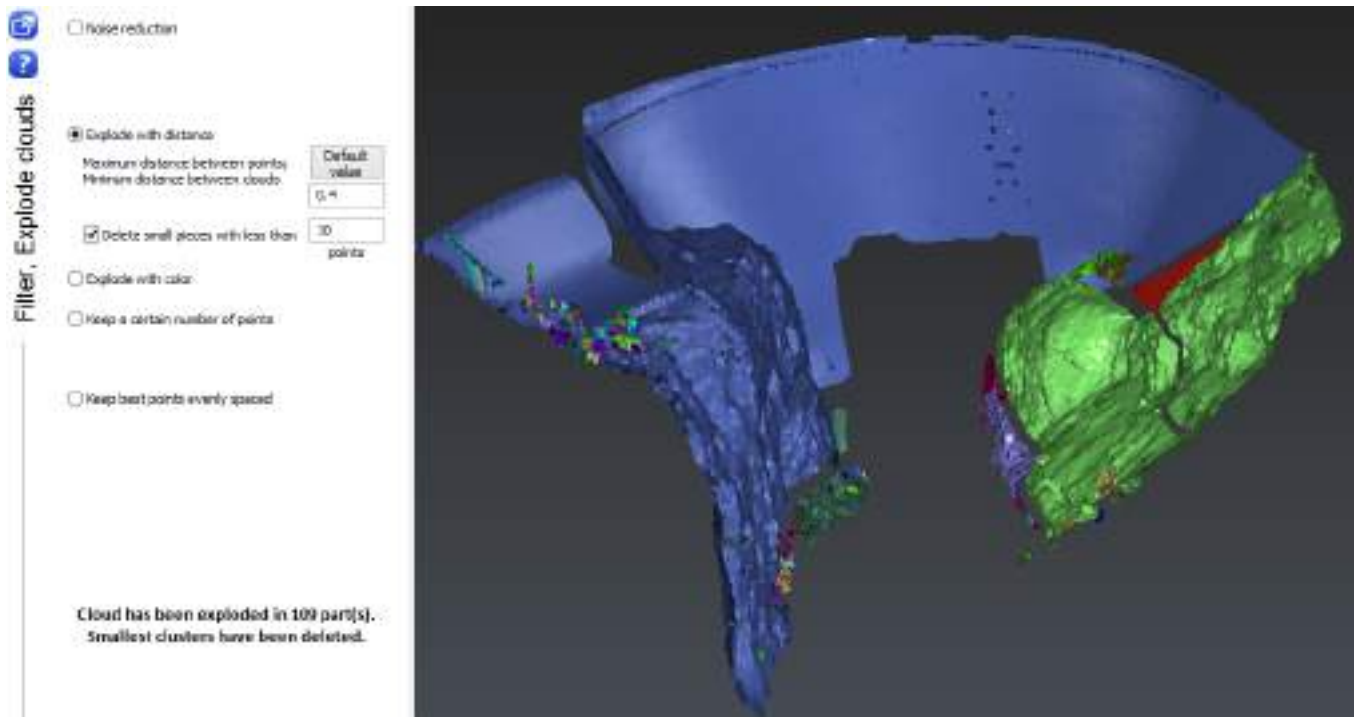


Figure 22: Explode a cloud with a distance criterion

All the sub clouds are added into the Cloud Group and ordered from the biggest to the 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. Once the explosion is completed, you can replay the same filtering process on one of the remaining sub-clouds.

Exercise: Reduce a point cloud

Keep using the file “CleanPointCloud.rsh”, select the biggest cloud and go to [Cloud\Filter / Explode cloud\(s\)](#).

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
- By keeping best points evenly spaced

Keep a certain number of points

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 than for the reduction of a cloud during the import.

In our example, the main cloud still contains about 1.5 million points. If we want less than 1 million points, the cloud can be reduced to keep only 60% of the points.

Keep best points evenly spaced

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

You can use the point cloud called “Copy DamRock” to test this filter. Display it in the scene, select it, go to [Cloud\Filter / Explode cloud\(s\)](#), and choose “Keep best points evenly spaced”. Enter 0.5 as the average distance between points and click on “OK”. The initial cloud had nearly 2 570 000 points and the reduced cloud contains only a bit less than 92 000 points. You can verify the number of points in the cloud with the “Properties” option in the contextual menu.

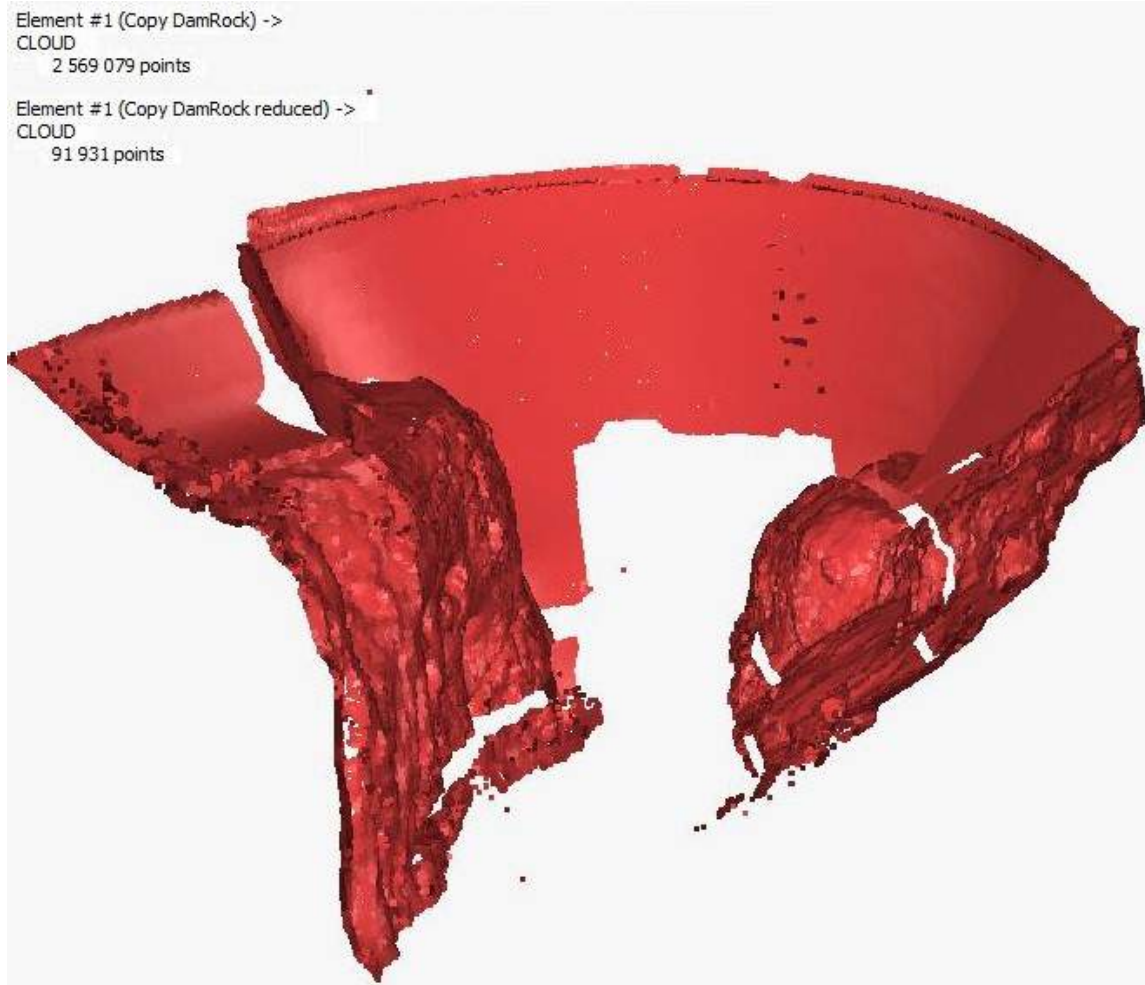


Figure 23: Reduce a point cloud

ALIGNMENT – REGISTRATION

Align clouds together

In most cases, you have to align all your scans in the same coordinate system, based on the shape of the scanned object or based on particular points (like 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, and 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.

Alignment according to the shape

Exercise: Best fit between clouds with overlapped area

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

Select all of them (for example press CTRL-A) and then go to [Move-Align\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 the options “Update coordinate system”.

Then click “Preview”, you will see a short report at the additional text box on the right in order to see transformations applied on each object, the registration standard deviation error and the registration mean error of each component. You can copy-paste this report 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.

```
Transformation of element n°1
(CloudToBestFit 1):
=====
> No movement
> Best fit mean error: 0.0523053
> Best fit standard deviation: 0.0739334
```

```
Transformation of element n°2
(CloudToBestFit 2):
=====
> Roll (around X): -0.757178 degrees
> Pitch (around Y): -0.201158 degrees
> Yaw (around Z): -1.26375 degrees.
> Gravity center translation:
DX-0.01521,DY0.2484,DZ-1.383
> Best fit mean error: 0.0516913
> Best fit standard deviation: 0.0686199
> Object n°2 has moved less
than: 1.402202
```

```
Transformation of element n°3
(CloudToBestFit 3):
=====
> Roll (around X): -0.211505 degrees
> Pitch (around Y): -1.07495 degrees
> Yaw (around Z): -1.68735 degrees.
> Gravity center translation:
DX0.02418,DY0.3931,DZ-1.462
> Best fit mean error: 0.051223
> Best fit standard deviation: 0.0694027
> Object n°3 has moved less
than: 1.607514
```

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 Figure 23, 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.



Figure 24: Comparison between two meshes with/without best fit alignment

According to specific points

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

Open the file “AlignTargets.rsh”. This file contains two clouds to align according to surveying targets. Show both clouds, and then select “Cloud 2” and go to [Move-Align\Best align N points](#).

The screen is automatically divided in two parts. On the left, there is the selected cloud (the one that will move), and on the right, the other one (the one that will not move). In the top ribbon, select only the option “Surveying target” and set a target with checkered pattern with a diameter of 0.1.

Select the option “Define constraints”, and then select the option “Preserve orientation of the Z axis” in the list box below as the Z axis is aligned with the vertical in both clouds. Unselect the option “Apply best fit” as we want to do an alignment according to targets only.

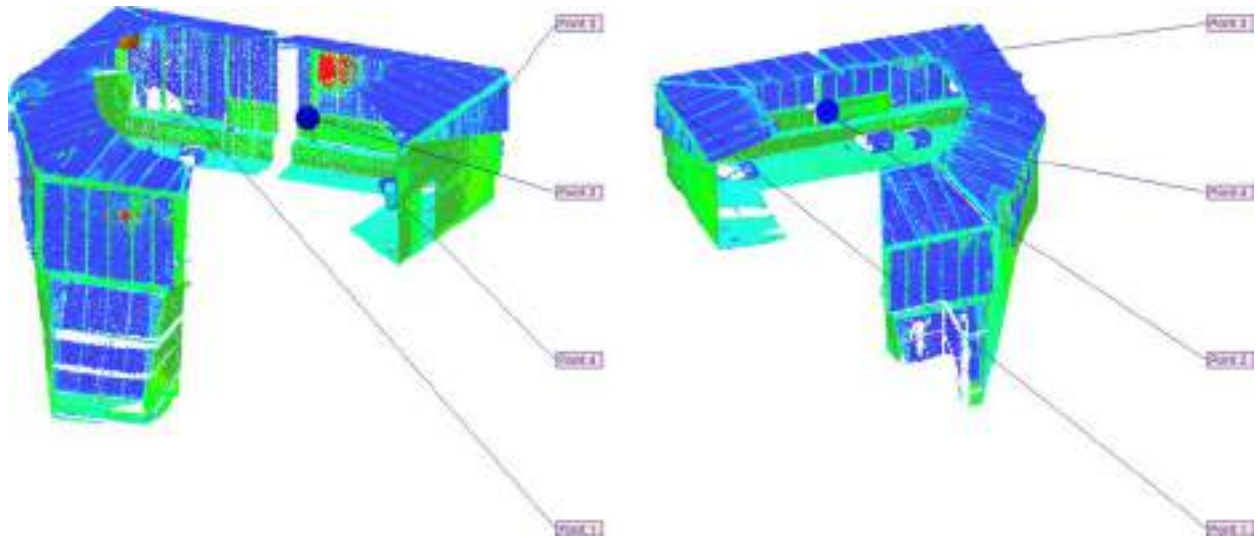


Figure 25: Use target centers to align point clouds.

Find the target corresponding to the point 1 on the Figure 24 and then click a point anywhere on the target in the left view. The created point will be automatically its 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.

Once the four couples of points have been clicked in both views, you can click "Preview".

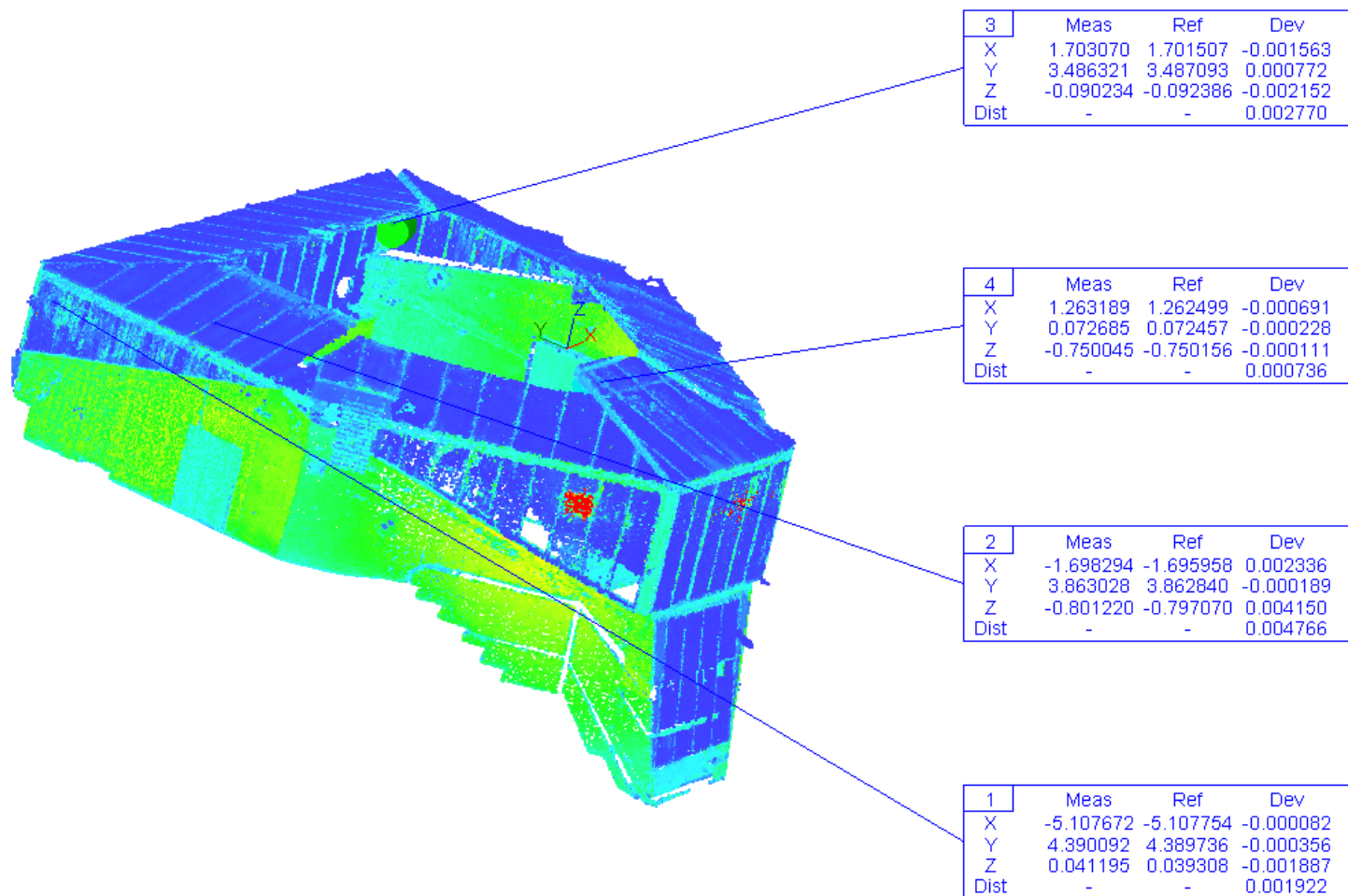


Figure 26: Labels to show deviations during the alignment according targets (less than 3mms in this case).

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”. Click on the label corresponding to the biggest deviation (left or right view) in order to inactivate this couple of point. The arrow on the target becomes gray. Click on preview again. The deviations should be lower and alignment is done with only 3 couples of points.

Click “OK” to validate. Both clouds are now aligned: you can use the command [Cloud\Merge clouds](#) 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 [Measure>Edit color](#) to take only one color level.

Create 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 façade, you may want to have the X axis orthogonal to the wall.

Exercise: Create a Coordinate System (CS) related to a 3D model

Open the file “BestFitOnRef.rsh”. This file contains two meshes, “Theoretical Dam (bad CS)” and “Theoretical Dam (good CS)”. Select and show only the mesh with the bad coordinate system “Theoretical Dam (bad CS)” (You will probably need to type ‘A’ to make a zoom all). Then press X, Y or Z on the keyboard. 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 on the keyboard. Now the axes are coherent.

Show only the mesh “Theoretical Dam (bad CS)”, select it and go to [Move-Align\Align Coordinate System](#).

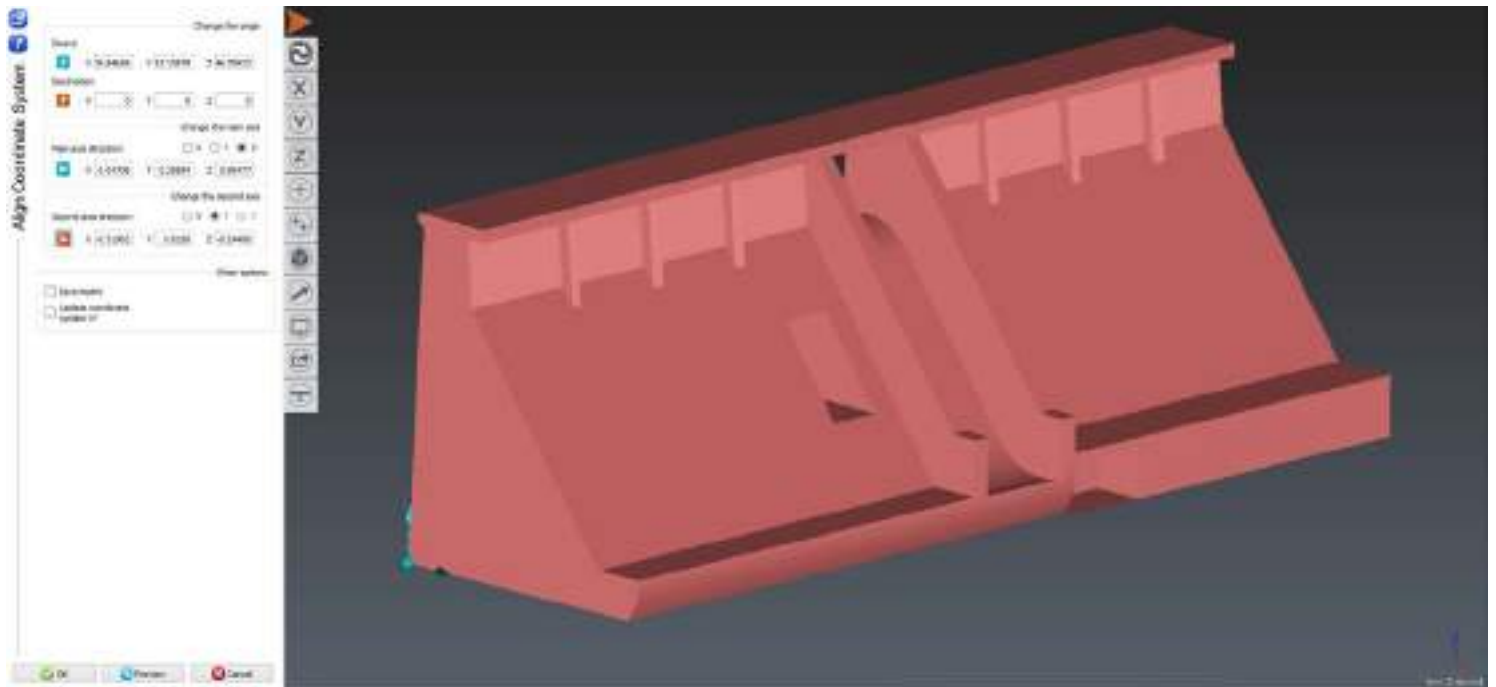


Figure 27: Create a coherent coordinate system on a dam

You have to respect an order when using the command (top -> bottom). 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 Figure 26). Let (0, 0, 0) for destination point as we want to have the bottom left corner of the dam at the origin of the new frame.

Then, change the main axis: choose Z, and click to define the new Z axis. For example use and click on an horizontal face of the dam, or use and click two points on a vertical edge.

Finally, define the second axis in a similar way by choosing Y and clicking on .

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.

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.

According to the shape

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

Open the file “BestFitOnRef.rsh”. This file contains a cloud “Measured Dam” and a mesh “Theoretical Dam (good CS)”. We will see how to align the cloud on the mesh.

Show only “Measured Dam”, and then select the point cloud and go to [Move-Align\Best align N points](#).

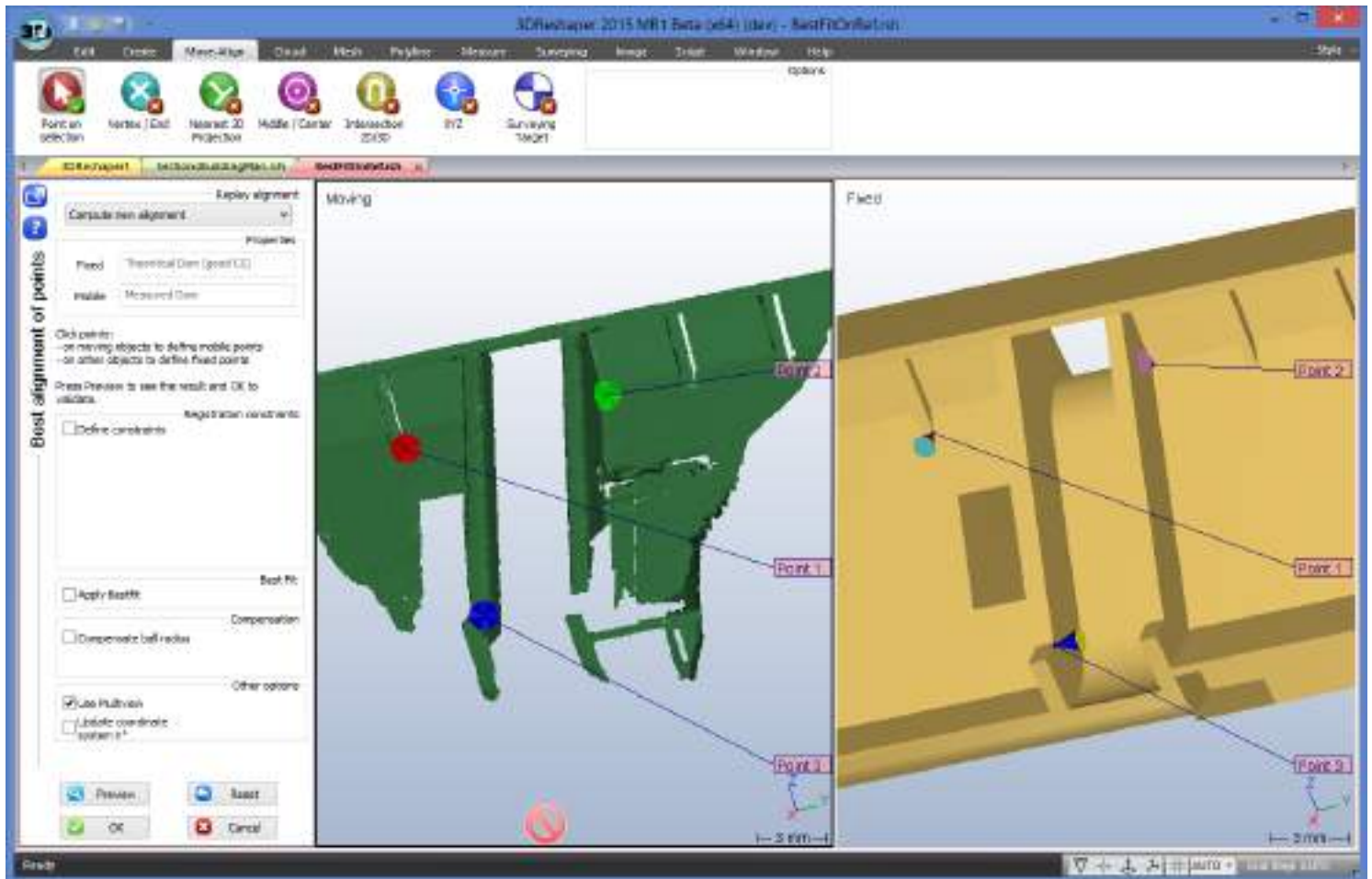


Figure 28: Enter points during the Best Align N Points command

The screen will be divided in two parts:

- On the left, the selected object(s), in this case the cloud “Measured Dam”. All objects on the left will move.
- On the right, the non-selected visible object(s), in this case the mesh “Theoretical Dam (good CS)”. All objects on the right will **not** move.

Select the option “Compute new alignment” in the list on the top of the dialog box. Then enter couples of points in order to do a rough alignment by saying this point on the measured cloud is approximately this point on the mesh. To do this, select

the option “Point on selection”, click a point on the cloud and then the corresponding point on the mesh. Do this for 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 27).

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

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

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

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

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

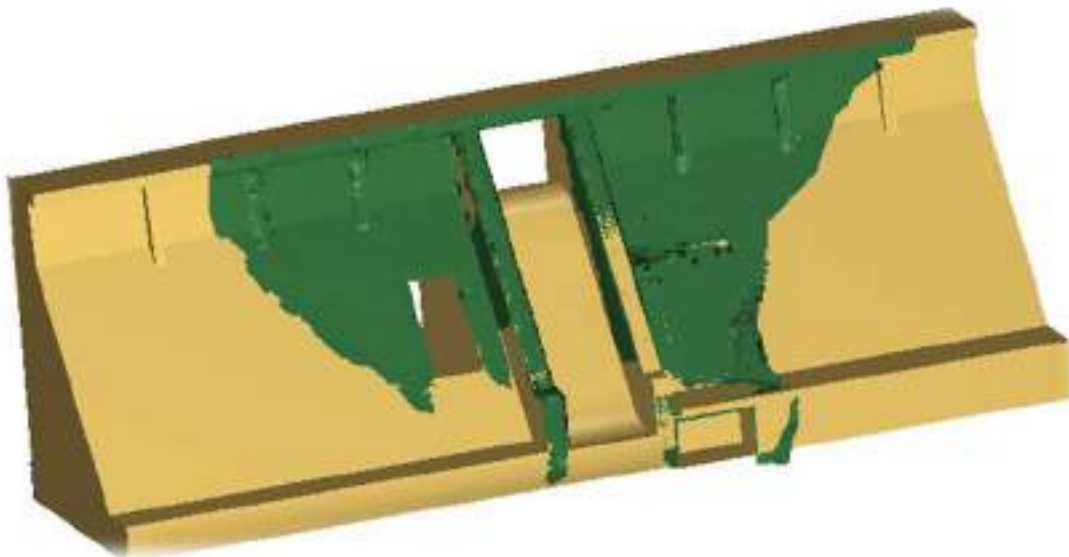


Figure 29: Cloud and mesh aligned with the best fit method

According to specific points or geometry

Exercise: Align a point cloud on a reference model according to the geometry (RPS)

Open the file “RPSOnRef.rsh”. In this file you will find a point cloud “CloudToAlign”, a circle extracted from the cloud “Extracted Circle on CloudToAlign”, (see Compute best shapes), a CAD object “Reference”.

The goal of this exercise is to align the point cloud on the reference thanks to the circle center and some points on the cloud.

Show the cloud, the circle and the reference, then select both the circle and the cloud and go to [Move-Align\RPS registration](#). The screen will be divided in two parts, on the left side you will have the cloud and the circle (will move), on the right only the reference (will not move).

Now you have to define couples of points in order to set up the RPS alignment: mobile points (clicked on objects selected when entering the command) and fixed points (clicked on non-selected objects: the CAD reference). For each pair of points, the constrained directions (X, Y and/or Z) have to be defined. In order to define a strict RPS alignment, one direction (X, Y or Z) has to be constrained 3 times, another direction has to be constrained twice and the last direction has to be constrained by one pair of points only. For this reason, a strict RPS alignment can be obtained using only between 3 and 6 pairs of points. In this exercise we will define 4 couples of points.

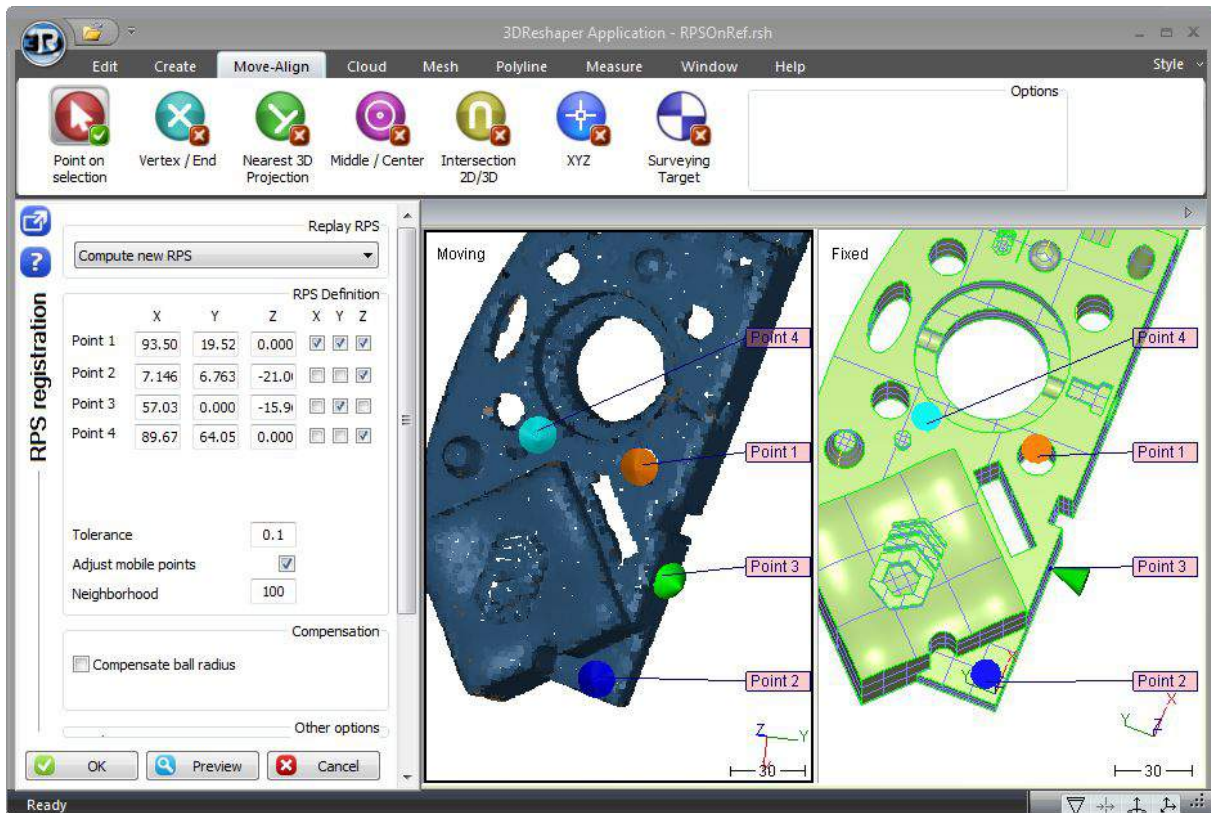


Figure 30: Couples of points to define the RPS alignment

- Point 1 – Circle center: select the option “Middle / Center” and then click a point on the blue circle and another one on the corresponding circle on the CAD object (orange point in Figure 29).
- Point 2 – A point on the small triangular part: select the option “Point on selection” and click one point on the cloud and one on the CAD on this little plane (dark blue point in Figure 29).
- Point 3 – A point on the side: keep the option “Point on selection” enabled and click one point on the cloud and one on the CAD (green point in Figure 29).
- Point 4 – A point on the top plane: keep the option “Point on selection” enabled and click one point on the cloud and one on the CAD (light blue point in Figure 29).

Now we will define the constraints for each point:

- Point 1: select X, Y and Z in the dialog box as this point has accurate coordinates (circle center).
- Point 2: select only Z as this point is on a plane where the normal is Z on the reference.
- Point 3: select only Y as this point is on a plane where the normal is Y on the reference.
- Point 4: select only Z as this point is on a plane where the normal is Z on the reference.

X is constrained in one couple of points, Y in two, and Z in three, so we have a strict RPS alignment.

Now enter 0.1 for the “Tolerance”: it means that if the distance between the point clicked on the cloud and the one clicked on the reference is bigger than this value, a warning will appear (in order to control the alignment).

Click “Preview”, the cloud will move in the right view in order to see the alignment result. The point cloud should be aligned on the reference. You may have a warning in order to say that the tolerance is not reached on non-constrained axes, it is because it is difficult to pick exactly the same point on the fixed and mobile components in the planar areas. But if you have a look at the small report next to the dialog box, the deviation on constrained axes should always be 0.

Point 1	Point 2	Point 3	Point 4
Dx = 0	Dx = -0.04909 (unconstrained)	Dx = -0.05533 (unconstrained)	Dx = 3.37 (unconstrained)
Dy = 0	Dy = -0.3828 (unconstrained)	Dy = 0	Dy = 6.19 (unconstrained)
Dz = 0	Dz = 0	Dz = -3.356 (unconstrained)	Dz = 0

In order to automatically identify which point in the cloud is the best one for the registration, you can select the option “Adjust mobile point” and enter 10 for the “Neighborhood”. Have a look to the deviations; they should be smaller now (on unconstrained axes).

You can now click “OK” to validate the alignment.

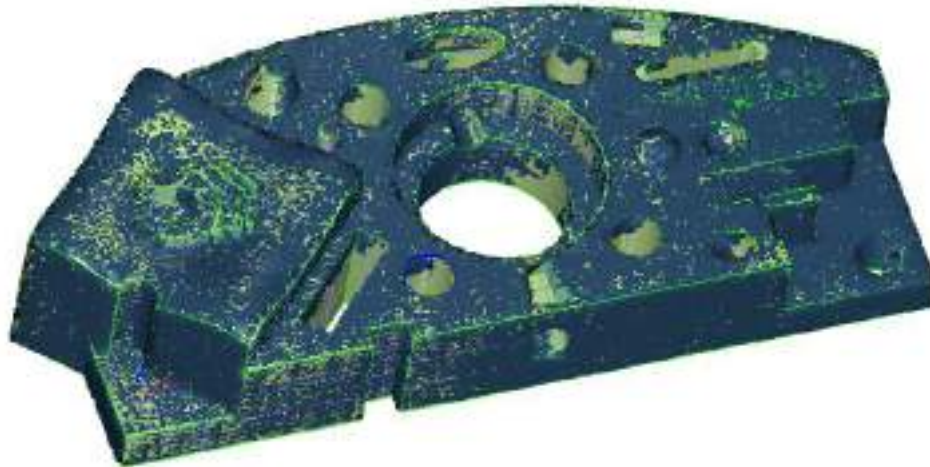


Figure 31: The cloud aligned on the reference thanks to the RPS alignment

MESHING AND MESH IMPROVEMENT

3DReshaper allows you to create a model by meshing point sets. This operation has several goals:

- To get an accurate surface model of your measured object.
- To control the quality of your digitalization (precision, lack of points, etc.).
- To keep only the most relevant points of your digitalization and thus reduce the model.
- To improve the accuracy of the result by eliminating incoherent points, filtering and (or) smoothing.
- To be able to export the result of your digitalization in other software even if this software is unable to treat files of several million points.
- To make reverse engineering.
- To reproduce the digitalized part: machining or making with rapid prototyping is possible with a mesh, but not with a 3D point cloud.
- To make 3D presentations, animations, photo-realistic rendering.
- To make finite element computations.
- ...

3DReshaper has various tools to mesh your point sets. A few parameters are enough, and make 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

3D Mesh creation

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

Open the file "Victory.rsh". This file contains two point clouds: one with some measurement noise (Victory + noise), and one already filtered with the command [Cloud\Filter Explode Cloud\(s\)](#) and ready to mesh (Victory) (see Noise reduction).

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).

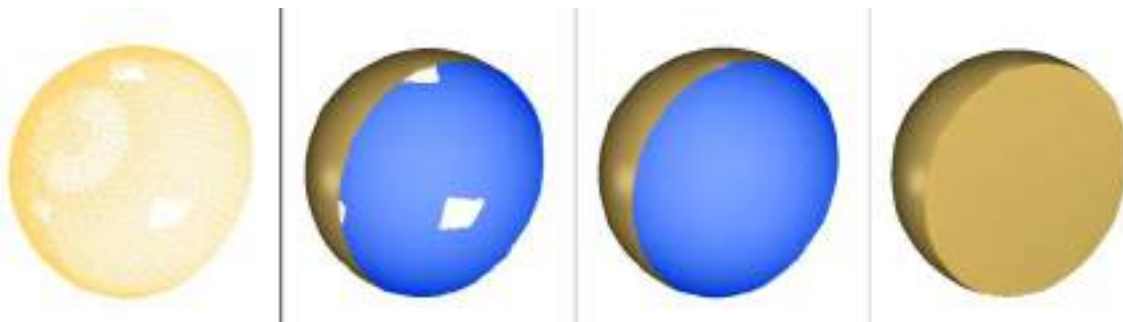


Figure 32: Holes management during the 3D mesh. From left to right:
1. The cloud to mesh; 2. Hole detection; 3. Keep only the external border; 4. Try to create a watertight mesh

You can find more information about holes in the “Holes filling” paragraph.

During this exercise, we will always select the option “Try to create a watertight mesh” as we want a closed volume.

Regular Sampling

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

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

Change the representation mode to “Flat + Wire” in order to see triangles shape. You will notice that they are quite regular and equilateral. The distance between vertices is approximately 0.6, the value entered for the “Average distance between points”, in other words the size of the grid projected on the cloud.

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

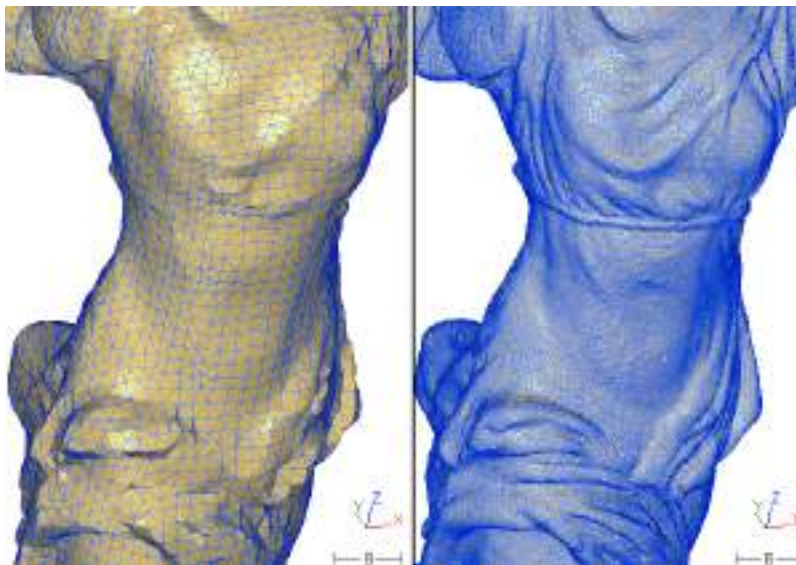


Figure 33: Regular Sampling: this method creates regular meshes where the size of the triangles depends on the average distance between points.

But in some cases, a regular meshing is not the best choice because the level of details is not the same all over the mesh, because of the measurement noise if you enter a small average distance or because the surface appears too faceted. In these cases, you can use the “Meshing in two steps”.

Meshing in two steps

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

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

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

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

There are two meshing methods:

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

Some parameters are common to both methods:

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

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

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

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

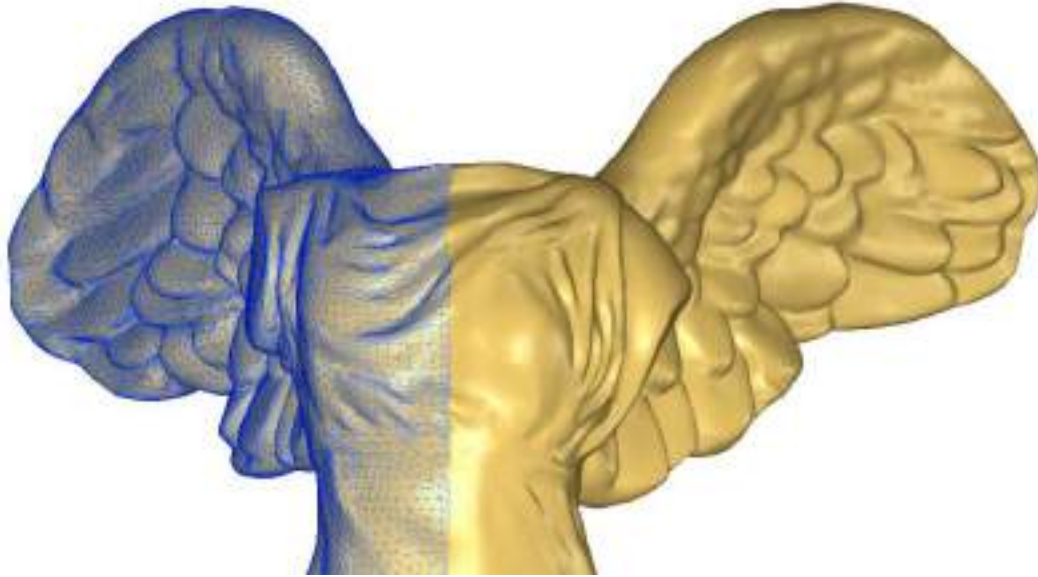


Figure 34: Refine by interpolating new points

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

As you can see in the Figure 34, the mesh is spikier, so we added some noise to the mesh. In order to avoid this, we should try again with a bigger deviation error. This noise does not appear with the method “Interpolate new points” because it has been reduced during the computation of the new points.



Figure 35: Comparison between the two refinement methods; Left: Interpolate new points; Right: Take points of the cloud

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

2D Mesh creation

Generally, to mesh a point cloud, the function most frequently used is [Mesh\3D Mesh](#). Unfortunately, this function does not always give the expected results when the points have been measured according to a direction.

Exercise: Generate a mesh from a cloud digitalized in top view

Open the file “2DMesh.rsh” that contains a measurement of a half glasses digitalized in top view. The measurement process did carry out a deviation error simplification every time the trajectory of the measurement probe was rectilinear. It results that the point density is highly non-homogeneous. If you make a rotation of the model, you will be able to note that the vertical walls contain very few points or even no point at all, the horizontal plane contains disseminated points and the “shape” part contains a high density of points.

For this kind of clouds, the quickest way to create the mesh is the command [Mesh\2D Mesh](#).

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

Note: If you enter 0 for the “Deviation error”, all the points will be meshed.

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

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

Note: If you want to create 2D meshes from points and polylines, you can use the command [Mesh\Constraint meshing](#) in order to add polylines as “breaking lines” or “feature lines” in your mesh.

Extrusion

In 3DReshaper, you can extrude a polyline along a path in order to create a mesh.

Exercise: Design a tube from a profile

Open the file “NeutralAxis&Extrusion.rsh”. This file contains a circular profile and the neutral axis of the tube to design.

Select the profile and go to [Mesh\Extrusion](#). Click the button “Select 1st path” and click a point on the neutral axis. Unselect all the options and click “Preview”. The result is not a regular tube 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. The tube radius is now the same everywhere.

Note: The option “Create a closed volume” will close the two holes at the beginning and at the end of the tube, the option “Make perpendicular to the path” will move the profile in order to be perpendicular to the first path vector.

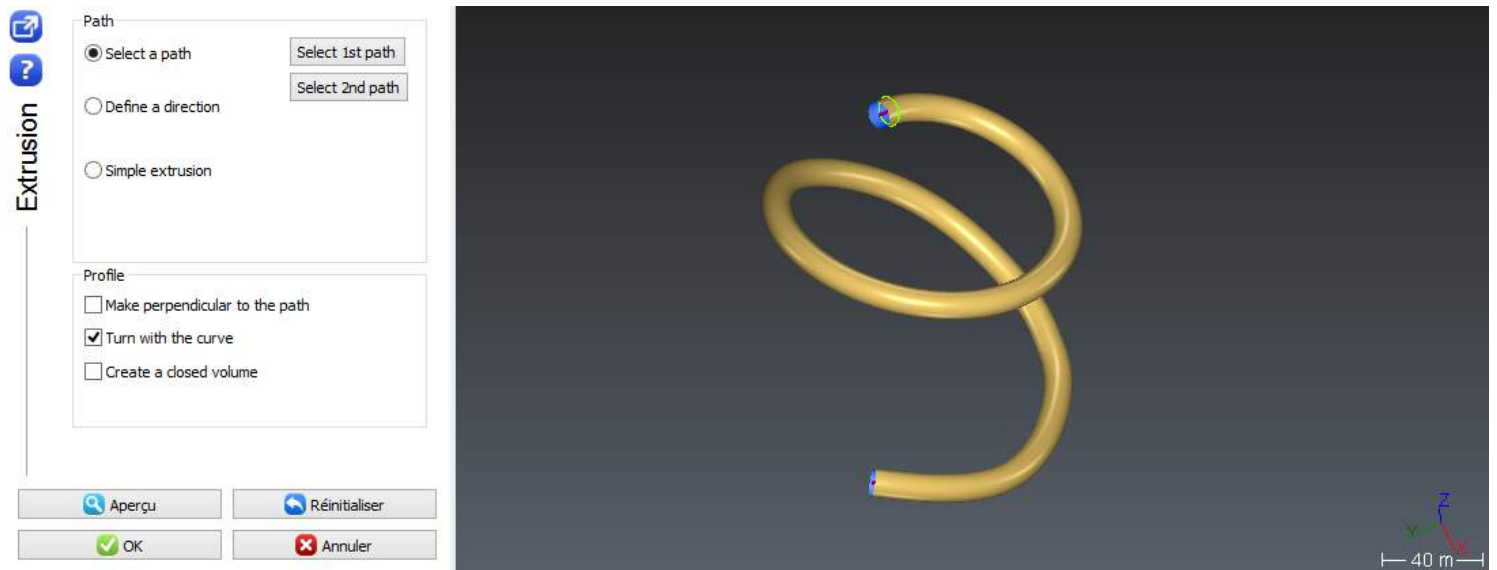


Figure 36: Extrusion along a path

Mesh improvement

In 3DReshaper there are a lot of tools in order to improve meshes. Main ones are described in this part.

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

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

Cut / remove triangles

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

Set the selection mode as "With a polygon", and then select the options "Select Through" and "Inside or crossing". Select through means that we will select all triangles inside the polygon, even the hidden ones. Set the view in order to see the entire wing and draw a polygon around it (like in the Figure 36).

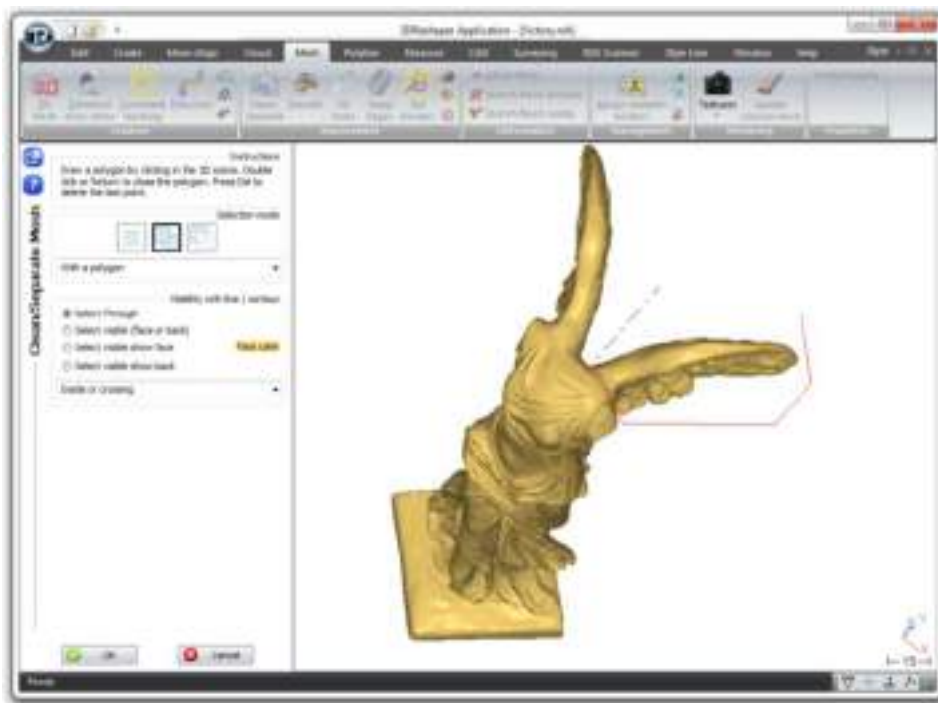


Figure 37: Cut a part of a mesh by selecting triangles inside a polygon

Press “Enter” to validate the selection. Then do the same for the second wing.

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



-> To restart the selection



-> To add to the selection



-> To remove from the selection

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

Refine mesh

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

Keep the file “Victory.rsh” opened and select both the cloud “Victory” and the mesh “Wing 1 (to refine)”, and then go to [Mesh\Refine Mesh\From a point cloud](#). We have to select the cloud AND the mesh because we will refine the mesh with the points from the cloud. As you can see, it is the same command as the one described page 41.

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:

- 0.02 for the deviation error
- 0.2 for the minimum triangle size
- 0.5 for the outlier point distance
- “No free border modification”, so that we can very easily merge all the parts at the end

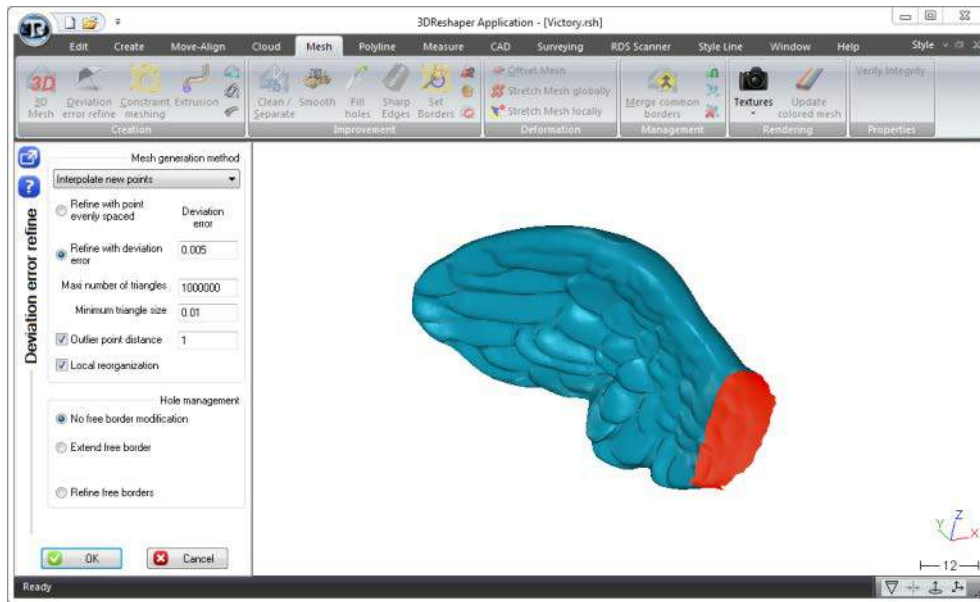


Figure 38: Refine with deviation error and interpolate new points

Click “OK” 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 size is smaller.

Reduce the number of triangles

Sometimes you need to reduce the number of triangles if:

- You want to export a very light mesh
- You do not need a lot of details on your mesh

Keep the file “Victory.rsh” opened and select the mesh “Wing 2”. Then launch the command [Mesh\Reduce](#). This command contains two tabs as there are two methods to reduce a mesh:

- Angle: means that you enter a number of triangles; 3DReshaper will reduce the mesh in order to respect this number.
- Deviation: means that you enter a maximum deviation; 3DReshaper will minimize the number of triangles and respect the given deviation.

Generally, the “Deviation” method gives a better result. So select the “Deviation” tab, and then:

- Enter 0.5 for the deviation
- Select the options “Optimize vertices position” and “Try to preserve equilaterality”
- Unselect the option “Reduce on free borders” in order to not modify the external border
- Select the option “Preserve sharp edges”, and set the angle to 10 and the specific deviation to 0.05

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

Merge meshes

With common borders

As we did not refine or reduce triangles on borders, it will be very easy to merge all the meshes in order to have a closed volume. You just have to select the three meshes (Main part, Wing1 and Wing2), then go to [Mesh\Merge common borders](#), select the option “Do not modify borders” and click “OK”. You will have only one mesh without any holes.



Figure 39: The Samothrace Victory with 3 levels of details

Exercise: Merge two meshes with different borders

Open the file “MergeMeshes.rsh”. This file contains 3 meshes, and the goal of the exercise is to merge the meshes together.

With different borders and without overlapped area

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

Select both meshes and go to [Mesh\Join 2 contours](#). Then click a point on each mesh border. The result is not the expected one because the complete borders have been joined, while we want the junction to be only on the middle part.

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

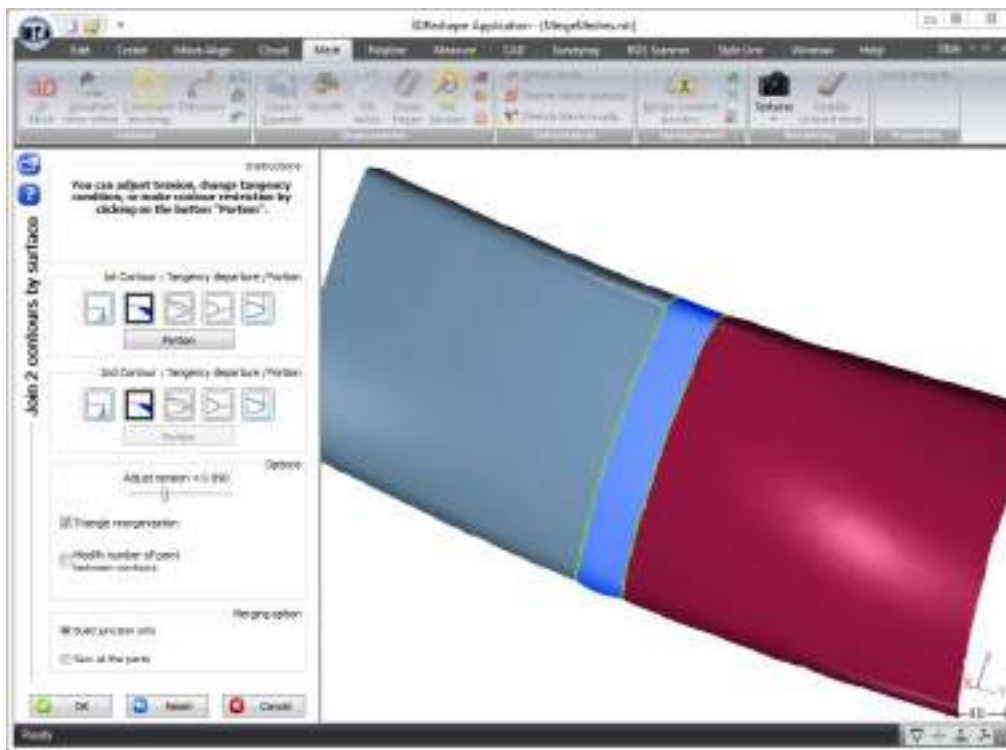


Figure 40: Merge two meshes with different borders and without overlapped area

Then you can click the buttons in order to change the tangency criteria (buttons with pictures next to the “Portion” button). Best choice in this case is “Tangent to the surface” (second button from the left). Select the option “Triangle Reorganization” in order to improve the result. Do not forget to select the option “Sew all the parts” in order to have only one mesh at the end.

Then click “OK” to validate. You can use the smoothing in order to improve the result on the sewing zone (see Local smoothing).

With different borders and overlapping area

Keep the file “MergeMeshes.rsh” opened and undo your work if necessary. Show only meshes “Part 2” and “Part 3”. As you can see, there is an intersecting area between the two meshes. We will have to remove triangles on this area and then use one of the methods described previously in order to merge the meshes (depending if the border is the same or not). There are three possibilities:

- Remove triangles and create a gap between the two meshes (use the command [Mesh\Clean / Separate](#)).
- Remove triangles and make the two borders strictly identical :
Select one mesh and the “Section” and go to [Mesh\Constraint meshing](#). Select the options “No direction, 3D computation” and “Cut the mesh along the polylines”. Do the same with the second mesh. This command will cut the meshes according to the polyline. Then, delete all the small parts or show only the 2 biggest parts, select the two biggest meshes and go to [Mesh\Merge common borders](#) in order to create only one mesh.
- Select both meshes and go to [Mesh\Stitch Meshes](#). Keep the default value and click Preview. The command will automatically stitch meshes according to the overlapping area (note that this command is not limited to 2 meshes).

Compound meshes

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

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

Exercise: Improve global aspect and edges

Generally, 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.

Smoothing

Local smoothing

Open the file “Smoothing&Edges.rsh”.

Select the mesh and go to [Mesh\Smooth\Replace a part](#). Select the option “With a freehand drawing contour” for example, and enable both options “Add new points inside hole” and “Curvature filling”. Then draw a contour around an aberrant zone like in the Figure 40. When you release the button, the surrounded area will be automatically replaced by a new smoothed mesh. You can repeat this on several areas in order to remove all small defaults.

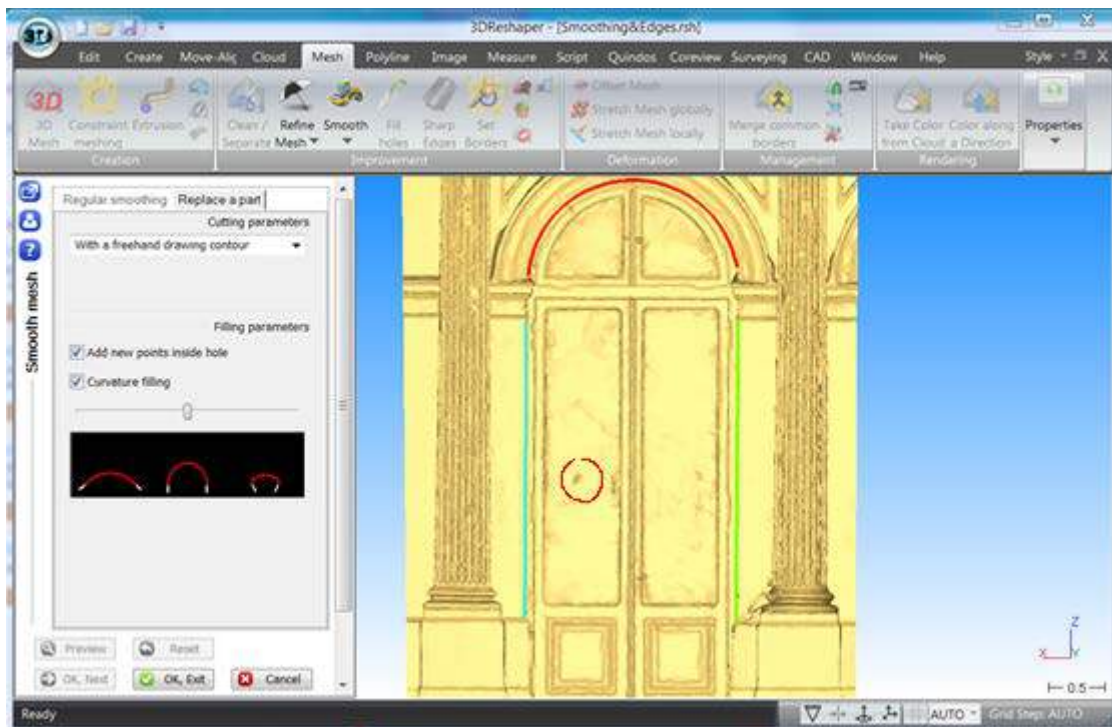


Figure 41: Replace a part of a mesh

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

Click “OK, Exit” to validate all the corrections.

Global smoothing

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

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

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

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

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

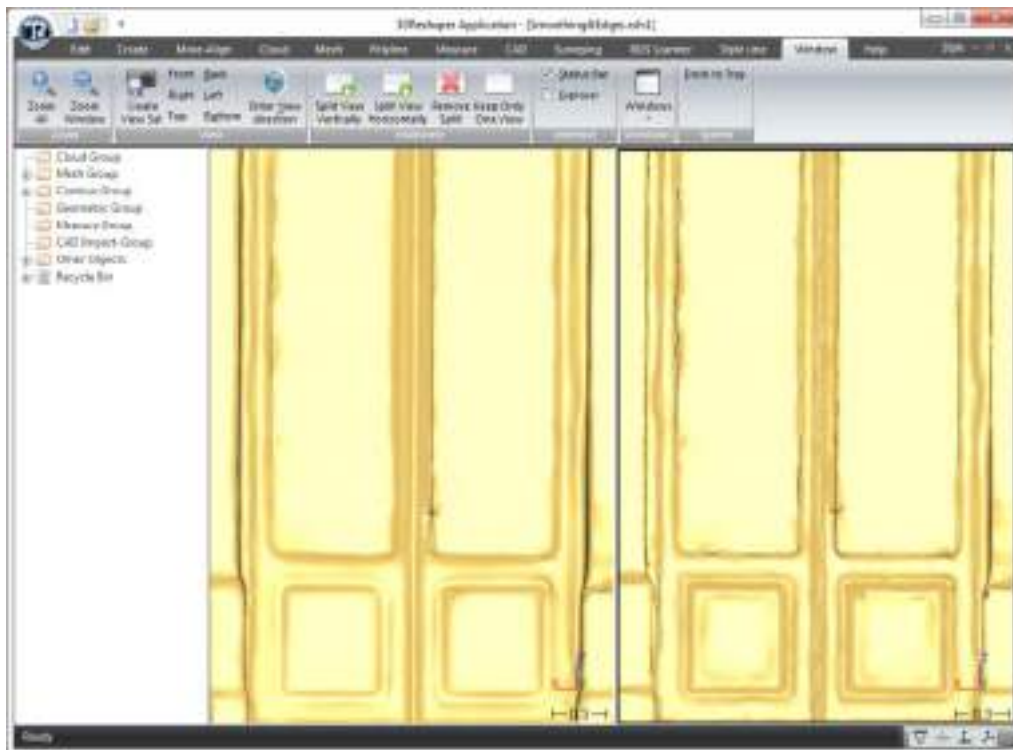


Figure 42: the smoothing without deviation control makes a smoother surface (on the left), while the smoothing with deviation control preserves the accuracy (on the right)

Re-create sharp edges

In 3DReshaper, 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 3DReshaper modifies the mesh automatically. There are several tools to create these polylines (like Fictive Line Extraction, Planes Intersections, etc.); they are all described in the part “Sections and Polyines”.

Keep the file “Smoothing&Edges.rsh” open. In the contour group, you will find three polylines:

- Edge Vault (obtained with the command Extract Feature Line -> Fictive Line)
- Edge Corner 1 (obtained with the command Extract planes and Planes intersection)

- Edge Corner 2 (obtained with the command Extract planes and Planes intersection)

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

Once the point is clicked, press the “Preview” button. The 3 sharp edges will appear on the mesh. Click “OK” to validate.

Do not hesitate to try to click different points in order to see the difference.

Holes filling

In 3DReshaper, concepts of “holes” and “free border or external border” are identical.

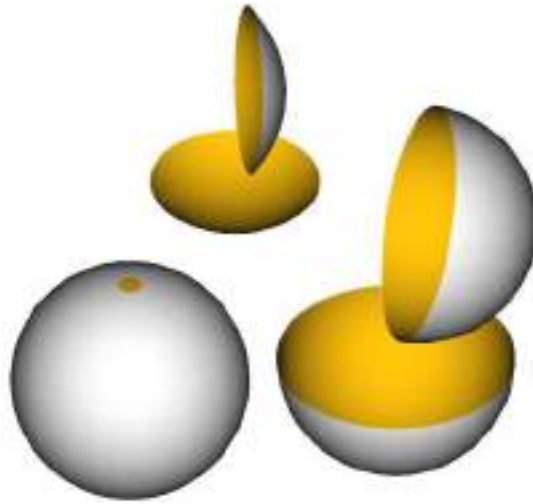


Figure 43: Spheres drilled with a small hole or spherical skullcaps are strictly identical from the topological point of view

Imagine for example a hollow sphere in which you pierced a small hole on the surface. The edge of this element is a circle that you call “hole”. Imagine now that the dimension of this hole increases until reaching the complete diameter of the sphere! This element that is always a circle is now called “external border”. This concept of “hole” or “external border” is thus only a question of vocabulary, but from a strict mathematical point of view, it is the same thing.

Exercise: Fill holes with curvature filling on a mesh

Open the file “FillHoles.rsh”. It represents the face of a statue, but the back of the head could not be measured, as well as a part of the nose. If you expand the “Mesh group” in the object explorer, you will see the 4 steps of this exercise. The goal is to recreate the complete head, but it is not possible to use directly the command “Fill holes” on the back of the head because the hole is really too big and because the neck part is not a hole to fill.

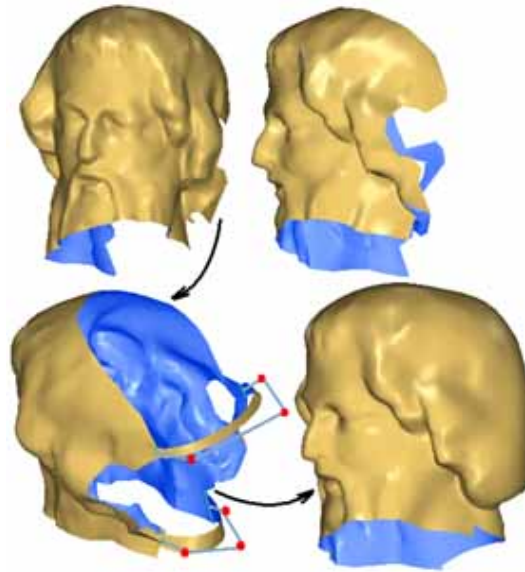


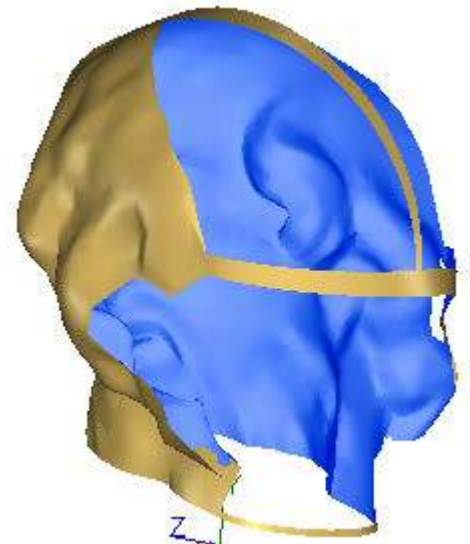
Figure 44: In the case of the file "FillHoles.rsh", it is necessary to preserve continuity with the surface of the border to recreate the back of the head.

So the first step is to start from the object "1-StartingMesh" and to create bridges like on the picture. The result to obtain is something like "2-ReadyToFill"; you can click on this object in the object explorer to have an idea of the result to obtain (you can also click on it during your bridge command).

Create a bridge between free borders

Launch the command [Mesh\Bridge](#). We will create two bridges: a first one in order to recreate the back of the neck, and a second one in order to make a round part on the back of the head.

- Click the two free edges as shown on the Figure 43. 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.
- Move the slider to adjust the "Middle twist orientation".
- When you are satisfied with this first bridge, select "Sew all the parts" and click "OK Next" to start your second bridge.
- Click the two free edges as shown on the Figure 43. The default bridge you obtain looks too much like a circle.
- Press the "Ctrl" key to select the 3 red balls together.
- Move the 3 balls together to get something like "2-ReadyToFill".
- Note that you can also create a junction attached on another junction like you can see on the opposite picture.
- Click "OK Exit".



Fill holes

Now we can fill all the holes with the command [Mesh\Fill holes](#). Select the object you have obtained previously or simply select the object "2-ReadyToFill" and launch the command.

- Select the option "By click" and click all the holes except for the neck hole.

- Activate options “Filling Holes” and “Curvature filling”. Adjust curvature filling slider to increase or decrease the volume of the head.
- Press the preview button. You should obtain something like the object “3-AfterFill”.
- 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 [Mesh\Smooth](#) and you will obtain a shape like “4-ClosedMesh”.

Improve borders

In 3DReshaper there is a dedicated command to automatically fit mesh borders with polylines in order to have perfect borders/holes.

Exercise: Reconstruct perfect holes on a mechanical part

Open the file “MeshImproveBorders.rsh”. In this file there are a mesh and 5 polylines corresponding to 5 holes. They have been obtained thanks to Features Extraction (see chapters “Compute best shapes” and “Extract geometric features”) and polylines improvement (see Improve polylines).

Select the mesh and the 5 polylines and go to [Mesh\Set 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 the value.
- Click the button “Click two points” and then enter two points in order to define the distance.

Enter for example 10, a warning message is displayed saying that 2 borders are too close. Click anyway on “Preview”. The result is bad because two holes have not been reconstructed due to value too high.

Enter now 1.5 then click “Preview” again. The result is now perfect like in Figure 44. You can click “OK” to validate.

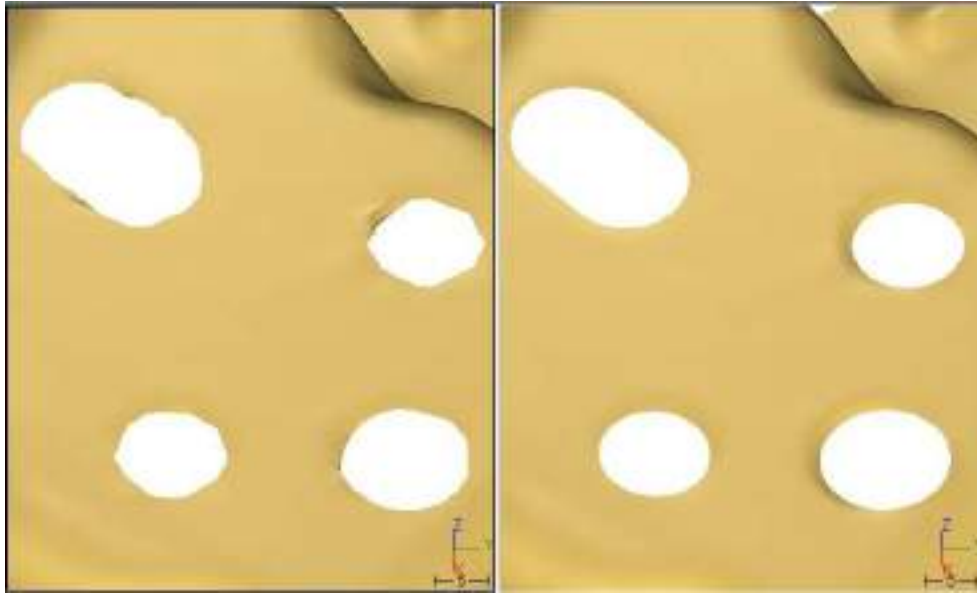


Figure 45: Borders have been improved in only one click

Color a mesh with colors from the cloud

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

Exercise: Apply the point cloud colors on a mesh

Open the file “MeshColor.rsh”. This file contains a colored point cloud and two meshes with different triangle size. Show only the point cloud and the mesh with small triangles, select these two objects and go to [Mesh\Take color from cloud](#). Select the options “Previous color, if any, is overwritten ONLY if one point in the cloud can provide the color” and “Only if the distance to vertex is less than 1% of the mesh size” and click “Preview”. The color has been correctly applied on the mesh. Click “OK” to validate.

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

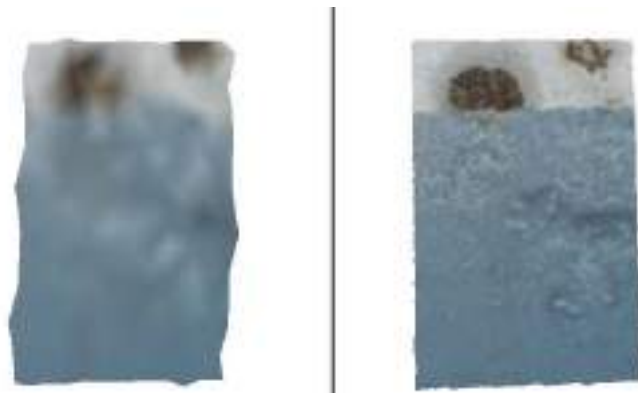


Figure 46: When we apply colors from the point cloud on a mesh, the triangle size is very important.

SECTIONS AND POLYLINES

3DReshaper provides multiple tools to create, manage and improve polylines for a large variety of applications, such as inspecting cross sections, 2D drawings, computing contour lines, or even extracting feature lines to reconstruct sharp edges of a mesh.

In 3DReshaper, polylines can be imported, created manually, created by drawing sections on clouds and meshes, or extracted by detection of feature lines on a mesh.

Draw polylines

Polylines can be created manually with the command [Create\Polylines](#). Click two or more points in the 3D scene (see How to click a point in the chapter 3DReshaper Basics). The last point can be canceled with “Del” or “Backspace” key. Press “Enter” key to end a polyline. Click a new point to begin a new polyline or press “Enter” again to stop drawing.

Create sections

With 3DReshaper it is possible to create different kind of sections on a mesh or on a point cloud:

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

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

Exercise: Dynamic sections on a mesh

Open the file “SectionsDynamic.rsh”. It contains the mesh of the dam used in the exercise concerning point cloud processing.



Figure 47: Create a planar freehand section on a mesh

Select the mesh and go to [Polyline\Freehand Sections](#). Choose the option “Unique planar section” 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. Be careful to choose “Point on selection” in the upper ribbon in order to click points on the mesh.

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

Now validate the result. A new polyline is created (or a set of polylines. In this case, explode the set and continue the exercise with the longest polyline). Select the polyline and the mesh and go to [Polyline\Sections along Curve](#). You can see an arrow on the polyline indicating its direction.

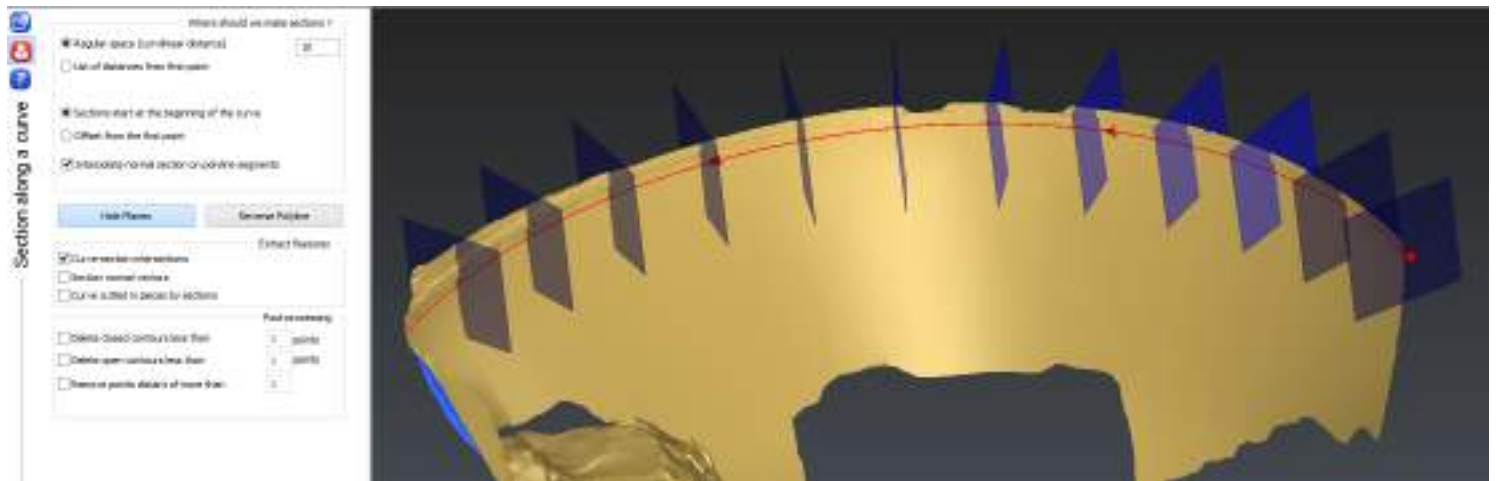


Figure 48: Compute sections along a curve with a regular step

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

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

With the advanced parameters, it is possible to create points on the polyline where sections are created. See the Help files for more detailed explanations.

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

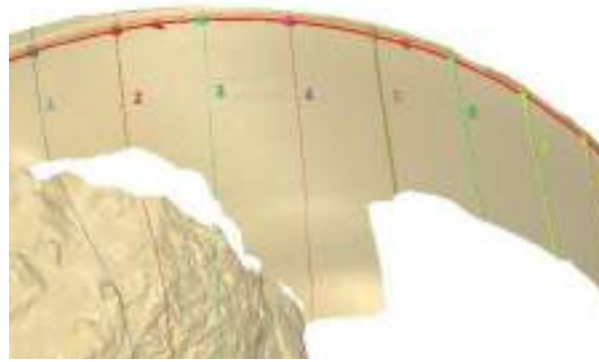


Figure 49: Polylines and sets of polylines

Note: You can explode the sets of polylines by selecting them and go to [Polyline\Explode Polylines](#).

Exercise: Radial sections on a mesh

Keep using the file “SectionsDynamic.rsh”. In the scene, show only the mesh “DamRock” and the polyline “Center axis” located in the Contour Group. We are going to create regular sections on the mesh with a different tool allowing to compute planar sections around an axis.

Select the mesh and go to [Polyline\Radial Sections](#). Use “Center axis” to define the axis (▶ to define its direction, + to define its position). The axis is displayed in the 3D scene (in yellow, in the image below), with arrows showing the direction of the rotation. If you set the view in Z (press “Z” on the keyboard), you can see that the axis is vertical.

Enter the number of sections that 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 ($360/30 = 12$).

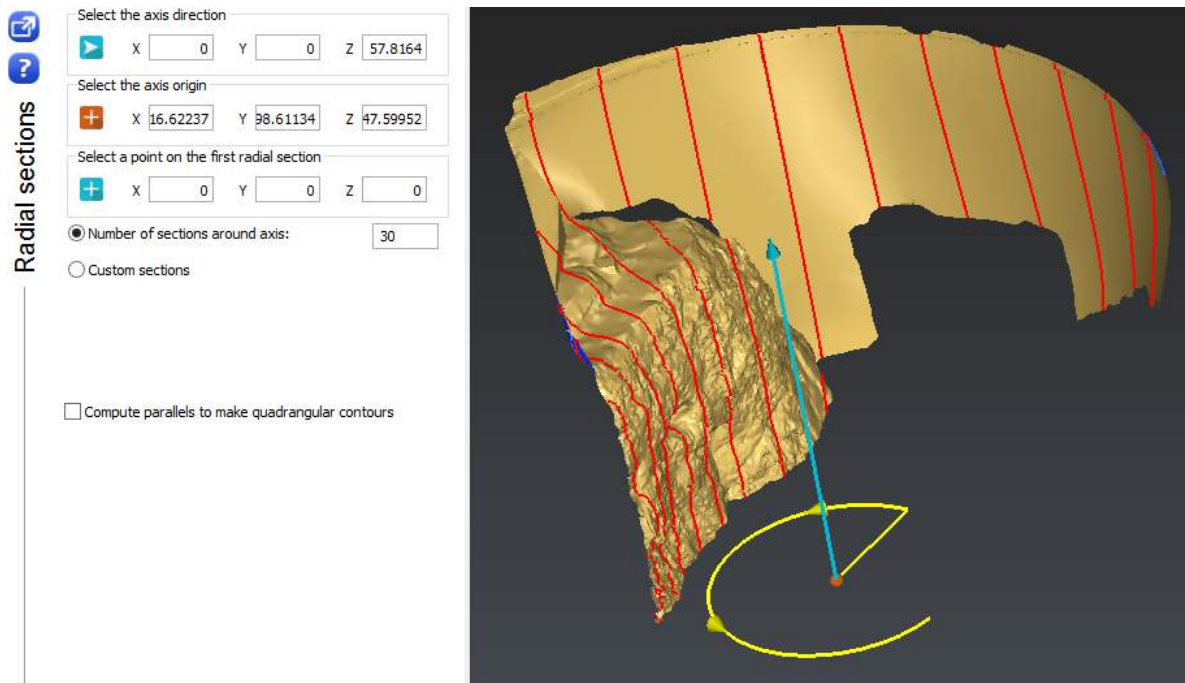


Figure 50: Create radial sections on a mesh

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

Open the file “SectionsContourLines.rsh”. It contains the mesh of a mountain and we want to create contour lines on it. Select the mesh and go to [Polyline\Planar Sections](#). This command allows computing one or more sections, all in parallel planes defined by a given direction.

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

When you press “Preview” a little message box appears in the bottom right corner to inform you that 36 polylines have been extracted. Click “OK”. All the polylines are inserted in a new folder called “Planar sections along Z MountainLake” in the Contour Group. Each polyline is named after its value in Z.

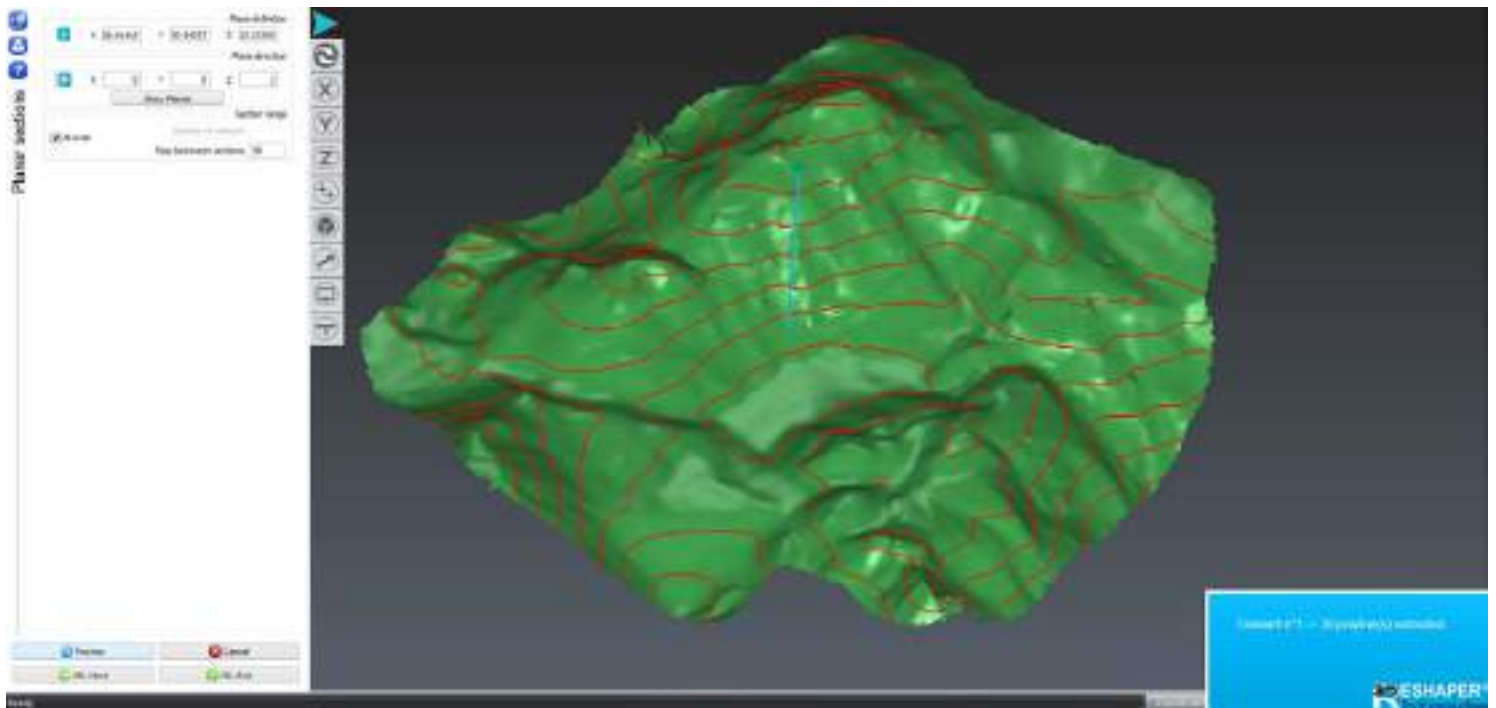



Figure 51: Create contour lines

Exercise: Create a planar section on a point cloud

Planar sections can also be computed directly on point clouds. Open the file “SectionsBuildingPlan.rsh”. It contains a point cloud of the ground floor of a building. The inner points have been removed in order to lighten the file, only points around the walls have been kept. We will use this example again in the following paragraphs to show how to create the 2D plan of the building.

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

To define the section plane, place the mouse in the scene and press “X” on your keyboard to set the view along X axis. You may need to press also “A” to make a zoom all. Click on  in the dialog box and zoom in the scene to click a point on the top of the point cloud.

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

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

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



Figure 52: Parameters to create a section on a point cloud

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

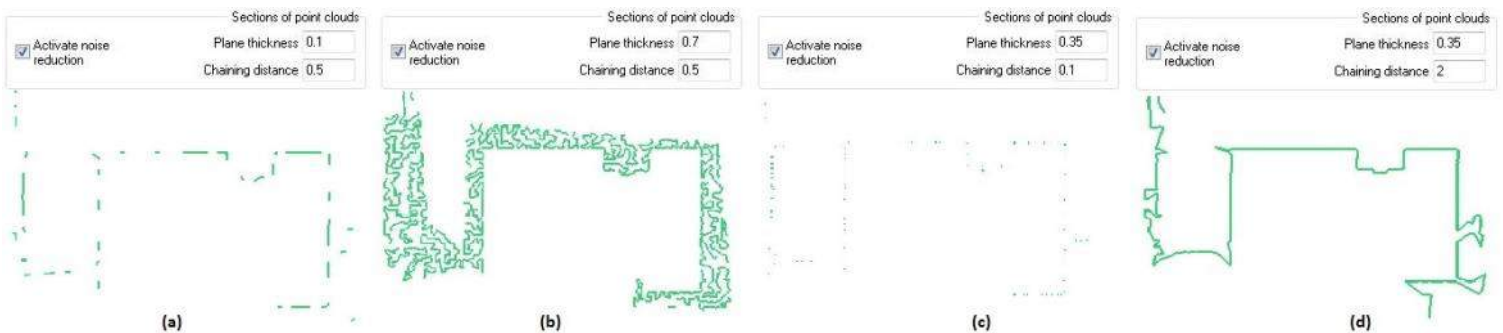


Figure 53: Section on a point cloud - Compare values for plane thickness and chaining distance

If you enter a plane thickness too low, there will not be enough points so you will have very short polylines as in Figure 52(a) and you can lose some details. If you enter a plane thickness too high, the points on the floor will be taken into account, so you will have very noisy polylines as in Figure 52(b).

If you enter a chaining distance too low, you will have a lot of short independent polylines as in Figure 52(c). If you enter a chaining distance too high, all the points will be chained together as in Figure 52(d). You will obtain long polylines and you may need to cut them for further processing.

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

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

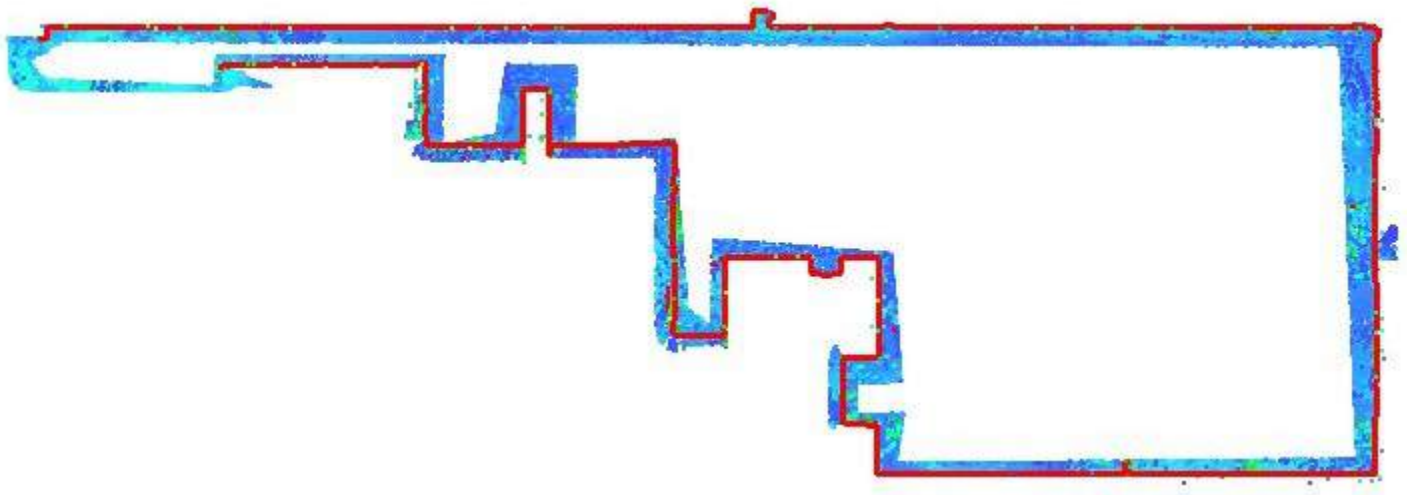


Figure 54: 2D plan of a building

Manage polylines

Cut polylines

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



Figure 55: Polyline cut in three parts

Chain / group polylines

Polylines can be automatically chained together by computing new segments between unconnected points. You can try to chain the polylines called “Lines to chain” in the file “SectionsBuildingPlan.rsh”. Select the polylines and go to [Polyline\Chain/Group Polylines](#). Select the options “Chain and create new segments” and “As many Polylines as necessary”. The new segments appear as dotted lines.



Figure 56: Automatic chaining of polylines

Polylines can also be grouped in a set of polylines. For example, in the file “SectionsContourLines.rsh” seen previously, after creating the planar sections, we could group the polylines and create one set of polylines per elevation.

Note: A set of polylines can always be exploded again by using [Polyline\Explode Polylines](#).

Improve polylines

Exercise: Create the 2D plan of a building

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

Simplify / reduce a polyline

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

Select a small noisy polyline (for example “LineToChain1”) and go to [Polyline\Reduce](#). Set the number of points to have in the reduced polyline. If you want a straight line, enter 2. Choose the option for the polyline positioning. The option “Optimize point position” will compute a polyline going through the noise in order to reduce the standard deviation error. The option “Resample evenly” will keep the position of the two end points of the polyline and create new points evenly spaced.

This tool can also be used to recreate the right angles on a polyline. You can try it on the polyline called “RightAngles” in the file “SectionsBuildingPlan.rsh”. See the result in the picture below.

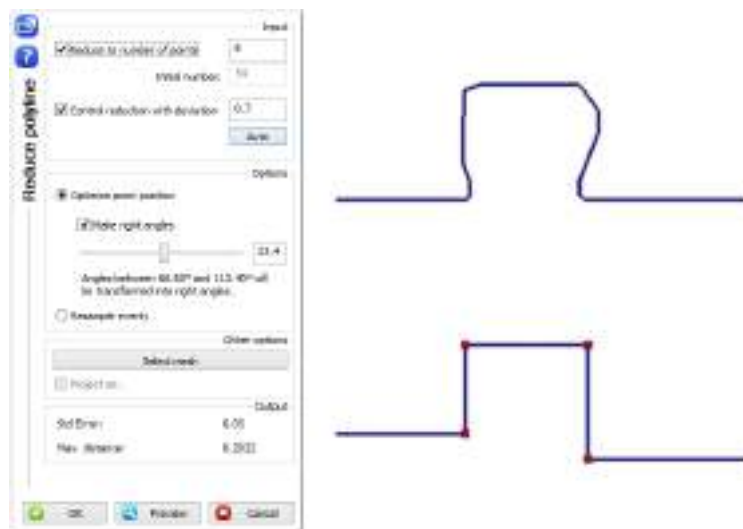


Figure 57: Reduce a polyline and make right angles

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

Smoothing

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

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

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

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

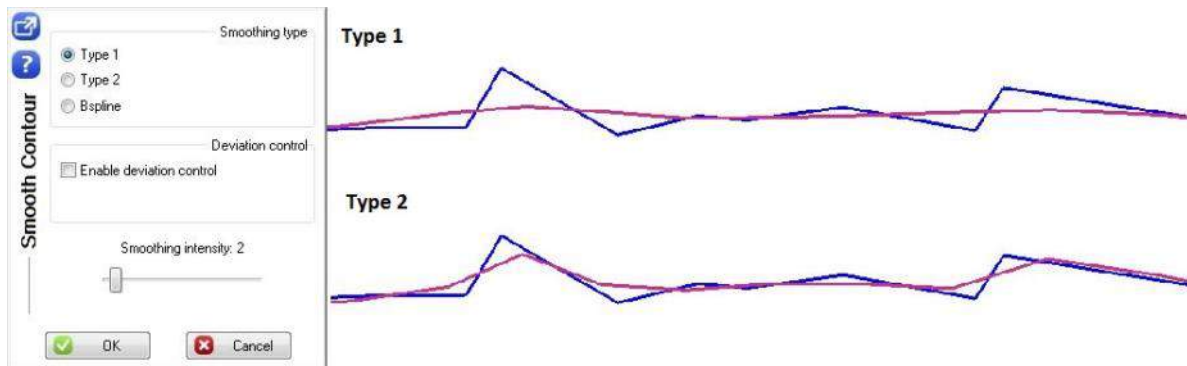


Figure 58: Smooth a polyline

Stretching

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

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

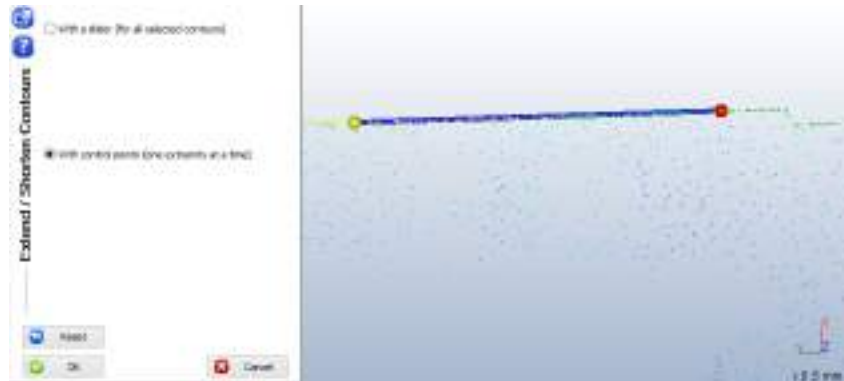


Figure 59: Extend or shorten a polyline

A polyline can also be modified with several control points. Select a polyline and go to [Polyline\Stretch Contour](#). Two control points are automatically displayed at the extremities of the polyline. You can add intermediate control points by increasing the number of “Control points evenly spaced” or by clicking specific points on the polyline in the 3D scene.

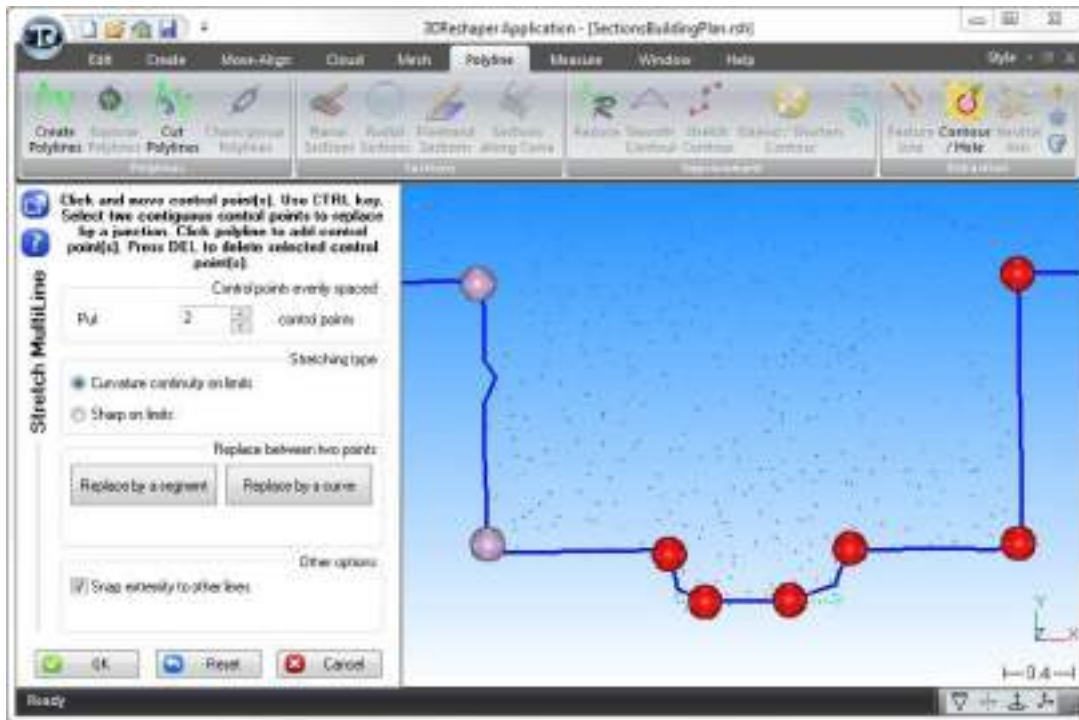


Figure 60: Stretch a polyline with several control points

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

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

At any time, you can:

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

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

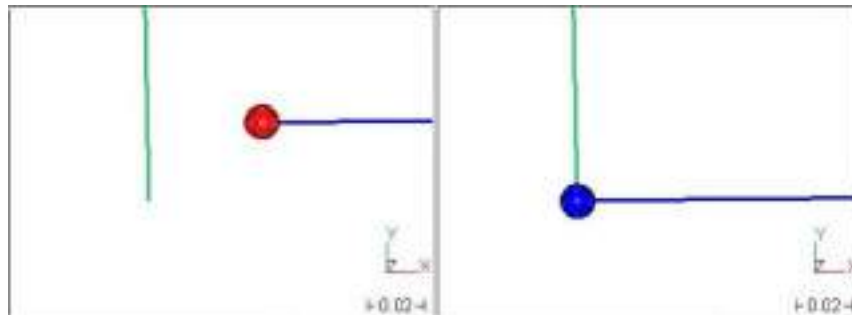


Figure 61: Connect two polyline

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

Offset

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

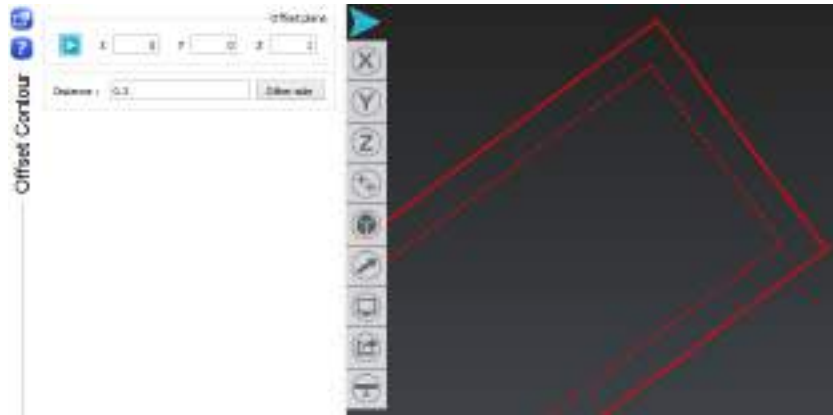


Figure 62: Offset a polyline

Feature / Border / Fictive lines

3DReshaper provides tools to compute polylines following sharp edges, fillets or small radii of a mesh.

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

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

Three types of lines can be extracted from a mesh:

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

The command [Polyline\Feature Line](#) is an additional tool that is not included in the standard configuration of 3DReshaper. It is however automatically included with the CAD plugin, as it is essential for reverse engineering process.

Note: The extracted lines could also be useful for constraint meshing. See chapter 3D Meshing and Mesh Improvement.

Exercise: Extract a feature line from a mesh

Open the file “FeatureLines.rsh”. It contains the mesh of a pillar and four points which will help you for the first steps. Select the mesh and go to [Polyline\Feature Line](#). This command is divided in four tabs concerning respectively:

- Feature line extraction
- Border and Fictive lines extraction
- Smoothing lines

- Sharp edge reconstruction

Feature line

Click a first point on the mesh, choose the method and click the second point on the mesh. The detection starts only when you click the second point. The line will start and end at the nearest vertex from the points clicked and go through the vertices of the mesh (excepted if “Projected and smoothed contour” is selected).

You can choose between four methods to extract a feature line. Each method tries to find the shortest line between the selected points, but uses different constraints:

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

Note: the “Projected and smoothed contour” option is not compatible with the other options: changing for an option not compatible will reset the points.

To preview the different methods as shown in the picture below, proceed as follows: click the first point near “Start_Point”, choose the first method, and click the second point near “End_Point”. To see the next method, press the key “DEL” to delete the last point, change the option and click on the “End_Point” again (Press the “DEL” key twice to test the fourth method).

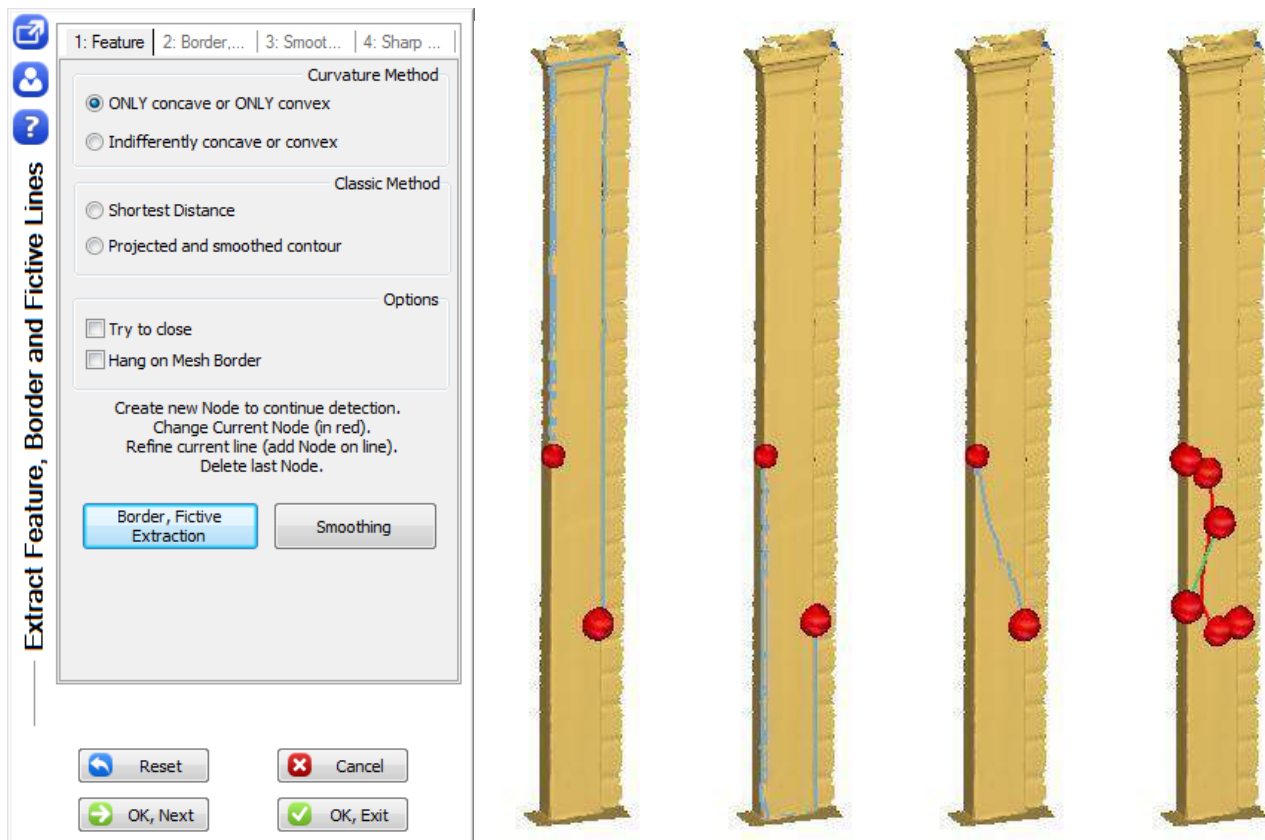


Figure 63: Four methods to extract a feature line

Clicked points appear as red balls. You can move these points by clicking on them and keeping the button pressed. The green ball is a point which was hanged onto another line or mesh contour if “Projected and smoothed contour” is selected.

Note for the 3 first methods: you can add intermediate points on the way, or add points after the last one to stretch the line. The last point clicked can always be canceled by pressing the key “DEL”. You can click on the first ball to reverse the direction of the line.

Several advanced options are available in this dialog box. You can check the detailed information in the Help files from 3DReshaper.

Once the feature line is extracted, you can either extract also the border and fictive lines, or smooth the feature line, or even extract another feature line with “OK, Next”.

Exercise: Rebuild a sharp edge using a fictive line

Keep working with the file ‘FeatureLines.rsh’. Show the points “Point 1” and “Point 2”. Select the mesh and go to [Polyline\Feature Line](#). Choose the option “ONLY concave or ONLY convex”. Click the first point near “Point 1” and the second point near “Point 2”. Check that the feature line lies only on one edge, then click on “Border, Fictive Extraction” to continue with the extraction of the two border lines and the fictive line.

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

Border lines

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

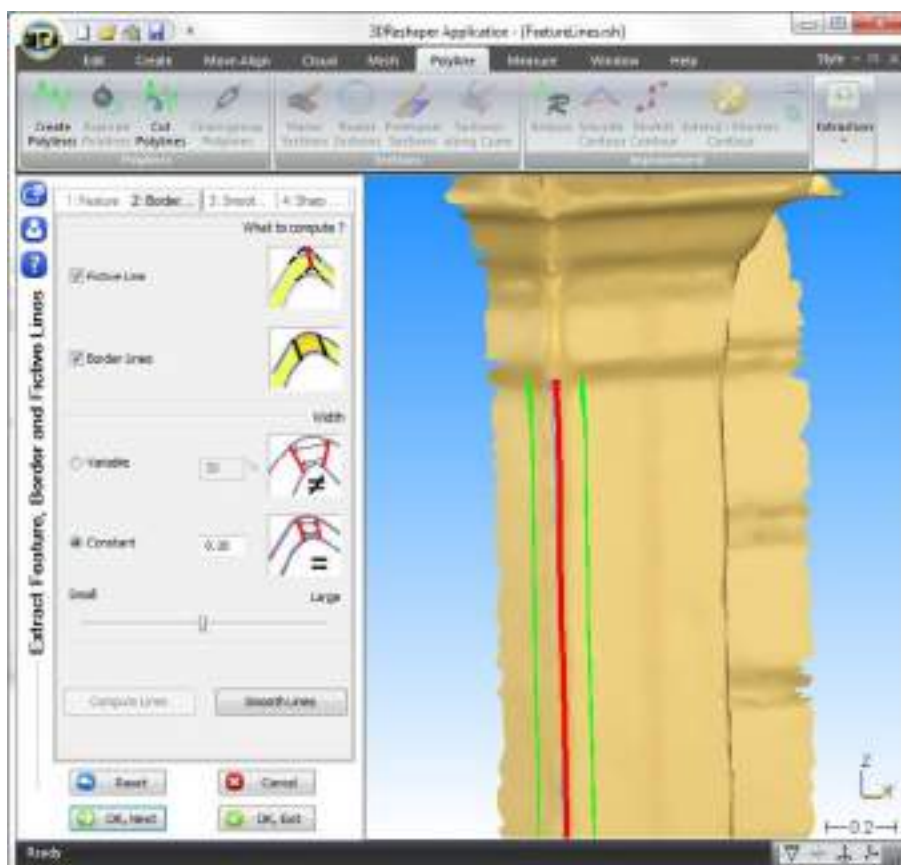


Figure 64: Extract the fictive line and the two border lines

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

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

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

Smooth lines

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

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

Various options are available in this tab:

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

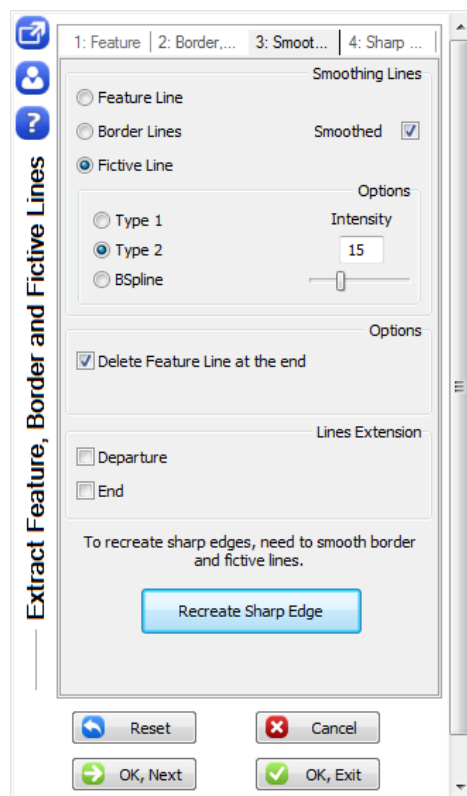


Figure 65: Smoothing tab for lines extraction

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

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

Sharp edge reconstruction

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

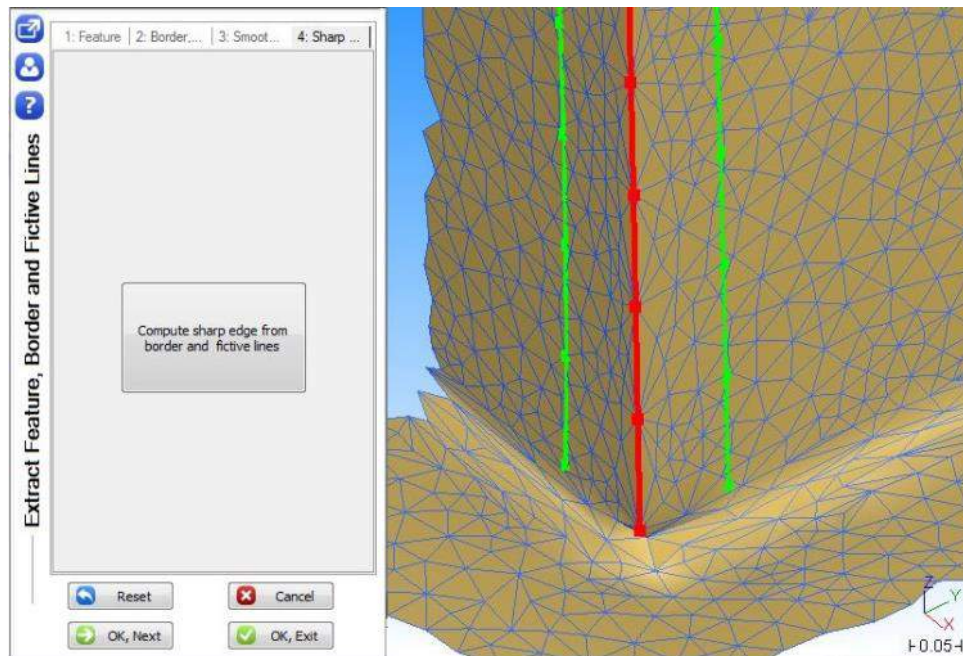


Figure 66: Rebuild the sharp edge

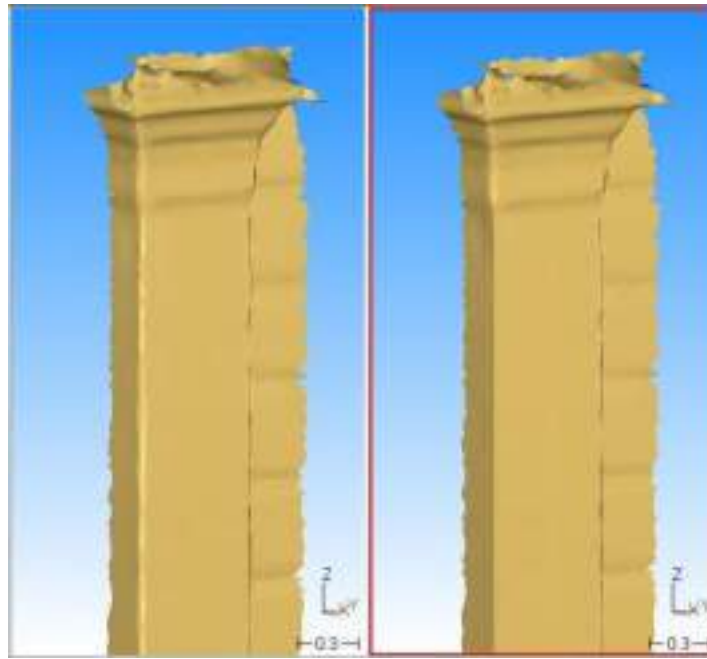


Figure 67: Compare the original edge with the reconstructed edge

You can try the Exercise 7 called “Meshing of a façade point cloud” (downloadable online), to see the reconstruction of a vault with the same command [Polylines\Feature Line](#). In this exercise you can also see how to rebuild a sharp edge in a different way. The fictive line is created by intersecting two planes with [Create\Planar intersection](#), and the edge is rebuilt with [Mesh\Sharp Edges](#).

Neutral axis

To work on tubular shapes, it can be very helpful to have access to their neutral axis, 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.

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

Open the file “NeutralAxis&Extrusion.rsh”. 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 [Polyline\Neutral axis](#).

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

Here click on “Automatic: Compute Line”. Check the option “Transparent mesh” in order to see the axis inside the mesh. Note that in the options there is a field called “Diameter” where the approximate diameter of the shape is given, regarding the computed line.

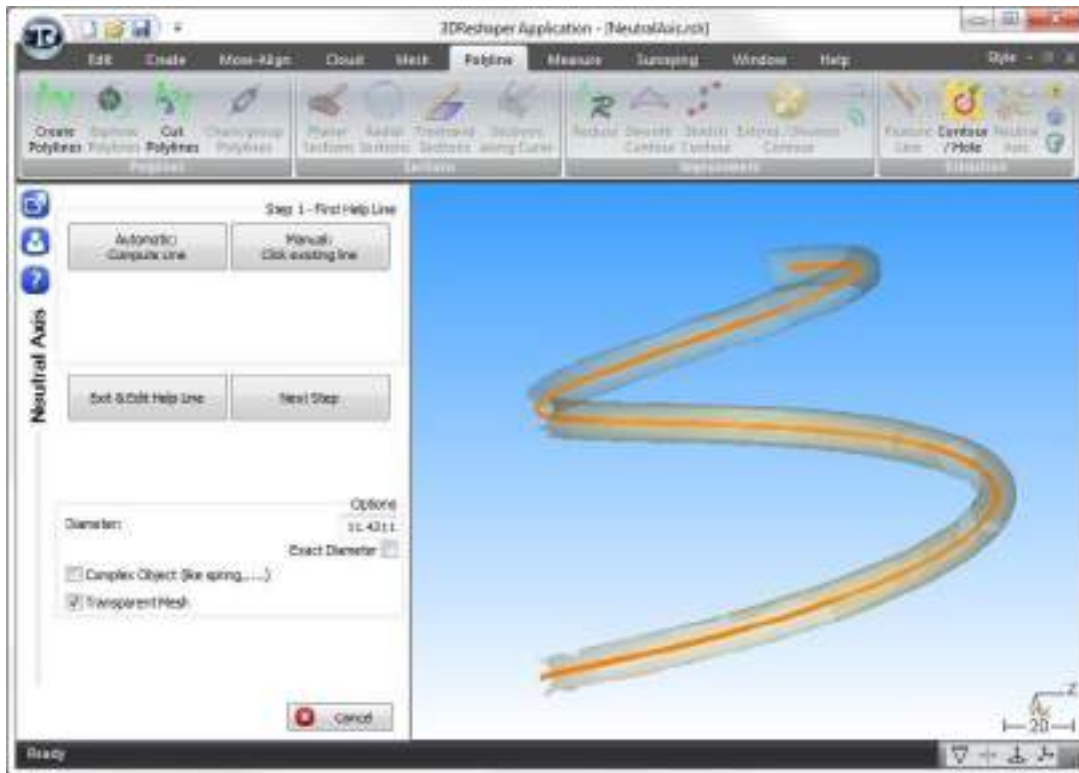


Figure 68: Compute first approximate axis of a mesh

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

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

Now try the computation on the **“spring cloud”**. Show only the point cloud, select it and go to [Polyline\Neutral axis](#). Compute the Help Line automatically. Here it is the central axis of the spring which is found, and not the inner axis. You can notice that the computed diameter is the external diameter of the spring. To find the neutral axis inside the wire of spring, you can enter the diameter of the fiber computed previously. Enter 11 in the field “Diameter” and click again on “Automatic: Compute Line”. For such a shape, in the second step where the precise neutral line is calculated, it is necessary to tick the “complex shape” option considering the fact that the expected neutral line has a revolute shape.

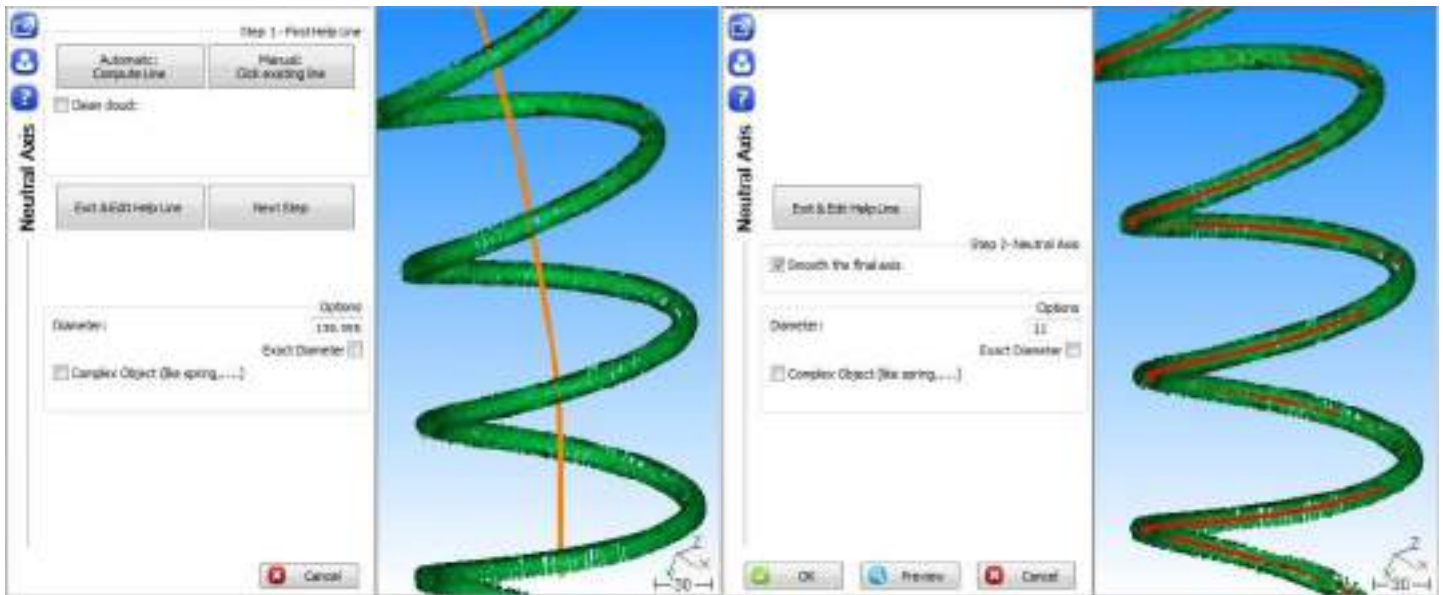


Figure 69: General and inner neutral axes of a spring

Note: When you select a point cloud, an option called “Clean cloud” is available in the first step of the neutral axis computation. It gives the possibility to select the points at a certain distance around the first Help Line, and to compute the axis with only the selected points.

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 Reconstruction” in the “Reconstruction part”

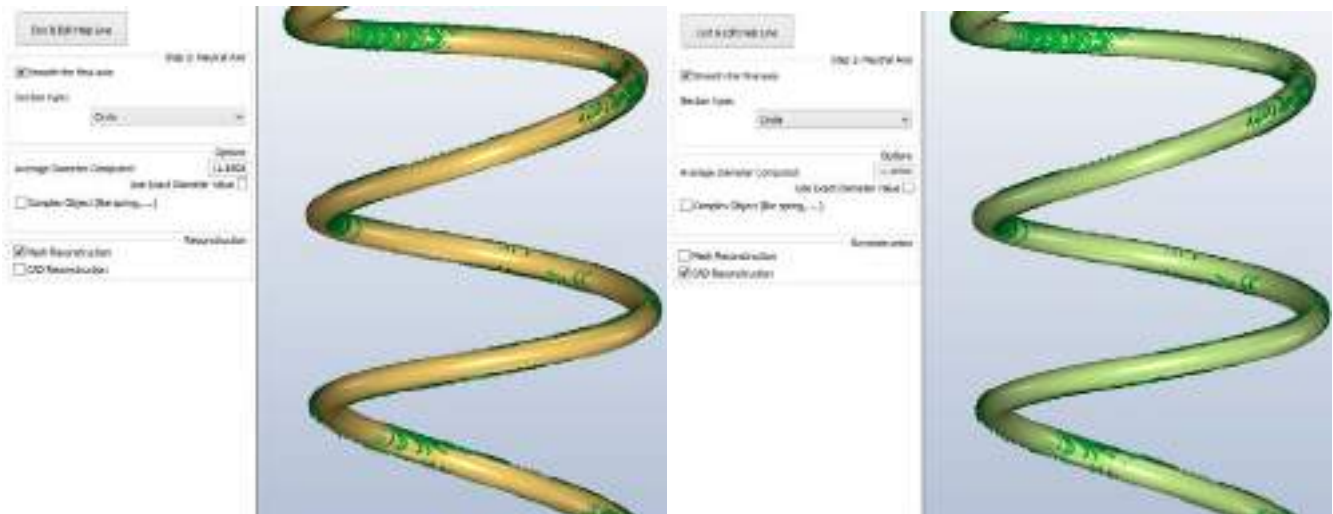


Figure 70: Mesh and CAD Reconstruction

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

- [Polyline\Join 2 polylines](#): to compute a curve to link the extremities of two polylines.
- [Polyline\Contour / Hole](#): to extract the lines of the free contours and the holes in a mesh.
- [Polyline\Intersect 2 meshes](#): to extract the intersection lines between two meshes.
- [Polyline\External contour](#): to compute the external contour of a mesh seen from a specific direction.

Several other tools are available in 3DReshaper to extract and manage polylines. They are situated in the menu Create:

- [Create\Intersection](#): to compute various intersections, between polylines, meshes and geometric shapes.
- [Create\Projection](#): to compute various projections, such as a point or a polyline on a plane or on a mesh.
- [Create\Planar intersection](#): to compute the intersection between a plane and a line, or two or three planes.

MEASUREMENT, INSPECTION AND REPORTING

Measure volumes

Exercise: Measure the volume of a closed mesh

Open the file “VolumeClosed.rsh”. It contains the meshes of two sections of a pipe. Select one mesh and go to [Measure\Volume](#). If the mesh is not closed, a window opens with a warning message saying “Warning! The volume(s) computed is (are) approximated because the mesh(es) is (are) not closed.” This message also invites you to see the plugin Surveying if you are interested in computing cubature.

The result is displayed in a label.

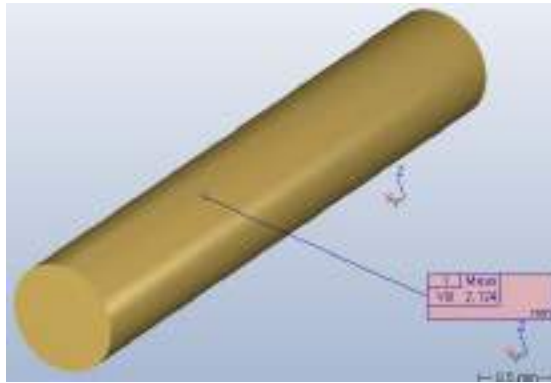


Figure 71: Volume of a closed mesh

It is possible to select several meshes before launching the command.

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

Note: When a mesh is closed, you can also see its volume using the “property” command. Right-click on the object in the tree or in the scene and click on “Properties”.

If you wish to compute cubature or volumes between two meshes, you can use [Surveying\Volume / Cubature](#) in the Surveying module. See the chapter Surveying Module for more details.

Make measurements with the mouse

In 3DReshaper there is a single tool in order to measure coordinates, distances, angles, etc. This command is [Measure\Measure with mouse](#).

Exercise: measurements on a mesh

Open the file “VolumeClosed.rsh” and **show only** the mesh “Straight Pipe” with a transparency of 60%. Then launch the command [Measure\Measure with mouse](#). Select only the option “Middle/Center” and click on the circle at the extremity of the cylinder. The center of each extremity is detected like on the Figure 71.

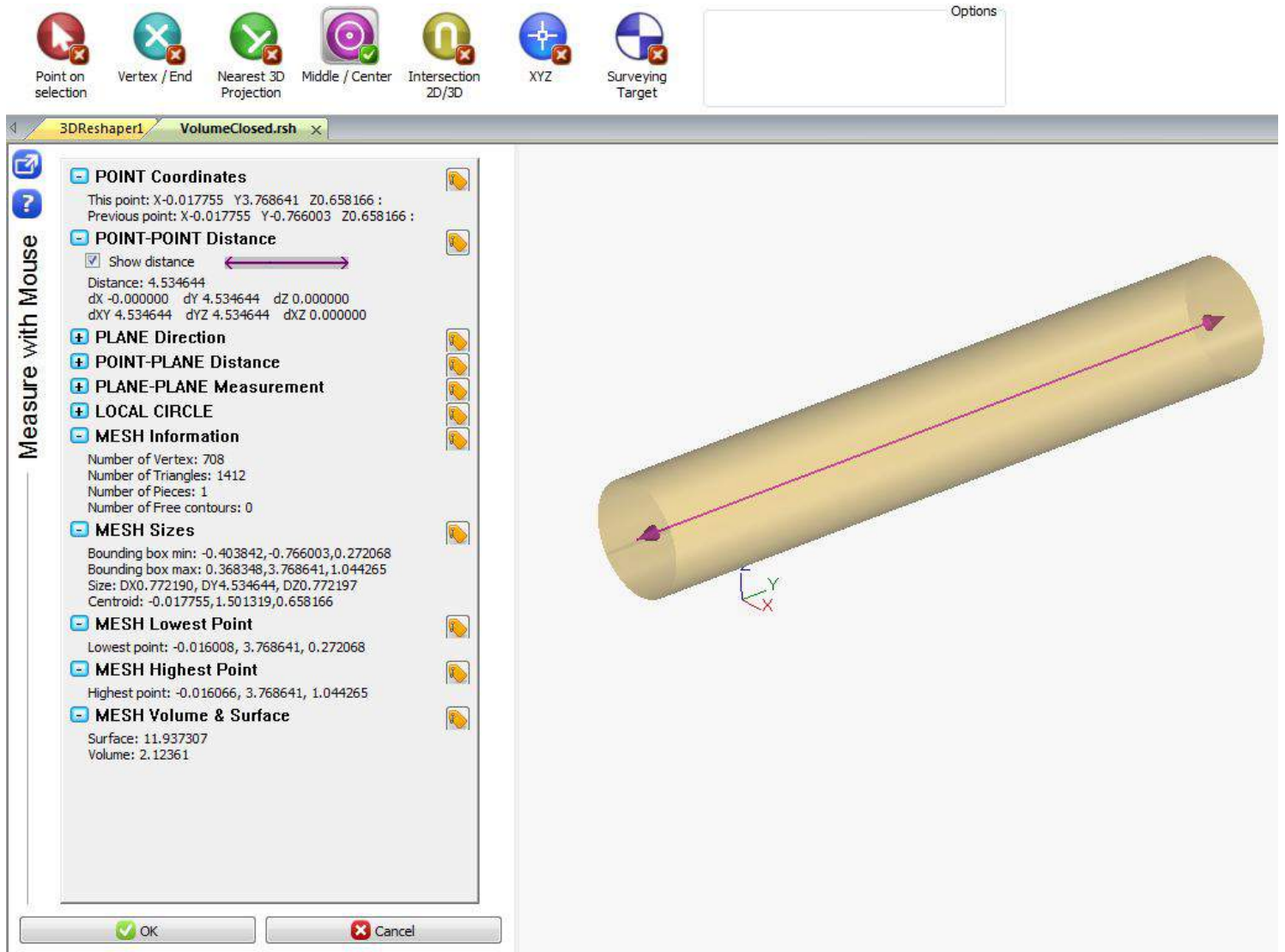


Figure 72: Measurement on a mesh

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

Near each measure information, there is a small button in order to create the label with the corresponding data. Thus, you can create a label per measure and associate a reference value to get the deviation error (cf. Figure 71). All these labels can be exported later in a report.

Geometric features

A lot of geometric features are available in 3DReshaper: Lines, Planes, Circles, Cylinders, Spheres, Cones, Round Slots and Square Slots.

Create a geometric feature

You can define a geometric feature with corresponding commands from the “Create” menu. For each command, you can click some points or enter characteristics in order to create the shape.

For example, go to [Create\Circle](#) and:

-
- Untick the option "Click points on scene"
- Tick “Fixed center” and enter X=10, Y=5, Z=0.
- Tick “Fixed axis” and enter X=0, Y=0, Z=1.
- Tick “Radius” option and enter 2.

Validate the circle, then, select it and go to [Measure\Measure with the mouse](#). Click “OK” to create the corresponding label and check the properties.

Compute best shapes

In 3DReshaper, you can compute a best shape from each kind of object (clouds, meshes, polylines, etc.).

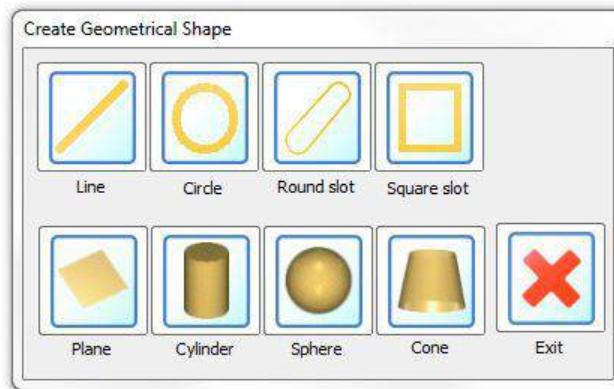


Figure 73: Geometric shapes available in 3DReshaper

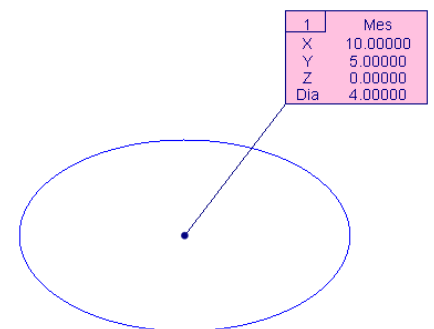
Exercise: Compute best shapes from clouds and polylines

Open the file “BestShape.rsh”.

To begin, select the polyline “Circle” and go to [Measure\Best Shape](#). Click the “Circle” button as we want to compute a circle. Select the calculation method “All on (Min Std Dev)” as we want to compute the best circle from all the points on the polyline. Adjust the slider at the middle of the dialog box to eliminate noisy points. Click the button “Label” in order to create a label and validate the circle.

Now select the cloud “Circle Inner Cloud” and go to [Measure\Best Shape](#). Click the “Circle” button as we want to compute a circle. Select the calculation method “Inner” as we want to compute the best circle corresponding to the hole and click “Preview”. Click the button “Label” in order to create a label and validate the circle.

Finally, select the cloud “Cylinder” and go to [Measure\Best Shape](#). Click the “Cylinder”



button as we want to compute a cylinder. Select the option “Fix radius/diameter” and set the radius to “8.25” and select also the option "Fixed axis" and Z direction for the normal direction as we want to add some constraints. Adjust the slider at the middle of the dialog box to eliminate noisy points. Click the button “Label” in order to create a label and validate the cylinder.

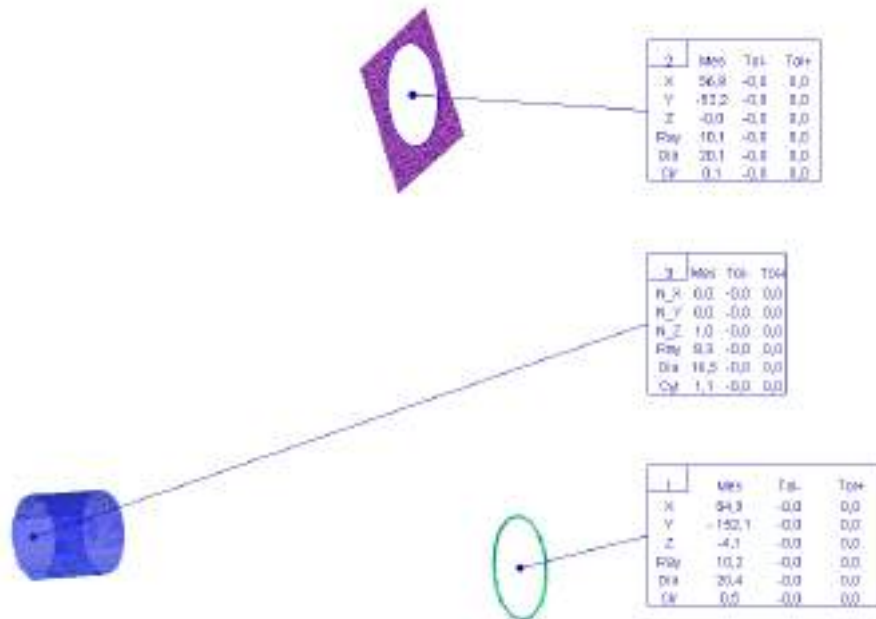


Figure 74: Best shapes computed from point clouds or polylines

Extract geometric features

Unlike the command [Measure\Best Shape](#), the command [Measure\Extract Shape](#) does not compute the geometric feature from the entire object; it works locally on the object, in other words it extracts only the points that fit with the entity that we are looking for.

There are two kinds of extraction:

- Classical extraction: requires a nominal feature.
- Quick extraction: requires clicking point(s) on the object.

Exercise: Extract a local plane from a point cloud and a reference plane

Open the file “RPSOnRef.rsh” and show only the plane (in the Geometric Group) and the cloud “CloudAlreadyAligned” in the “Result” folder.

Select both the plane and the cloud and go to [Measure\Extract Shape](#).

Disable the two options about clipping and click “Preview”. As you can see, the extracted plane is not the one we want because it has been computed from the entire cloud. This is the reason why we have to enable clipping options. Enable the tangential clipping and enter 0. Enable the normal clipping and enter 1. Click “Preview” again, the plane is now good. Click “OK, Exit” to validate.

Now show only the “Circle to extract” (in the Geometric Group) and the cloud “CloudAlreadyAligned” in the Result group. You can select the circle and right click on “zoom on” to automatically focus on the circle. Select these both objects and go to [Measure\Extract Shape](#). Select the option “Inner” as the circle we want to extract corresponds to a hole.

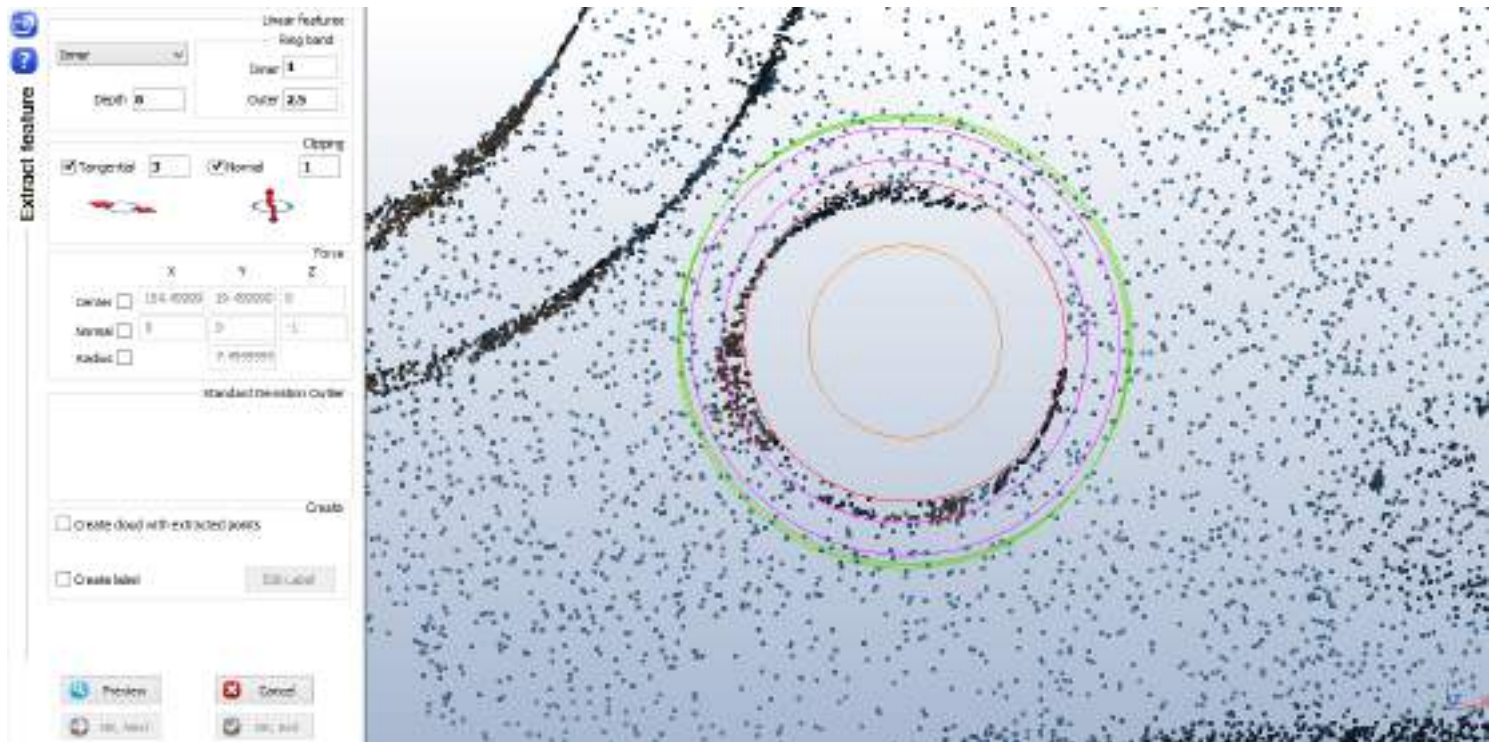


Figure 75: Clipping and Ring band for circle extraction

On the Figure 74 you can see the different circles temporary displayed in order to visualize the ring band and the clipping:

- Pink circles represent the ring band: it is the area where points are taken in order to compute the plane where the extracted circle will be.
- Orange circles represent the tangential clipping: points outside this area will be ignored.
- Green circles represent the nominal clipping: points outside this area will be ignored.
- Red circle represents the circle selected with the cloud when entering in the command.

Set the same parameters as in the Figure 74 and click “Preview” to visualize the result: you should see immediately extracted points and measured circle. Note that the nominal circle (in red) is quite far from the real extracted one (in blue).

Click “OK, Exit” to validate.

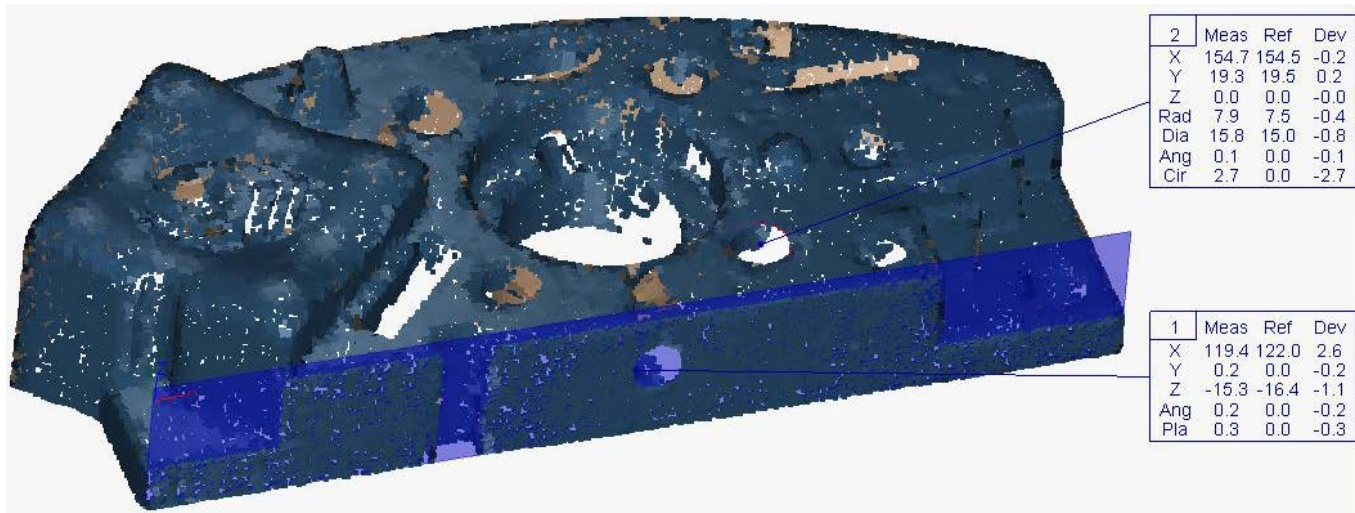


Figure 76: Circle and plane extracted from the cloud

Note: If you select only the cloud, you will have to define the geometric feature in a first step. To do this, just click some points in order to create a shape similar to the one you want to extract.

Note: In order to take into account a chamfer, a depth parameter can be used in order to measure the circle radius at a different height than the plane on which is measured the circle.

Exercise: Extract planar contours from a point cloud

Open the file “RPSOnRef.rsh” and show only the cloud “CloudAlreadyAligned” in the “Result” folder. Go to [Measure\Extract Shape](#) and click the button “Quick Plane”.

Unlike the previous method, with the “Quick Plane” option you do not need a reference feature, but you have to click point(s) on the cloud to extract the shape.

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

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

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



Figure 77: Planar contours extracted from a point cloud

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 3DReshaper you can compare:

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

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

Compare polylines

Exercise: Compute inspection between polylines

Open the file “NeutralAxis&Extrusion.rsh”. Select and show only the two polylines “Profile” and “Section to Inspect”. Select first “Profile” as it is the reference, and then select “Section to Inspect” with the “CTRL” key as it is the measure. Launch the command [Measure\Compare / Inspect](#).

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

Click “Preview” to compute the inspection, then click on “Edit colors” to magnify the distances and change thresholds and colors (see [Adjust inspection colors](#) paragraph for more information). Click “OK” to validate the display. Then click “Ok” again to validate the inspection. A new object called “Compare Profile / Section to compare 1” is added in the Measure Group.

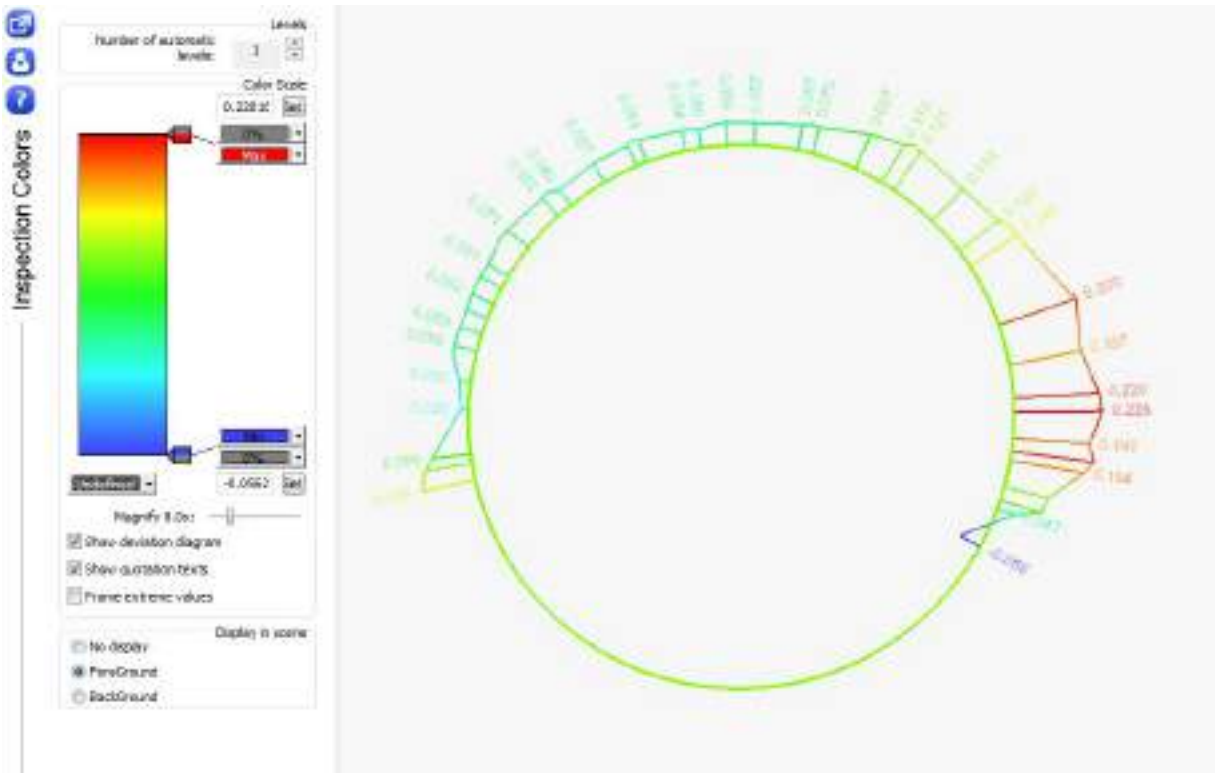


Figure 78: Inspection between two polylines

Note: If you select two sets of polylines, 3DReshaper will automatically create pairs of contours and then compute several inspections of two polylines.

Exercise: Compute inspection between a surface and a cloud

Compare clouds, meshes & surfaces

Open the file “CompareInspect.rsh”, then show only the cloud “Measure in millimeters” and the CAD “Reference in millimeters”. Then select the CAD **first** as it is the reference, and then with the “CTRL” key select the cloud, and go to [Measure\Compare / Inspect](#).

Choose to apply the color on the reference and select the “3D Inspection” method. You can unselect the options in order to ignore particular points (points too far from the CAD object or projected on edges).

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

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

A new object called “Compare Reference in millimeters / Measures in millimeters 1” has been added in the CAD Group.

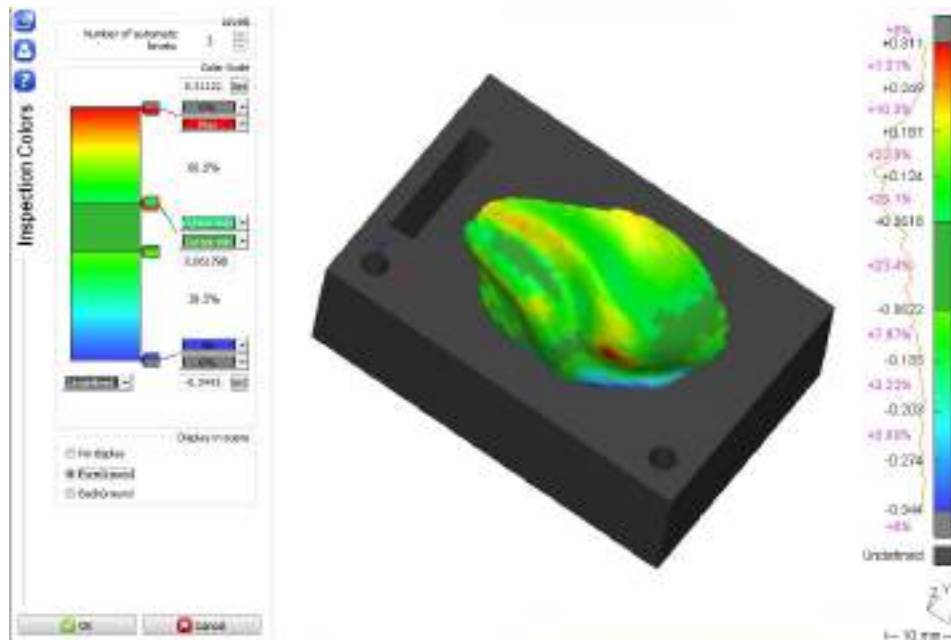


Figure 79: Comparison between a cloud and a CAD

Adjust inspection colors

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

- A result of an inspection
- A cloud with intensity values
- A mesh where curvatures have been highlighted ([Mesh\Show curvature on mesh](#))
- A cloud extracted from [Measure\Best Shape](#) or [Measure\Extract Shape](#)
- A cloud colored along a direction

Keep the file “CompareInspect.rsh” open. Select the result you have computed in the previous paragraph and go to [Measure>Edit Colors](#).

As you can see in the result of the previous exercise, the coloring has been automatically divided into 3 parts because the range of the result was found nearly symmetrical around zero:

- The dark green central zone is the area where the deviation is low and could be interpreted as the “good” zone of the part. Normally, the values on the cursors should be adjusted to meet the fixed tolerance.
- The zone above the central zone is the area “over” tolerance.
- The zone below the central zone is the area “under” tolerance.

You can easily change this representation by setting the “number of automatic levels” to one.

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

- Change colors on both sides of the cursor with button “Cursor max” and “Cursor min”.
- Change the cursor value (field just below “Cursor min”).
- Remove it by pressing “DEL” on the keyboard.

Note: Each cursor has two colors:

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

Try to reproduce the color scale from the Figure 79. You have to:

- Set the number of levels to 5.
- Set the max value to 0.3 and the min value to -0.3.
- Set other cursors to 0.18, 0.05, -0.05 and -0.18.
- Change colors above the cursor “0.18” and below the cursor “-0.18”.
- Select the option “Foreground” in order to display the color scale in the 3D scene.

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

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

Click “OK” to validate.

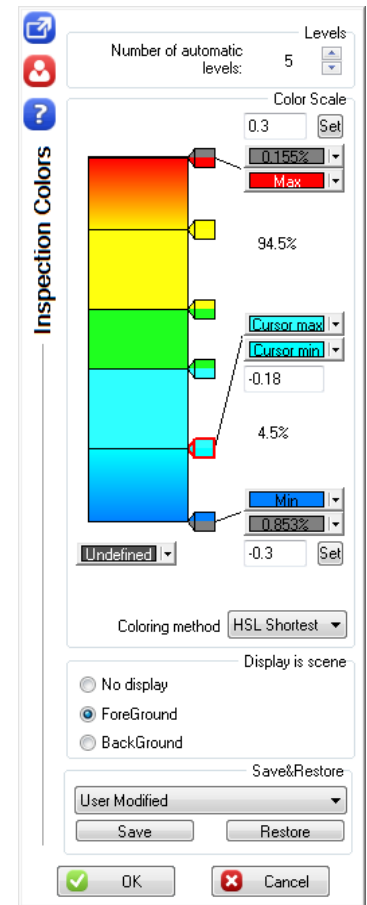


Figure 80: Edit color mapping

Labels & Reporting

Exercise: Create a complete report from a 3D inspection

Open the file “CompareInspect.rsh” and show only the color mapping “Comparison in millimeters” located in the Measure Group.

Create Labels

First launch the command [Measure\Label Aspect](#) in order to customize labels aspect. Select the label type “long” and the option “Take automatically a smaller label if necessary”. Then, click “OK”.

Now launch the command [Measure\Create / Edit Label](#). Set the high tolerance to 0.2 and the low tolerance to -0.2. Then select the option “Point on Selection” in the 3DReshaper ribbon and click some points on the color mapping. Click for example points with different colors (red, light blue, dark blue, green, yellow...).

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

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

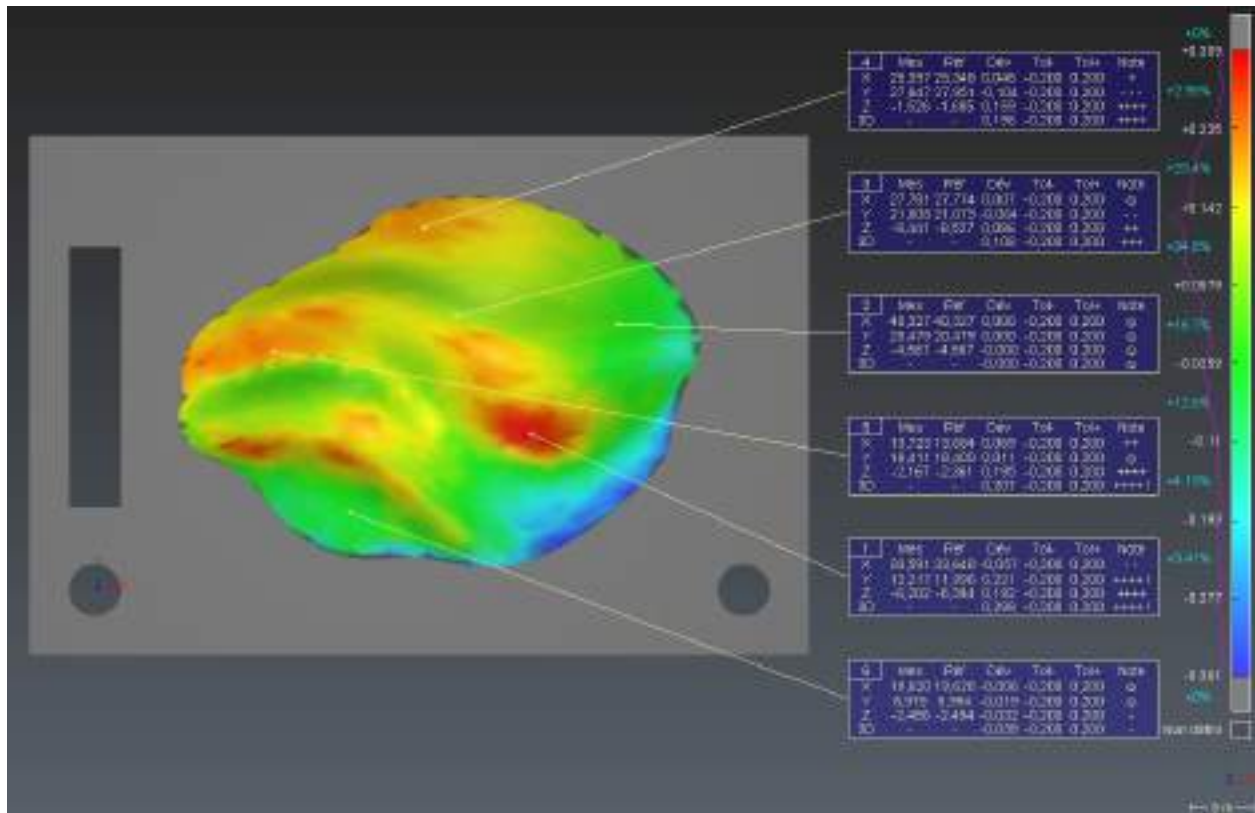


Figure 81: Labels created from the 3D inspection

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

Edit Labels

Select the label(s) you want to edit and launch the command [Measure>Create / Edit Label](#). You will be able to change tolerance, comments, etc.

You can also set which elements you want to show in the label thanks to the check boxes.

Note: See the Help files to have more details about all the information you can display in a label.

Create View Sets

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

Keep the file “CompareInspect.rsh” open. Show only the color mapping and one label and then go to [Window>Create View Set](#). Enter a name and select the option to store the visibility. Click “OK” to validate.

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

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

Customize and export a report

Keep the file “CompareInspect.rsh” open. Once you have created labels and/or view sets, you can edit a report.

Go to [Measure\Label Report](#):

- Select the option “All labels”.
- Click the button “Select ViewSet” and then select all the items and click “OK”.
- Click the button “Setup...”, select the “Landscape” orientation, answer the questions on the bottom of the dialog box and click “OK”.
- Click the “Preview” button.

You will have a report with all the labels at the beginning and all the views at the end.

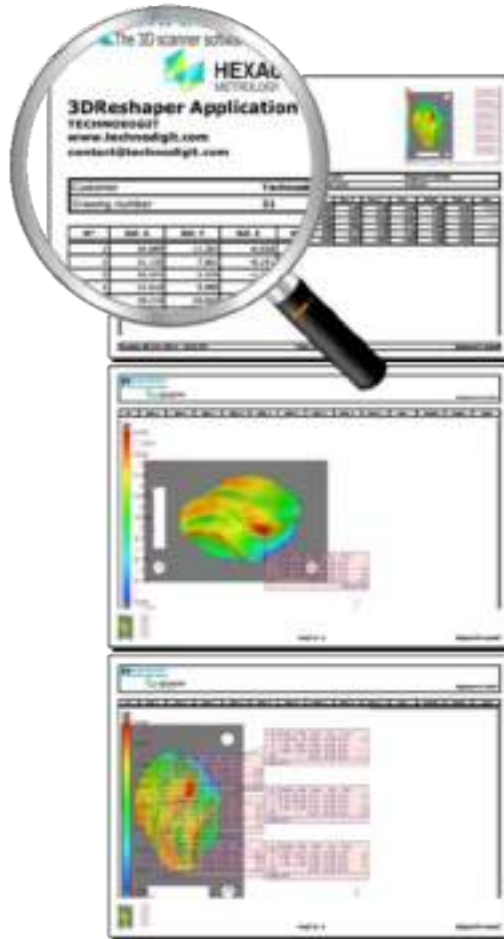


Figure 82: 3D Inspection report

You can print this report as a PDF file if you have a PDF printer installed on your machine. You can also export it as an HTML or CSV (excel) file.

Finally, you can custom this report by changing headers and footer (more information on our forum: <http://technodigit.com/forum> : [Report customization](#))

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