

## Point Cloud to Mesh Tutorial

---

### Objective

Learn how to create a qualified polygonal model from point clouds. Use the Point Cloud to Mesh function to triangulate point clouds. Use the Fill Hole and Unify Normal functions to repair meshes. Use the Decimation and Smoothing functions to reduce the number of triangles and increase the smoothness of meshes while keeping an accurate geometric shape. Introduce a few mesh editing tools to fix irregular boundaries.

### Import File


Click the **Import** command from the File menu. Choose the **Point Cloud File** type (\*.txt, \*.asc, \*.xyz) displayed in the Files of Type box. Highlight a file to open by choosing one of the format options that describes the data structure. The imported file name is shown in the Object Tree Window on the right side of the screen. Clicking an object in the tree window will select the object for use.

### Step 1: Point Cloud Processing

It may be necessary to process the point cloud data before triangulation. For example, use the **Denoise Point Cloud** command to remove noise from the point cloud, which can speed up the triangulation process. Use the **Decimate Point Cloud** command or the **Remove Redundant Points** command to reduce the size of point cloud.

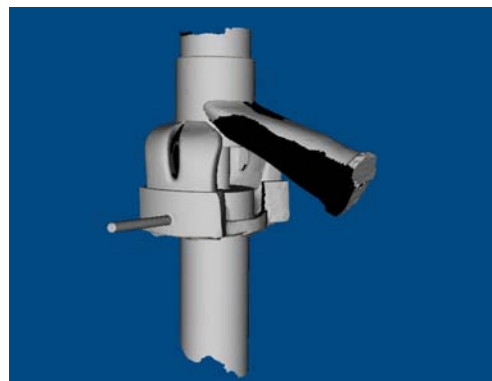
*Note: The **Remove Redundant Points** command removes redundant points according to a defined minimum distance and curvature weight. It makes a more uniform distribution of points on flat regions than the **Decimate Point Cloud** command does, but it takes a slightly longer time.*

### Step 2: Triangulate Point Clouds

Click the **Point Cloud to Mesh** command (*Cloud > Point Cloud to Mesh*), or click the icon  in the Toolbar. Specify the noise level of the points in the Parameter Window, and press the **[Apply]** button to start the triangulation process. When finished, you will get an accurate meshed model.



Original point clouds




Triangle meshes

### Step 3: Repair Meshes


After triangulation, the object appears to have some dark areas. You may turn on two-sided lighting of surface using the shortcut **[Ctrl] + L**. The total part becomes visible. The flipped triangles are displayed in gray as well. You can then press **[Ctrl] + L** again to turn off the two-sided lighting. VRMesh uses single light in default to speed up the surface display.

The suggested repair steps are:


#### 1. Extract floating parts *(Optional)*

Click the **Extract All Parts** command (*Modify > Extract All Parts*), or click the icon  in the Toolbar. Specify the **Minimum Triangles**. The program removes all parts that contain fewer triangles than the defined amount.

#### 2. Fill small holes

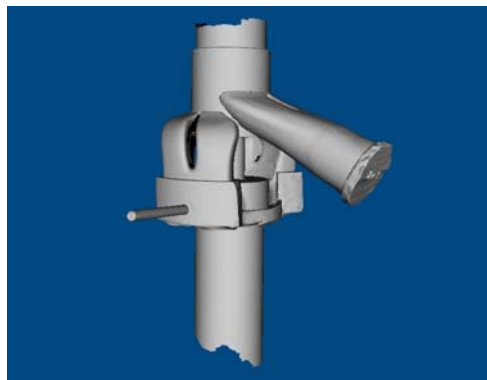
Click the **Fill Holes** command (*Modify > Fill Holes*), or click the icon  in the Toolbar. Specify the **Hole Radius**. Generally, it's not necessary to check the **Fill by Curvature** option when filling small holes. Please note that a red circle will appear in the drawing window indicating the defined maximum area that a hole will be filled. You can drag the mouse across the text field and thus slide the value. Please note that filling small holes before unifying normals is an important step to produce a better result.

#### 3. Unify normals

Click the **Unify Normals** command (*Modify > Unify Normals*), or click the icon  in the Toolbar. Select the **Fix Errors – In Depth** checkbox. The program will detect and delete ill-connected triangles while unifying the normals.


#### 4. Fill holes again


Click the **Fill Holes** command (*Modify > Fill Holes*), or click the icon  in the Toolbar. Adjust the value of **Hole Radius** to fill remaining holes.



Mesh repair

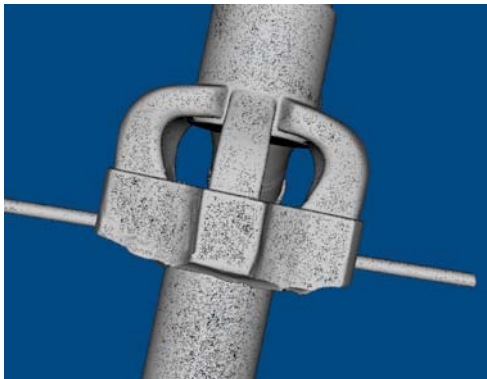
For big holes, you can fill them individually.

First step, click the **Pick Boundary** command (*Curve > Surface Curve > Pick Boundary*), or click the icon  in the Toolbar. And then, click near a preferred hole to generate a boundary curve.

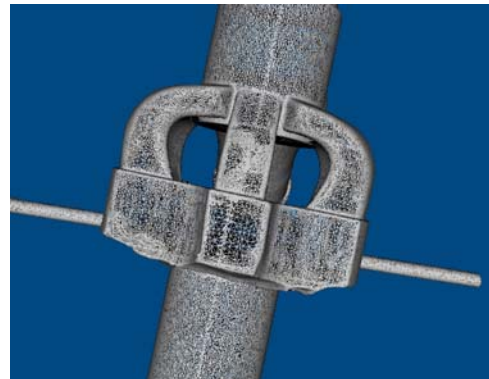
Second step, click the **Fill Boundary by Curvature** command (*Modify > Fill Boundary > Fill Boundary by Curvature*), or click the icon  in the Toolbar. The program fills the hole smoothly using the curvature-based method.

#### Step 4: Decimation

Click the **Decimation** command (*Modify > Decimation*), or click the icon  in the Toolbar. Select the **Quadric** type, specify the **Decimation Percentage**, and press the **[Apply]** button to reduce the number of triangles.




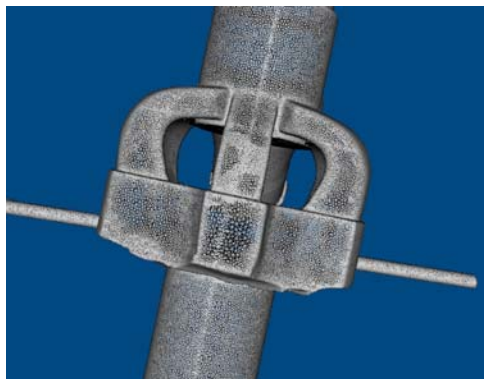
927,019 triangles



Decimated 70% off

#### Step 5: Smoothing


Click the **Smoothing** command (*Modify > Smoothing*), or click the icon  in the Toolbar. Select the **Remesh** smoothing type, specify the **Smoothing Iteration** at around 12, and press the **[Apply]** button to remove noises from the meshes. Please note that the preservation of important surface features such as sharp edges and corners is controlled by the **Smoothing Weight** parameter. This value defines the degree of the smoothing for sharp edges.

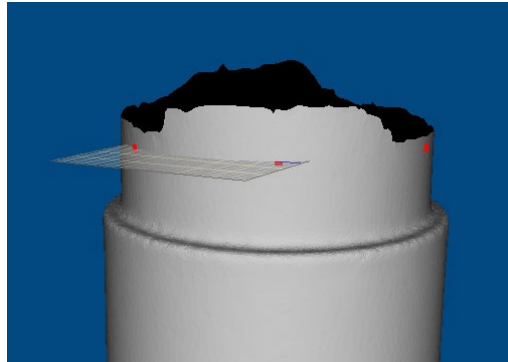



Remesh smoothing

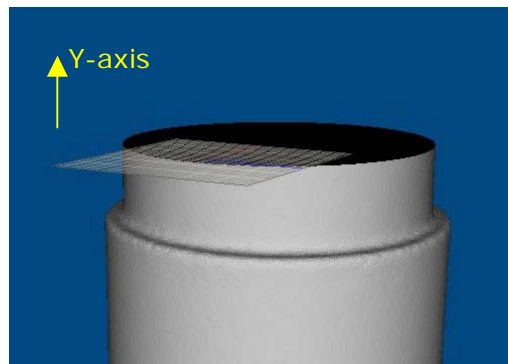
## Step 6: Edit Meshes


VRMesh provides users comprehensive mesh editing tools. In this example, you will learn how to fix an irregular boundary.

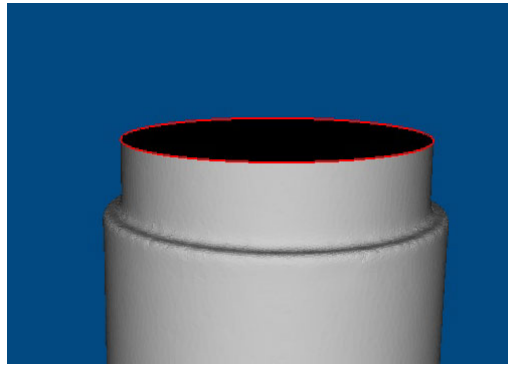
1. Align work plane. Click the **Align Plane to Object** command (*Transform > Align Plane to Object*), or click the icon  in the Toolbar. Place three dots on the surface near the irregular boundary. Specify the **Parallel** type to place the plane direction along the surface normal direction, and press the **[Apply]** button to align the work plane, as shown below.




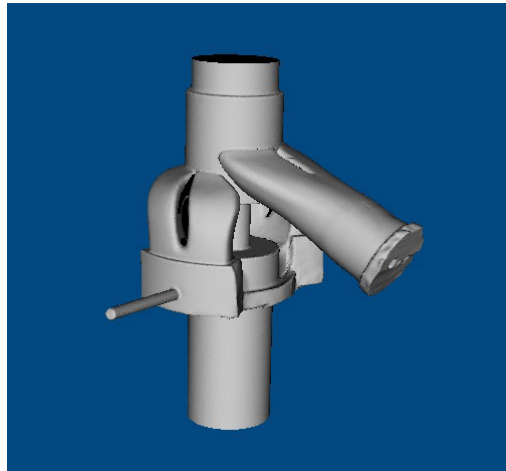
2. Clip meshes. Click the **Clip by Plane** command (*Modify > Knife > Clip by Plane*), or click the icon  in the Toolbar. Select the **Work Plane** type, specify **Align to Y-axis**, and press the **[Apply]** button. The program clips the meshes along the normal direction of the work plane, as shown below.



3. Create a boundary curve. Click the **Pick Boundary** command (*Curve > Surface Curve > Pick Boundary*), or click the icon  in the Toolbar. And then, click near the boundary to generate a boundary curve, as shown below.



4. Extrude the boundary. Click the **Extrude Boundary along Normal** command (*Modify > Extrude > Extrude Boundary along Normal*), or click the icon  in the Toolbar. Select the **Contour** as the extrusion direction, specify the **Extrusion Length**, and press the **[Apply]** button. The program extrudes the boundary along the specified normal direction. The final model is shown below.



**End of Tutorial**