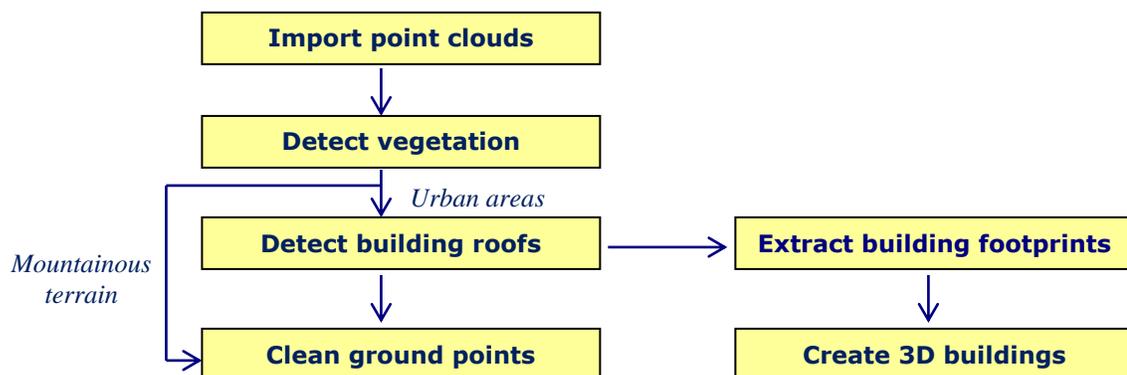


## Point Cloud Classification

### Introduction

VRMesh provides a powerful point cloud classification and feature extraction solution. It automatically classifies vegetation, building roofs, and ground points. 99% classification jobs can be automatically done in one click by setting a few simple parameters.

Unlike traditional point cloud classification workflows that have to extract ground points first before classifying vegetation and buildings, the workflow of point cloud classification in VRMesh is to detect vegetation first, then extract building roofs, and finally clean ground points. A basic processing workflow in VRMesh is shown below:



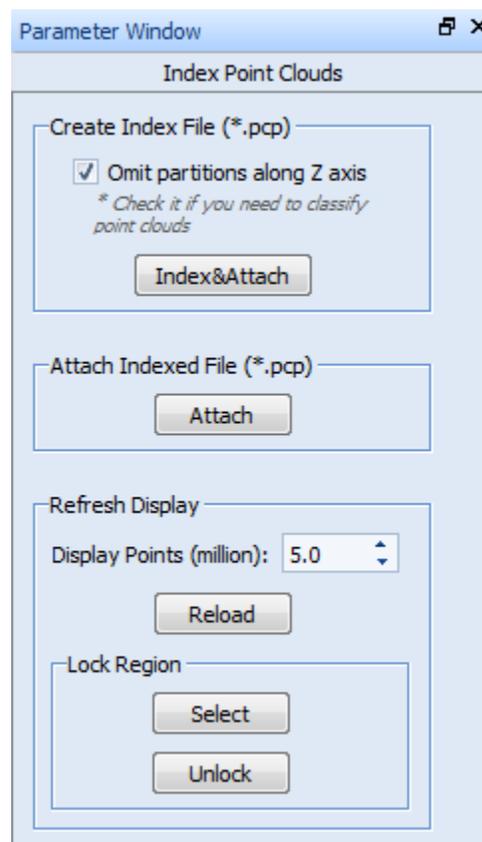
In addition, VRMesh supports batch processing of multiple files. It is recommended that using a sample file to figure out a customized workflow with tailored parameter settings. The batch mode will process all files in a specified folder one at a time using the same settings. Output results will be automatically saved to a subfolder at the same location as the original folder.

## Index/Attach Data

There are two commands that can be used to read a point cloud file (\*.txt, \*.asc, \*.xyz, \*.las, \*.pts, \*.dxf). The **Import** command imports all data into the window. The **Index/Attach** command retrieves inquiring points into the window. Depending on your computer settings, you may choose to attach a point cloud file into the window if it contains more than 3 million points. The following instruction will guide you through the process.



Click the **Index/Attach** command in the File menu, or click the icon in the Quick Access Toolbar. The Index-Attach working panel appears in the Parameter Window on the right side of the screen, as shown below:



### Step 1: Index/Attach

Press the [**Index&Attach**] button. A dialog box is opened to allow you to choose a name, directory, and file type for the file. The program will create an index file with a \*.PCP extension saved to the same folder as the original file, and then attach the indexed file to the current project. The attached file will be marked with an asterisk in the Object Tree Window, e.g., \*Object.

You only need to create an indexed PCP file once for the original file, which means that you can directly press the [**Attach**] button if the indexed file has already existed.

## Step 2: Refresh Display

Press the [**Reload**] button to display the point cloud data with the preferred size. Or, press

the [**Select**] button. The cursor changes to a pencil [  ], which allows you to draw a contour and right click the [Keep Inside Region] or [Keep Outside Region] option to cut a region of interest from your model. The [**Unlock**] button allows you to merge back the selected region to your model.

## Bare Earth Extraction

The creation of a bare-earth surface involves classifying vegetation, buildings, and ground points, and then generating a grid surface or a triangulated surface to represent the topography. The following instruction will guide you through the bare-earth generation process.

### Step 1: Detect Ground Points

Open up the **Survey Wizard** by pressing the icon  in the Classification menu. Select the following three commands and click on one of the right side buttons  to adjust the value of parameters for each selected command:

#### 1. Detect Vegetation

This is the first fundamental step in VRMesh workflow for point cloud classification. The program calculates the roughness on point clouds to identify vegetation. Normally, the value of roughness is set to 0.25 for an urban area, and 0.6~0.8 for a mountainous terrain. The value of minimum points required for a valid region is normally 5~50.

#### 2. Detect Building Roofs

This step automatically detects building roofs in point clouds. The maximum value of a roof slope is normally set to 50~85. You also need to define the minimum height of a roof and the minimum / maximum number of points required in a roof.

You may ignore this step if your data is from a mountainous region without buildings.

#### 3. Clean Ground Points

This step automatically deletes floating parts to get well classified ground points. There are two clean types. The **Flat Region** type is preferred to a flat urban region. The **Steep Region** type is preferred to a steep mountainous region.

Press the [Batch] button to execute the selected commands. After ground points have been identified. You may use the **Show Points by Classification** command  in the Analyze menu to view each class of points. If ground points are not enough, you may use the **Add Point to Ground** command  in the Classification menu to assign more points to the ground class.

## Step 2: Create a Bare-Earth Surface

There are two ways to generate a bare-earth surface in VRMesh. You may directly create a grid surface to represent the bare earth, which is simple and fast. Or, you may choose to triangulate point clouds into meshes, which is more accurate in representing the bare-earth topography.

For creating a grid surface, click the **Create Grid Surface** command  in the Feature Extraction menu to create a grid surface from visible points on the screen. You may use the

**Show Points by Classification** command  in the Analyze menu to control which class of points are shown on the screen. Please note that a grid surface generated using this command may not accurately represent the topography of a steep mountain. It's better to create a triangle-meshed surface for steep terrain.

For creating a triangulated bare-earth surface, use the **Split Classified Points** command



in the Feature Extraction menu to split the ground points. Then, use the **Denoise**

**Point Cloud** command  in the Point Clouds menu to smooth the point cloud. And

finally, use the **Point Cloud to Mesh** command  in the Point Clouds menu to convert the point cloud into a triangle mesh. If the generated mesh has a large number of triangles,

you may use the **Decimation** command  in the Mesh Editing menu to reduce the number of triangles into an acceptable amount depending on your computer system.

## Building Footprint Extraction

VRMesh provides an automatic solution for extracting building footprints from point clouds and generating 3D building models. It doesn't require a good bare earth before identifying building roofs in VRMesh workflow. Therefore, VRMesh can successfully extract building footprints from areas where there is very few ground points available. The following instruction will guide you through the footprint extraction process.

### Step 1: Detect Vegetation

Click the **Detect Vegetation** command  in the Classification menu. Specify the parameter settings. Normally, the value of roughness is set to 0.25 for an urban area, and 0.6~0.8 for a mountainous terrain. The value of minimum points required for a valid region is normally 5~50.

### Step 2: Detect Building Roofs

Click the **Detect Building Roofs** command  in the Classification menu. The maximum value of roof slope is normally set to 50~85. You also need to define the minimum height of a roof and the minimum/maximum number of points required in a roof.

### Step 3 (Optional): Clean Ground Points

Click the **Clean Ground Points** command  in the Classification menu. The command automatically deletes floating parts to get well classified ground points.

### Step 4: Extract Building Footprints

Click the **Extract Building Footprints** command  in the Feature Extraction menu once you get the classified building roofs. Specify the number of points in each grid cell, normally 8 ~25, which is used by the program to create a grid surface for footprint extraction. Define a minimum area for a valid roof. Select the *Square Up* option if preferred, the program will refine all corners to 90 degree.

## Step 5: Create 3D Buildings

Select the **Create 3D Building** checkbox in parameter settings of the **Extract Building Footprints** command, or click the **Create Building from Footprint** command in the Feature Extraction menu. You can define a roof shape as an *Approximate* type, or a *Flat* type. The approximate type means the roof shape is close to the actual shape. The flat type means all roof shapes are horizontal. You may also specify the maximum number of triangles to represent a roof.

## Powerline Processing

Learn the general procedure for processing powerline data in VRMesh:

- Classify points into vegetation, buildings, and ground.
- Reclassify high points hit on wires by absolute elevation or the height from ground.
- Manually place catenary strings between towers.
- Detect powerlines along catenary strings.
- Analyze clearance using a tunnel object.

### Step 1: Classify Point Clouds

Open up the **Survey Wizard** by pressing the icon  in the Classification menu. Select commands to classify vegetation, building roofs, and ground points.

### Step 2 (Optional): Reclassify High Points

Those high points hit on wires or towers may have been classified into the building class in step 1. If it is clear enough for placing a catenary curve along those points between towers, you can ignore this step and go directly to step 3. If it is necessary to reclassify high points,

you may use the **Absolute Elevation** command  or the **Height from Ground** command  in the Classification menu to get more accurate results.

### Step 3: Place Catenary Strings

Click the **Place Catenary String** command  in the Feature Extraction menu to manually place a catenary curve along points. You may use the **Adjust Catenary String** command  in the Feature Extraction menu to further adjust the curve.

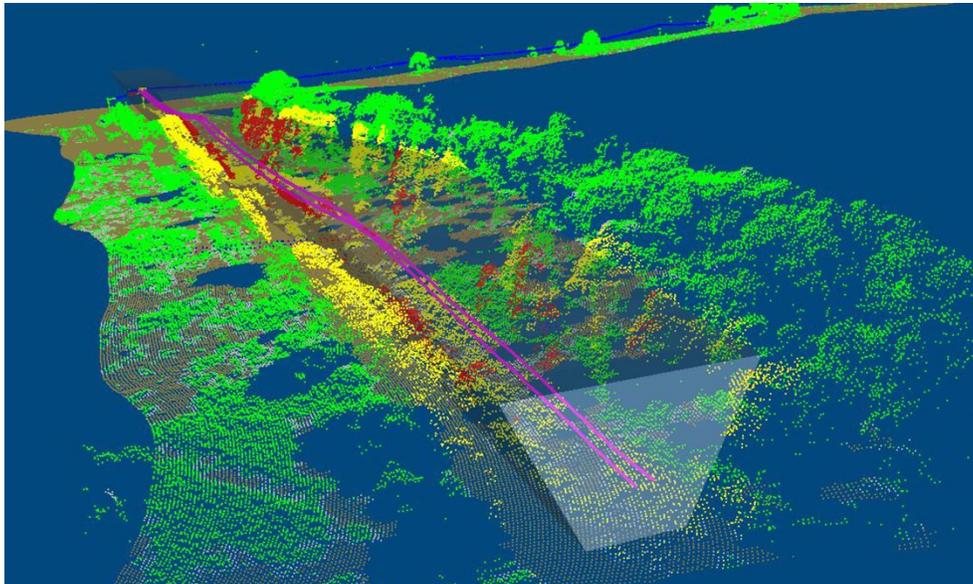
### Step 4: Detect Powerlines

The **Detect Powerlines** command  in the Feature Extraction menu will automatically classify points along a selected catenary curve in a given class. You need to specify a source

class and a target class. Define a maximum offset distance from catenary curve to search for points, which is normally set to a little bit more than half of the width of the powerline. Define a search gap between consecutive hits on a wire. It is recommended that you first try a relative short search gap for the detection.

### Step 5: Analyze Clearance

The **Classify by Tunnel** command  in the Classification menu allows you to classify points based on the shape of a tunnel, which can be used for conducting a clearance analysis between powerlines and surrounding vegetation. The program will use a tunnel section template to classify points which are inside the tunnel, outside the tunnel or close to the tunnel section. You need to select a catenary curve to represent the centroid of a tunnel before using this command. If the *Keep Tunnel* option is selected, the tunnel object will be kept. The transparency of the tunnel object can be adjusted by clicking the **Render** button  on the right side of the Home menu.



**End of Tutorial**