VRMesh Triangulation for AutoCAD 2014

User’s Guide
# Table of Contents

## GETTING STARTED

1. **Product Overview** ................................................................. 1

2. **Command Wizard** ................................................................. 2

3. **Workflow Tutorials** ............................................................... 4
   - Tutorial 1: Point Cloud Decimation ........................................ 4
   - Tutorial 2: Point Cloud Triangulation .................................... 7
   - Tutorial 3: Cross Section Generation .................................... 11
   - Tutorial 4: Large Dataset Processing ................................... 13

## TOOL REFERENCE

1. **Remove Isolated Points** ...................................................... 17

2. **Decimate Point Cloud** ......................................................... 17

3. **Denoise Point Cloud** ............................................................ 17

4. **Point Cloud to Mesh** ........................................................... 18

5. **Remove Floating Parts** ....................................................... 18

6. **Decimate Triangles** ............................................................. 18

7. **Seam Gaps** ......................................................................... 19

8. **Fill Holes** ........................................................................... 19

9. **Fill Fjords** .......................................................................... 19

10. **Unify Normals** ................................................................... 20

11. **Fix Errors** .......................................................................... 20
12. Remesh Smoothing ................................................................. 20
13. Reverse Surface .......................................................................... 21
14. Cross Section ............................................................................. 21
15. Intersection ............................................................................... 22
16. Polyline ...................................................................................... 22
17. Spline ........................................................................................ 23
18. Reset Points ............................................................................... 23
Getting Started

1. Product Overview

VRMesh Triangulation for AutoCAD 2014 is a powerful point cloud and mesh processing software that allows you to wrap point cloud data into accurate triangle meshes within the AutoCAD environment. It delivers best-in-class point cloud clean-up and triangulation functionalities for users to generate high quality meshes. It also provides the ability to create cross-section curves from triangle meshes and point clouds.

Features and Benefits:

- Advanced point cloud decimation
- Accurate triangulation
- Efficient denoising with edge-preserving capability
- Cross-section curve generation
- No limit concerning the point cloud and mesh size
- Compatible with AutoCAD 2014 and its family products (Architecture, Civil 3D, Map 3D...)

Command List:

- **VRMeshTriangulation**: Provide a command wizard consisting of a number of processing steps which are grouped logically for point cloud and mesh processing. The program will execute the commands you choose step-by-step.
- **VRMeshSection**: Create a cross-section curve from a mesh/point cloud, or clip a mesh.
- **VRMeshIntersection**: Find the line of intersection between two planes.
- **VRMeshPolyline**: Create a 3D polyline on a mesh/point cloud surface.
- **VRMeshSpline**: Create a 3D spline on a mesh/point cloud surface.
- **VRMeshReset**: Clear the filter added to a point cloud entity.
2. Command Wizard

The **Triangulation** command opens up an interface where the functions converting point clouds to meshes are grouped logically. The program will execute the commands you choose step-by-step.

![VRMesh Triangulation v1.0 (Registered)](image)

**Steps:**

1. Specify the data file you want to process. The **Displayed Data** option allows you to process the current dataset displayed in the AutoCAD drawing window. The **Data in File** option allows you to process all data in a file.

2. Select the commands you want to execute. Click on the right side button to open up the parameter setting panel of each corresponding command.

3. Press the [Batch] button and click on the preferred object in the AutoCAD drawing window to execute the selected commands. The result will be automatically saved as a *.pcp file (VRMesh index file) to the original file folder. If the generated mesh
contains lower than 500,000 triangles, it will be automatically shown in the drawing window.

**Personalize Settings**

The **Personalize** menu bar is located in the upper right corner of the command wizard. The menu bar contains the following submenus and commands:

- **Max. Loading Points**: This option defines the maximum number of points that be allowed to load into the physical memory on your computer for computation.
- **Ignore Color Information**: This checkbox allows you to hide the vertex color of an object to speed up the display in the AutoCAD drawing window.
- **Export Index File**: This command allows you to export a VRMesh indexed *.pcp file to a *.pts or *.stl file.
- **Parameter Setting**: The **Load Parameter Setting** command allows you to import a *.PAR parameter file and updates all the parameters automatically. The **Save Parameter Setting** command saves current parameter values into a *.PAR file. The default file is `VRMeshTriangulationV10DefaultParameters.PAR` in the subfolder named “Script” under your installation directory.
- **Display**: This command allows you to change the size of text shown on the command wizard.
- **License**: The **Activate License** command allows you to activate your license online by entering the License ID and Password, or manually activate your license by sending us activation codes. The **Return License Online** command allows you to transfer a licensed file from one computer to another over the Internet. Please note that the manually activated license isn’t allowed to be transferred from one computer to another. You may choose to use a VRMesh USB dongle to manage the license.
- **Help**: This command opens up the help file.
3. Workflow Tutorials

Tutorial 1: Point Cloud Decimation

VRMesh Triangulation provides two ways to process a data file. One way is to process the data displayed in the AutoCAD drawing window. The other way is to process all data in a file.

In this tutorial we will show you how to process the data displayed in the drawing window.

**Step 1:** Attach your point cloud file into the AutoCAD drawing window. The sample file below has around 1.5 million points shown in the window. We are going to reduce the amount of displayed point clouds to 60,000 points.

![Image of AutoCAD drawing window with point cloud](image)

**Step 2:** Click VRMesh in the menu bar and press the Triangulation icon to open up a command wizard.
**Step 3:** Select the *Displayed Data* option to allow the process to be executed on the displayed data. Then, select the *Decimate Point Cloud* command and open up the parameter setting panel. Choose the *Uniform* distribution type, set the number of remaining points to 60,000, and set the value of *Curvature Weight* to 0.4 (typical value: 0.2 ~ 0.5) for this example.

![Decimate Point Cloud Parameter Panel](image1.png)

**Step 4:** Press the [Batch] button in the wizard and click on the desired object in the AutoCAD drawing window to execute the command. The decimated point cloud will be shown in the window.

![Decimated Point Cloud](image2.png)
The Decimate Point Cloud command in VRMesh not only reduces the number of points, but also maintains an accurate representation of the point cloud. The simplified point cloud has significantly enhanced the visualization capability of the original file.

**Note:**

- You may use the Reset Points command to clear the filter added to a point cloud entity.
Tutorial 2: Point Cloud Triangulation

VRMesh Triangulation provides two ways to process a data file. One way is to process the data displayed in the AutoCAD drawing window. The other way is to process all data in a file.

In this tutorial we will show you how to process the data displayed in the drawing window.

**Step 1:** Attach your point cloud file into the AutoCAD drawing window. The sample file below has around 1.5 million points shown in the window.

![Point Cloud in AutoCAD](image)

**Step 2:** Click VRMesh in the menu bar and press the Triangulation icon to open up a command wizard.

![Triangulation Icon](image)
Step 3: Select the *Displayed Data* option to allow the process to be executed on the displayed data.

![VRMesh Triangulation v1.0 (Registered)](image)

Step 4: Select the commands you want to execute. Press one of the right side buttons to open up the parameter setting panel, and adjust parameters of each corresponding command as shown below.

- **Decimate Point Cloud**

  Choose the *Uniform* distribution type, which makes more uniform distribution of points on flat regions than the "Non-uniform" type. Set the number of *Remaining Points* to 80,000, which will generate around 1.6 million triangles. Set the *Curvature Weight* to 0.4. The higher the curvature weight, the more points will be kept around edges.

- **Denoise Point Cloud**

  The *Smoothing Weight* specifies smoothing constraints for sharp edges and corners. The higher this value, the less sharp edges and corners will be affected. The *Smoothing Iteration* defines the number of iterations for smoothing. Depending on how much noise is present in the point clouds, this step may not be necessary.
• **Point Cloud to Mesh:**

This step automatically converts point clouds to triangle meshes. No specific parameters are needed.

• **Remove Floating Parts**

This step deletes small sub-objects according to the connection of meshes. The *Min. Number of Triangles* defines the minimum number of triangles required for a valid part.

• **Decimate Triangles**

This step reduces the number of triangles while maintaining a good approximation to the original geometry. We set the remaining triangles to around 200,000 in this example.
**Step 5:** Press the [Batch] button in the wizard and select the point cloud object in the AutoCAD drawing window to execute the commands. The final triangle mesh will be loaded into the window as a new mesh entity.
Tutorial 3: Cross Section Generation

**Step 1:** Press the Cross Section icon and select a mesh in the drawing window to open up a control panel for generating cross-section curves.

**Step 2:** Adjust the parameter settings in the control panel. There are two operations available. The *Create Curve* option allows you to create a cross-section curve from a mesh or point cloud. The *Clip Mesh* option allows you to split a mesh in two pieces along a specified position. In this example we choose the *Create Curve* option to show you how to create cross-section curves from a mesh.

There are two ways to define the normal direction of the clipping plane. The *Align to Axis* option allows you to set the clipping plane to a specific position constrained to the X, Y, or Z axis. The *Align to Points* option allows you to select three or two points on the mesh to place the clipping plane. You may shift the clipping plane along a defined normal direction.

The *Adjust Direction* option allows you to further adjust the normal direction of the clipping plane.

The *Simplify Curve* option allows you to reduce vertices from a cross-section curve. The smaller the value of *Tolerance*, the more vertices are kept on a curve. You may select the *Square Up* checkbox to refine all corners to 90 degree.

The *Curve Color* square allows you to open a color palette to choose a displaying color for a cross-section curve shown in the drawing window.
Step 3: Press the [Apply] button in the control panel to create a cross-section curve on a preferred position.
Tutorial 4: Large Dataset Processing

VRMesh Triangulation provides two ways to process a data file. One way is to process the data displayed in the AutoCAD drawing window. The other way is to process all data in a file.

In this tutorial we will show you how to process all data in a file.

**Step 1:** Click VRMesh in the AutoCAD menu bar and press the Triangulation icon to open up a command wizard.

**Step 2:** Select the **Data in File** option to specify the data you want to process. The program will create an index file (*.pcp) to reorganize a large dataset into small blocks.

**Notes:**

- You may click the Personalize menu bar in the upper right corner to define the maximum number of points that be allowed to load into the physical memory on your computer. The default value is set to 4 million, which means a dataset containing below 4 million points will be fully loaded into physical memory for processing. Otherwise, the program will read each block from the index file and process it one by one.

- Clicking on the [PCP File Info] button will show the number of points and triangles in the selected *.pcp file.

**Step 3:** Convert Point Clouds to Meshes.

- **Decimate Point Cloud**

Choose the *Uniform* distribution type, which makes more uniform distribution of points on flat regions than the "*Non-uniform*" type. Set the number of *Remaining Points* to 3.
million, which will generate around 6 million triangles. Set the Curvature Weight to 0.3. The higher the curvature weight, the more points will be kept around edges.

- **Denoise Point Cloud**

  The Smoothing Weight specifies smoothing constraints for sharp edges and corners. The higher this value, the less sharp edges and corners will be affected. The Smoothing Iteration defines the number of iterations for smoothing. Depending on how much noise is present in the point clouds, this step may not be necessary.

- **Point Cloud to Mesh**

  This step automatically converts point clouds to triangle meshes. No specific parameters are needed.

- **Remove Floating Parts**

  This step deletes small sub-objects according to the connection of meshes. The Min. Number of Triangles defines the minimum number of triangles required for a valid part.
• Decimate Triangles

This step reduces the number of triangles while maintaining a good approximation to the original geometry. We set the remaining triangles to around 200,000 in this example.

![Decimate Triangles](image)

**Step 4:** Press the [Batch] button to execute the selected commands. The result will be automatically shown in the AutoCAD drawing window and be saved as a *.pcp file to the original file folder.

![AutoCAD drawing](image)

**Notes:**

• If the generated mesh contains lower than 500,000 triangles, it will be automatically shown in the AutoCAD drawing window. If the mesh has a bigger size, the program will export it as a *.pcp file to the original folder, and automatically update the “Open from” directory as shown below. The number of triangles in the PCP file will be shown at the bottom of the command wizard by clicking on the [PCP File Info] button.
- You may choose to export the mesh result as a *.stl file by selecting the Export Index File command in the Personalize menu bar. Or you may decimate the mesh again to an appropriate size for loading it into the drawing window by using the Decimate Triangles command in the wizard.

- You may use the Export Index File command in the Personalize menu bar to export point clouds as a *.pts file after point cloud processing, for example point decimation and denoising.
1. **Remove Isolated Points**

The Remove Isolated Points command deletes isolated points, which are separated from the majority of the points over a defined distance.

**Parameter:**

- **Isolated Distance**: This value defines how far the points that can be considered as isolated points are from the main points.

2. **Decimate Point Cloud**

The Decimate Point Cloud command reduces the number of points while still maintaining an accurate representation of the point cloud.

**Parameters:**

- **Point Distribution**: This value defines the distribution type of remaining points. The "Uniform" type reduces points according to a defined curvature weight that makes more uniform distribution of points on flat regions than the "Non-uniform" type.

- **Point Reduction**: Under the Uniform distribution type, there is a Curvature Weight parameter, which defines the degree of magnification for the distance between points for high curvature areas. The higher the curvature weight, the more points will be kept around edges.

  Meanwhile, there are three options to define the number of remaining points. Remaining Points defines how many points should be kept. Remaining Percentage defines how many percent of points will be remained. Min Distance defines the minimum distance required between points. If the distance between any two points is less than the given value, one of the points will be deleted.

  Under the Non-uniform distribution type, the number of remaining points is defined by a desired number or percentage.

3. **Denoise Point Cloud**
The Denoise Point Cloud command removes noises from point cloud with the preservation of important surface features like sharp edges and corners under the control of the *Smoothing Weight* parameter.

**Parameters:**

- **Smoothing Weight**: This value specifies smoothing constraints for sharp edges and corners. The higher this value, the less the sharp edges and corners will be affected.

- **Smoothing Iteration**: This value defines the number of iterations for smoothing.

4. **Point Cloud to Mesh**

The Point Cloud to Mesh command converts point clouds into triangle meshes.

**Parameter:**

- **Points from Terrain Contour**: Select this checkbox if points are generated from terrain contour lines by conventional survey.

5. **Remove Floating Parts**

The Remove Floating Parts command deletes small sub-objects according to the connection of meshes.

**Parameter:**

- **Min. Number of Triangles**: This value defines the minimum number of triangles required for a valid part. A part that contains fewer triangles than the defined value will be deleted.

6. **Decimate Triangles**

The Decimate Triangles command reduces the number of triangles while maintaining a good approximation to the original geometry.

**Parameters:**

- **Triangle Reduction**: There are two options to define the remaining number of triangles. *Remaining Percentage* defines how many percent of triangles will be remained. *Remaining Triangles* defines how many triangles should be kept.
7. Seam Gaps

The Seam Gaps command lets you fix overlapped regions and seam gaps between boundaries.

Parameters:

- **Check Overlapping**: If the checkbox is selected, the program will fix the overlapped regions before seaming gaps.
- **Overlap Tolerance**: Two surfaces are considered as overlapped, if the distance between the two surfaces is smaller than the given tolerance.
- **Gap Distance**: This value defines the maximum size of a gap for seaming. It is better to define the Gap Distance at least two times bigger than the Overlap Tolerance.

8. Fill Holes

The Fill Holes command fills all holes or some preferred holes whose area is smaller than the given value. This command has the capability to clear non-manifold edges.

Parameters:

- **Open Object**: If the checkbox is selected, the maximum boundary will be kept intact.
- **Averaged Hole Radius**: This value defines the maximum area that a hole will be filled.

9. Fill Fjords

The Fill Fjords command fills all "fjords" automatically. The "fjord" is defined as a long sharp hole, which may be a closed hole or an open hole around the boundary.

Parameter:
- **Vertex Angle**: This value defines the maximum vertex angle along a boundary. A new triangle will be added to cover the vertex if its angle is smaller than the given value.

### 10. Unify Normals

The Unify Normals command unifies the normals of a mesh so that they face in the same direction, inward or outward.

**Parameter:**

- **Fix Errors**: If the checkbox is selected, the program will detect and delete ill-connected triangles while unifying the normals.

### 11. Fix Errors

The Fix Errors option provides a group of commands for mesh repair.

- **Collapse Short Edges**: This command collapses triangles whose edge length is shorter than a given value.
- **Optimize Edge Connection**: This command optimizes a mesh by swapping an edge shared by two triangles.
- **Remove Multiple Edges**: This command removes triangles connected to multiple edges. An edge shared by more than two facets is considered as a multiple edge.
- **Remove Folded Triangles**: This command removes the folded triangles automatically. Two triangles are considered as folded, if their face angle is smaller than a given value. A face angle is the internal angle at which two adjacent faces meet.
- **Remove Intersected Triangles**: This command removes intersected triangles. Intersected triangles are triangles cutting each other. It can happen sometimes when a surface has intersections.
- **Remove Long Edges**: This command removes edges whose length is longer than a given value.

### 12. Remesh Smoothing

The Remesh Smoothing command removes noises from a mesh with the preservation of important surface features like sharp edges and corners under the control of the *Smoothing Weight* parameter.

**Parameters:**
13. **Reverse Surface**

The Reverse Surface command reverses the order of polygonal cells and/or vertex normals.

14. **Cross Section**

The Cross Section command allows you to create a cross-section curve from a mesh/point cloud, or split a mesh in two pieces.

**Steps:**

1. Click on the Cross Section command icon 📔 in the toolbar.
2. Select a mesh/point cloud in the drawing window to open up a control panel.
3. Adjust the parameter settings in the control panel.
4. Click on the [Apply] button to start the operation.

**Parameters:**
• **Operation**: The *Create Curve* option allows you to generate a cross-section curve from a mesh or point cloud. The *Clip Mesh* option allows you to split a mesh in two pieces.

• **Direction**: There are two ways to define the normal direction of the clipping plane. The *Align to Axis* option allows you to set the clipping plane to a specific position constrained to the X, Y, or Z axis. The *Align to Points* option allows you to select three or two points on the mesh to place the clipping plane. You may shift the clipping plane along a defined normal direction.

• **Adjust Direction**: This option allows you to adjust the normal direction of the clipping plane.

• **Simplify Curve**: This option allows you to reduce vertices from a cross-section curve. The smaller the value of *Tolerance*, the more vertices are kept on a curve. You may select the *Square Up* checkbox to refine all corners to 90 degree.

• **Curve Color**: The color square allows you to open a color palette to choose a displaying color for a cross-section curve shown in the drawing window.

15. **Intersection**

The Intersection command automatically finds the line of intersection between two planes.

**Steps:**

1. Click on the Intersection command icon in the toolbar.
2. Select a mesh/point cloud in the drawing window.
3. Define the first plane by placing three points on a mesh/point cloud surface.
4. Define the second plane by placing three points on another mesh/point could surface.
5. Press Enter or right click to accept the result. Press Esc to ignore the result.

16. **Polyline**

The Polyline command allows you to create a 3D polyline on a mesh/point cloud surface.

**Steps:**

1. Click on the Polyline command icon in the toolbar.
2. Select a mesh/point cloud in the drawing window.
3. Click on a mesh/point cloud surface to define vertices in a polyline. Each successive vertex defines a new node and segment in a polyline. Press Esc to remove the last vertex.
4. Press Enter or right click to exit.

17. **Spline**

The Spline command allows you to create a 3D spline on a mesh/point cloud surface.

**Steps:**

1. Click on the Spline command icon  in the toolbar.
2. Select a mesh/point cloud in the drawing window.
3. Click on a mesh/point cloud surface to define vertices in a spline. Each successive vertex defines a new node and segment in a spline. Press Esc to remove the last vertex.
4. Press Enter or right click to exit.

18. **Reset Points**

The Reset Points command clears the filter added to a point cloud entity.