

# **CHCNAV CoProcess SW**

User Manual Version 2.6



Mobile Mapping | DECEMBER 2023



### **Table of Content**

1.	Soft	ware Inti	roduction	1
		1.1.1.	Browse Platform	1
		1.1.2.	Terrains Module	2
	1.2.	Hard	dware Requirement	3
	1.3.	Soft	ware Install	3
	1.4.	Soft	ware Uninstall	4
	1.5.	Soft	ware Registration	4
		1.5.1.	Apply for Trial	4
		1.5.2.	Dongle and Registration Code	5
		1.5.3.	Redemption Code	6
2.	Soft	ware Use	erGuide	7
	2.1.	Terrain	s User Guide	7
		2.1.1.	DEM Producing	7
		2.1.2.	Contour&Elevation points Generation	24
		2.1.3.	Buildings outline Extraction	30
	2.2.	Terra P	arameter description	47
		2.2.1.	Isolated point (noise) filtering	47
		2.2.2.	Low point filtering	49
		2.2.3.	Ground point filtering(地面点)	51
		2.2.4.	Key Points Parameter Description	66
		2.2.5.	Contours Generation	69
		2.2.6.	Contours Smooth	71
		2.2.7.	Elevation points Generation	73
		2.2.8.	Contours Simplify	75
		2.2.9.	Generate elevation points	77
		2.2.10.	Building Filtering	80
		2.2.11.	Vegetation Filtering	87



		2.2.12.	Extracting building Outline	89
		2.2.13.	Buildings Editing	96
	2.3.	Road	ds User Guide	98
		2.3.1.	Introduction	98
		2.3.2.	Step-by-step	100
		2.3.3.	Import Point Cloud	100
		2.3.4.	Obtain pile by pile	101
		2.3.5.	Pile Add(Not Necessary)	106
		2.3.6.	Generate Section	107
		2.3.7.	Vertical section editing	109
		2.3.8.	Edit Section	110
		2.3.9.	Vector Editing	111
		2.3.10.	Exporting Survey Results	111
		2.3.11.	Generate Section View	111
	2.4.	Volume	e User Guide	113
		2.4.1.	Introduction	113
		2.4.2.	Sample data	113
		2.4.3.	Work Flow	114
		2.4.4.	Preprocessing	115
		2.4.5.	Projected Volume Calculation	118
		2.4.6.	Volume Comparison	122
		2.4.7.	Change Analysis	124
		2.4.8.	Bench Lines	128
3.	Soft	ware Use	erManual	130
	3.1.	File8	&Project	130
		3.1.1.	Open Resources	130
		3.1.2.	Save Resources	130
		3.1.3.	Open Point Cloud	130



	3.1.4.	Open Model	35
	3.1.5.	Export Point Cloud	36
	3.1.6.	Convert Terrain Model13	37
	3.1.7.	Convert Codata	38
	3.1.8.	System Settings	39
	3.1.9.	Theme	43
3.2.	Hom	ne14	44
	3.2.1.	View Direction	44
	3.2.2.	View Scale14	44
	3.2.3.	View Control	44
	3.2.4.	Scene Roam	47
	3.2.5.	View Management	49
	3.2.6.	Colorization	50
	3.2.7.	Tools	56
	3.2.8.	Clipping	58
	3.2.9.	Measurement	53
	3.2.10.	Slice	76
	3.2.11.	Windows	78
3.3.	Terrain	s18	32
	3.3.1.	Import Las	32
	3.3.2.	Work Flow	33
	3.3.3.	Block Management	)2
	3.3.4.	Category Editing20	)3
	3.3.5.	Building Extraction	13
	3.3.6.	Contour Edit	21
	3.3.7.	Text Annoation	26
	3.3.8.	DEM	29
2.4	D	J	



	3.4.1.	Project Manager
	3.4.2.	Road Parse
	3.4.3.	Road Survey
	3.4.4.	Other Tools
3.5.	Volume	264
	3.5.1.	Project Manager
	3.5.2.	Point Cloud Processing
	3.5.3.	Triangulation
	3.5.4.	Single Volume Calculation
	3.5.5.	Multi Volume Calculation
	3.5.6.	Vector Achievements
	3.5.7.	Vector Line Processing
	3.5.8.	Reference Plane
	3.5.9.	TIN Editing
	3.5.10.	Crop
3.6.	Help	
	3.6.1.	Help291
	3.6.2.	Version
	3.6.3.	Update
	3.6.4.	Language



### 1. Software Introduction

CoProcess software is based on the independent research and development of the underlying rendering and vector interaction engine, adopts the "1+N" new development and application model of "one platform, multiple industry modules", and provides SDK to support customers in secondary development of application modules. The system provides a variety of platform functions such as massive multi-source data management, 3D rendering, vector drawing and symbolization, measurement and quality inspection, and for the needs of different industries, targeted development of supporting industry application modules:

- Filter module for 3D lidar fields;
- Terrains module for traditional surveying and mapping fields;



#### 1.1.1. Browse Platform

### 1.1.1.1. Massive point cloud rendering

The software uses a custom format (\*.codata) to quickly visualize massive point cloud data, and can quickly build \*.las, \*.txt, \*.csv, \*.pts and \*.xyz into codata through LOD technology. Covers conventional renderings such as height, intensity, RGB, classification, single, time, returns, return number, etc. It provides multi-hue rainbow, blend and mix rendering, and EDL effects improve rendering detail contrast

#### 1.1.1.2. Scene Roam

Realize the scene roaming of point clouds, vectors and images, perform viewpoint roaming according to the viewing angle position, and perform trajectory roaming by setting the



browsing path.

### 1.1.1.3. Clipping

It provides rectangle and polygon selection of point cloud, and provides the functions of inner clipping, outer clipping, clearing and saving of point cloud.

#### 1.1.1.4. Measurement

It provides single point, multi-point, distance, area, density and angle measurement. In addition to basic measurement, the software also provides elevation inspection, density quality inspection and profile analysis functions.

- Provide point cloud density quality inspection, and can generate point cloud density reports.
- You can draw multiple sections, move the position of the section, adjust the width of the section, and open multiple sections at the same time for comparative analysis.

### 1.1.2. Terrains Module

### 1.1.2.1. Automatic Processing

Automatic processing to output DEM results that meet the accuracy requirements.

#### 1.1.2.2. Intelligent Block

According to the point cloud density and range, quickly and intelligently block and generate a block project.

#### 1.1.2.3. Automatic Filtering

Several sets of common parameters are provided for different terrains, which can quickly and accurately extract ground points under complex terrains. In addition, the software also provides filtering processing of outliers, low points, and key points.

#### 1.1.2.4. Manual Editing

After filtering, the point cloud can quickly construct a TIN, which is convenient for viewing the terrain. It provides quick classification functions such as manual editing of point clouds, simultaneous editing of point clouds and TINs, and section editing, as well as functions such as category saving, modification, and export, which improve the user experience and ensure that the classification result point cloud that meets the job requirements can be output.

#### 1.1.2.5. **DEM Linkage**

The 2D and 3D view synchronization update method can quickly realize the update of DEM . The 2D view loads DEM with large data area, which is convenient for operators to view the overall DEM and quickly locate the area range that needs to be modified in the edge part. Select the specified area in the 2D view to display it in the 3D view synchronously. The 3D view edits the DEM, and at the same time, the editing results are updated and saved to the overall DEM data in real time.

### 1.1.2.6. **DEM Editing**

DEM editing provides a variety of editing methods such as elevation leveling, elevation



smoothing, elevation deletion, patching invalid values, elevation patching, removing spikes, and adaptive smoothing to quickly edit DEM results.

### 1.1.2.7. Contour lines and elevation points

Generate contour lines based on point cloud in the block and DEM and elevation points, provide three ways to smooth the generated contours, and also provide contour simplification function.

Elevation points can be extracted directly based on ground points according to regular grid.

### 1.2. Hardware Requirement

Operating system	Microsoft Windows 7 (64bit)
Operating system	Microsoft Windows 10 (64bit)
CPU	Minimum: Intel Core i5,2+ GHz
CPU	Recommend: Intel Core i7, 3.5+ GHz
Mamaru	Minimum: 8GB
Memory	Recommend: 16GB+
CDII	Minimum: 4GB
GPU	Recommend: 8GB
Display resolution	1920*1080 above
graphics card	NVIDIA GeForce 1050 Ti +

### 1.3. Software Install

Note: Turn off 3<sup>rd</sup> party antivirus software, in case the install process being interrupted. And the install path needs complete English letters.

Open software package then click "I agree" and continue install:





- Waiting software install completely:
- When software installed successfully, click finish to close current interface:



### 1.4. Software Uninstall

First way, click "Start" to open windows start menu. Find CoProcess2 folder then click "Uninstall CoProcess2" and click yes to start uninstall.



Second way, go to "Settings - Apps", and find CoProcess2.2 then right click to uninstall.

## 1.5. Software Registration

The software registration supports four methods: Apply for trial, dongle, registration code and redemption code.

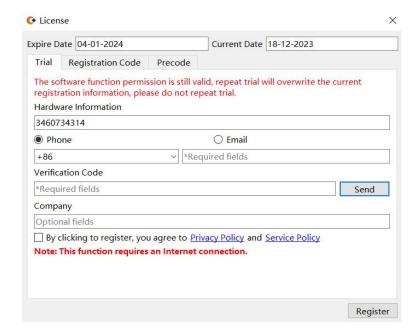
### 1.5.1. Apply for Trial

After the CoProcess software is installed, click the desktop icon of the CoProcess software. After CoProcess is opened, click Help -> Licenses, then the license dialog box will pop up.

On the "Apply for Trial" page, enter you mobile phone number, email address and company name(optional), click the "Send verification" button and enter the verification code you receive.



Click the "Register" button. After the software restarts automatically, you can use the charge module of the CoProcess software.



Apply for trial

#### Note:

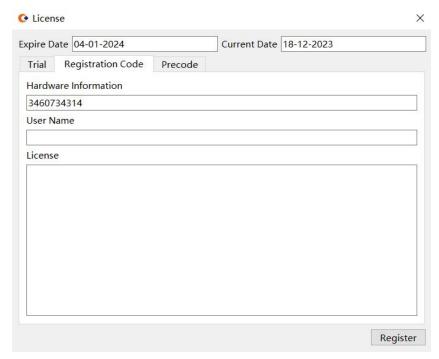
The Internet connection is required when registering with a verification code.

### 1.5.2. Dongle and Registration Code

After the CoProcess software is installed, click the desktop icon of the CoProcess software. After CoProcess is opened, click Help -> Licenses, then the license dialog box will pop up and switch to the second page "Registration Code".

Send the hardware ID to us, and you can get the authorization code. Enter the authorization code and click the "Register" button, then you can use the charge module of the CoProcess software.





Registration code

#### Note:

When the registration code is 3649838548, it means that the software does not detect the NIC information.

When the user name is SUPERDOG, it means that there has been dongle in the computer, please do not use the hardware ID to register! If the plug-in cannot be loaded at this time, please contact the customer service of CHCNAV for help.

### 1.5.3. Redemption Code

After the CoProcess software is installed, click the desktop icon of the CoProcess software. After CoProcess is opened, click Help -> Licenses, then the license dialog box will pop up and switch to the third page "Redemption Code".

On this page, enter you mobile phone number and company name(optional) and the redemption code you obtain.

Click the "Register" button. After the software restarts automatically, you can use the charge module of the CoProcess software.





Redemption code

#### Note:

Each redemption code can only be used once on one computer and cannot be redeemed repeatedly.

The current computer cannot use other exchange codes during the validity period of one redemption code.

The Internet connection is required when registering with the redemption code.

### 2. Software UserGuide

### 2.1. Terrains User Guide

### 2.1.1. DEM Producing

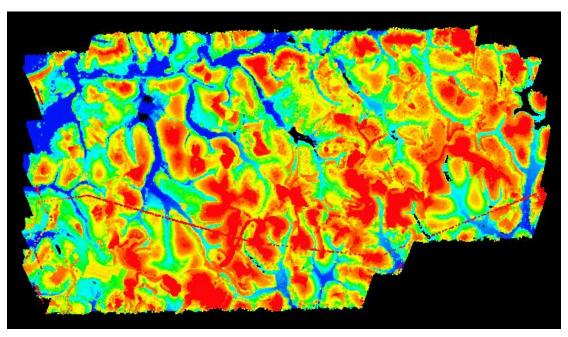
This document is intended to provide technical guidance for the production process of traditional topographic DEM for users using CoProcess. The entire process of DEM production is briefly presented using a single airborne lidar point cloud data of approximately 10G.

This document does not involve detailed explanation of specific functions. If necessary, please refer to CoProcess User Manual.

#### 2.1.1.1. Sample data

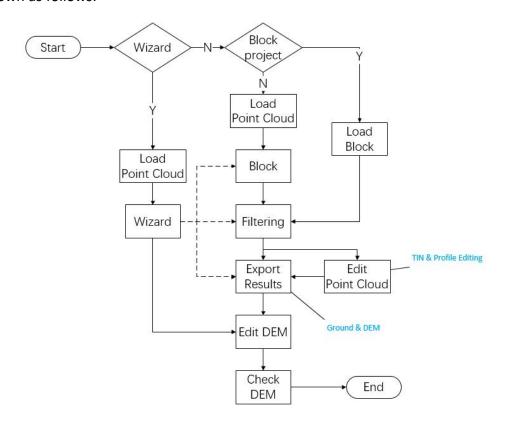
The sample data used in this demonstration is the original LAS point cloud collected by a single uav in a hilly area, with a total size of 11.5GB.





2.1.1.2. Work Flow

The production process of traditional topographic DEM using CoProcess is shown as follows:



Coprocess supports two DEM production methods: wizard and step-by-step.

Among them, wizard can greatly reduce the operation steps, guide new users to operate well and quickly generate DEM results, which is suitable for the scene with



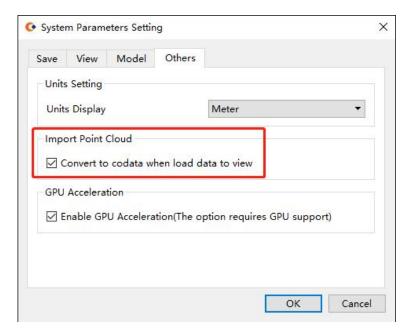
low precision and urgent demand for results.

However, due to the lack of fine-grained editing of the point cloud category, the resulting results may be less precise, in which case the step-by-step approach can be used.

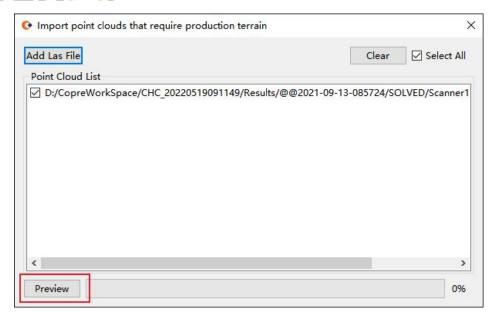
### 2.1.1.3. Wizard

The wizard can be divided into the following three steps: 1) Import point cloud data; 2) Click "Wizard" button to set processing parameters; 3) Perform wizard operations. The basic operations of these processes are described in detail below:

Step 1: Import the point cloud data to be processed and click "Wizard" button.



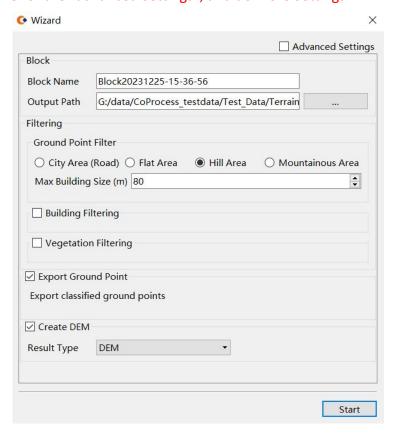




Step 2: Set block parameters, filtering parameters and the result type, click "Next" on the Wizard interface;

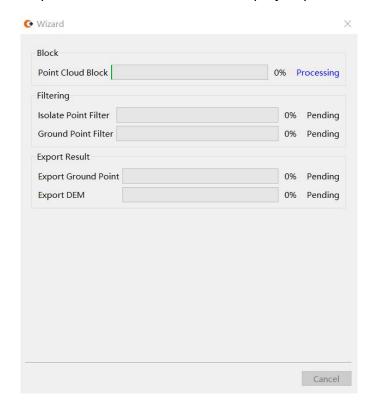
Filtering needs to select the corresponding processing parameters according to the actual terrain, select "Hill" here.

**Note**: if you need to adjust the concrete block and filter parameters, check the top right corner of the "advanced Settings", and do more Settings.





Step 3: Click next and click "Start" button. The software will automatically process the ground points and DEM under the block project path



After the processing is completed, click "Finish" to exit Wizard.

### 2.1.1.4. **Step-by-step**

The step-by-step can be divided into the following 6 steps: 1) Block; 2) Filtering;
3) Manual classification; 4) DEM generation; 5) DEM editor; 6) DEM quality inspection. The basic operations of this process are described in detail below.

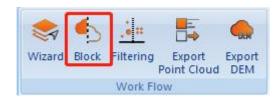
#### 2.1.1.4.1. Block

The amount of airborne point cloud data in a single sortry is up to 11.5G, and it is difficult to import all the data into the software for processing at one time.

Therefore, we can use the Block function to divide the original point cloud into some small blocks for convenient data loading and editing.

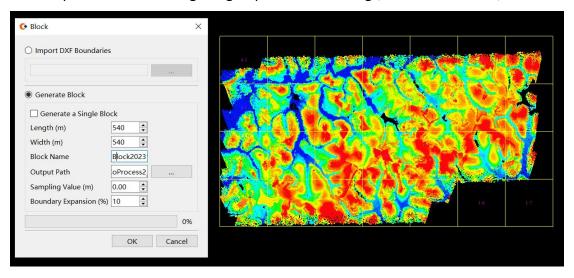
Step 1: Add the point cloud file to be processed in the workspace, and click "Block" button in the terrain module workflow to pop up the block window;





Step 2: Set the grid size, block name and save path in the window.

Step 3: After confirming the grid parameter Settings, click "OK" button;

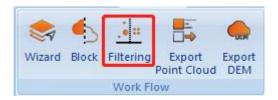


Step 4: After the block conversion is completed, a prompt will pop up whether to load the block point cloud. Click "Yes" here for better subsequent operations.

### 2.1.1.4.2. Filtering

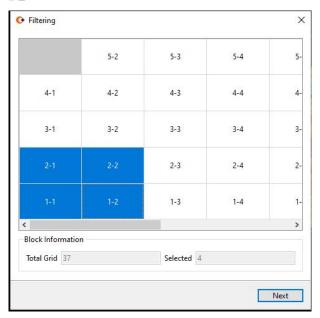
After the block operation is completed, the block point cloud is filtered to get ground points.

Step 1: Click the filtering button.



Step 2: Select the block to be filtered. Here we choose the four blocks in the lower left corner. If no selection is made, all blocks are selected by default.





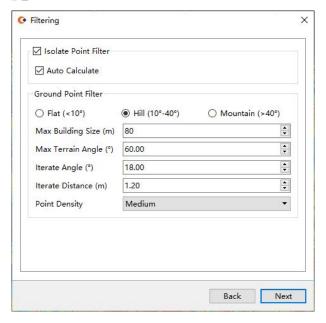
Step 3: Click "Next" to set the ground point filtering parameters. Considering the actual terrain of the demonstration data, "Hill" mode is selected for ground point filtering here. Users can adjust according to the actual terrain;

If there are special requirements for some filtering parameters in the actual production, the functions of "isolated point filter" and "ground point filter" in filtering processing can be used for parameter tuning, that is, adding small sections of data for processing, so as to find suitable filtering parameters.

For isolated point filter, after selecting Auto Calculate, the software will automatically calculate the filtering parameters.

Step 4: After filtering parameters are set, click Next and then click Start generation to wait for filtering to complete.





2.1.1.4.3. Manual Classification

After the completion of filtering, the results need to be checked and modified due to the existence of ground point misclassification or missing classification. To ensure smooth running, you are advised to load only one piece of point cloud data at a time.

Step 1: Load block data.

Refer to the preview data, activate the "Block Management -> Select Block " function, and select the grid to be loaded in the view to load and display.



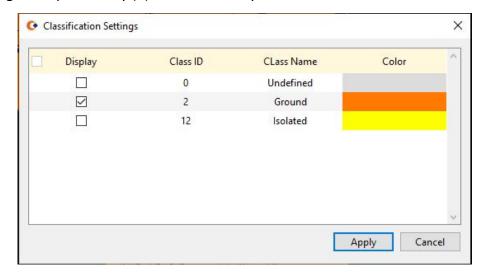
Step 2: Adjust the color mode.

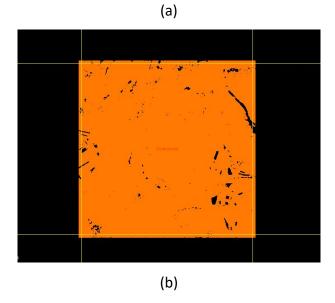
After grid points cloud data loaded, in order to facilitate to category editing of point cloud, use the shortcut toolbar to switch the color mode to category color and also set the categories to display.





Click color Settings to unshow all categories except ground points (a) and then show ground points only (b) to check for any errors in classification.





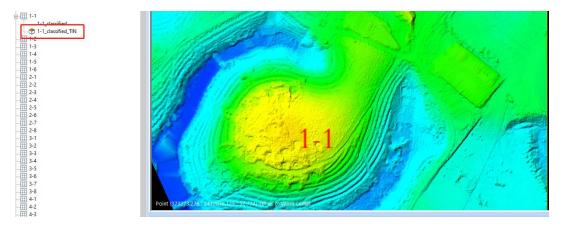
Step 3: Construct TIN to assist in viewing ground points.

After the ground points are loaded, it is difficult to detect misclassification or missing classification just by looking at the point cloud. Therefore, we can display the ground point information in the form of TIN through the TIN function.

Point cloud and TIN synchronously edit operation steps.

- 1.Load the partitioned point cloud file after classification and display the partitioned points to build TIN.
  - 2. Click "TIN" inCategory Editing and wait for TIN build to complete.





When doing the profile editing, we can look at TIN to determine whether the modification of the ground point has achieved the desired effect.

Step 4: Edit the point cloud category using Profile Editing.

When setting up the source and target classes, you can refer to the table below.

Condition	Source	Target
Misclassification	Ground	Undefined
Missing classification	Undefined	Ground

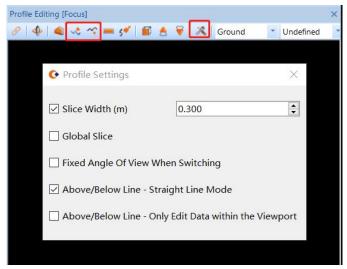
Steps for Profile Editing

1.Click "Profile Editing" button to open the section view and draw a section parallel to the block boundary of the main view so that "cut" the point cloud data of the main view into the section view;

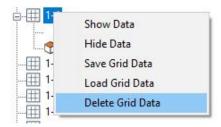


- 2.Combined with the main view and section view, carefully check whether there are wrong points or missing points point clouds, and use "upline " or" downline" to draw lines and edit:
- 3.After each profile is checked, use move up or move down to change the position of the profile shown in the profile view, and then edit the next profile;





4. After editing a single grid dot cloud, you can right-click the currently edited block data on the tree node and click "Delete Grid Data". The grid data will uninstall and save the currently edited point cloud data.



5. Repeat the process until the ground points of all blocks have been corrected.

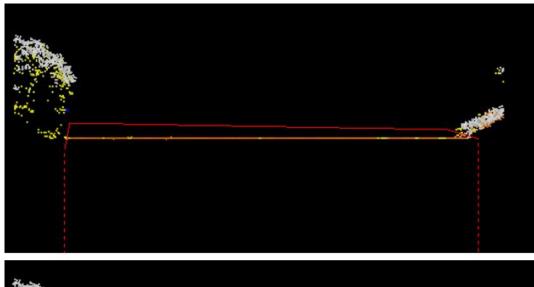
The following section editing steps will be introduced with specific examples.

### **Example 1: Profile editing operation:**

As shown, for a body of water, use "downline" in profile editing to correct the classification. Operation method:

- 1. Click on two points in the wrong section to draw a section;
- 2.Activate the downline tool, set the original category to ground and the target category to undefined;
  - 3.Draw a multi-segment line whose area below the line is full of misalignments;
- 4. Double click to end the drawing, you can complete the right and wrong classification points for correction;
  - 5. Repeat the process until all edits of the current block are complete.







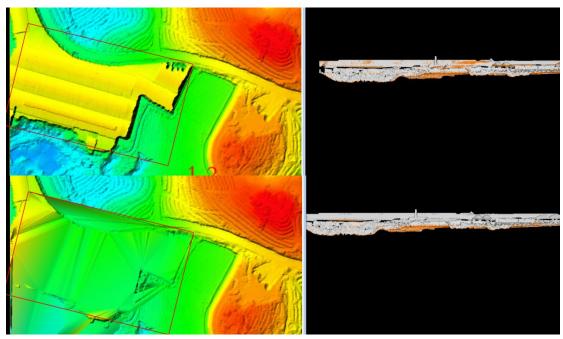
**Example 2: Point cloud and TIN synchronous editing:** 

As shown in the figure, the TIN display in the main view also changed after we corrected the misclassification of the roof as ground points in the section view by using the "upline" tool.

### Operation method:

- 1.Load block data with ground point information and construct TIN;
- 2. Open profile editing to modify misclassified ground points into undefined points in an area;
  - 3.Look at the TIN.





2.1.1.4.4. DEM Generation

Through the previous editing operation, we can get the ground point cloud after fine classification. Here, we will introduce the process of generating DEM by using these point cloud data.

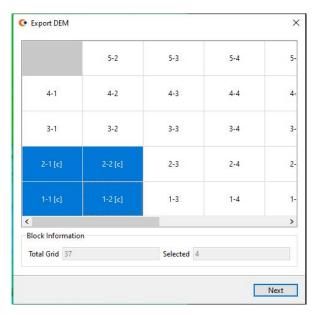
Step 1: Open the DEM generation tool. Click "Work Flow -> Export DEM" to open the DEM generation tool. The pop-up page is similar to filtering, and you can select blocks to be processed on this page.

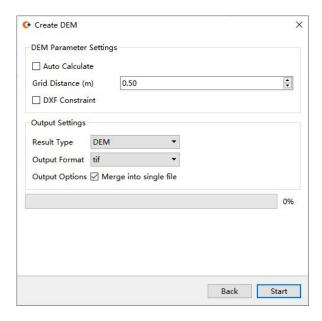
Step 2: After selecting blocks, click Next to switch to the page of DEM generation and set parameters of DEM generation;

Step 3: Set the output format and output type, and click Start button to start DEM generation. Here we check the "Merge into single file" checkbox;

**Note**: Auto Calculate is checked here. If you want to customize DEM parameters, deselect and set them.







Step 4: Click the "Start" button to start processing. After the output window prompt is completed, DEM will be successfully generated.

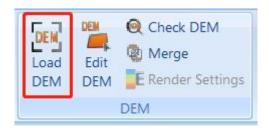
### 2.1.1.4.5. DEM Editor

To produce compliant products, manual inspection and careful editing of DEM are often required. Here we will introduce the DEM editing process.

CoProcess supports loading large DEM data by DEM Linkage for DEM 2-3D joint editing. The operation process is as follows:

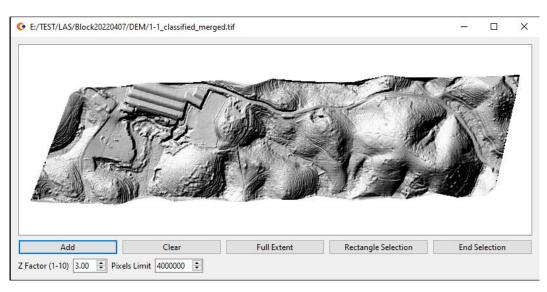


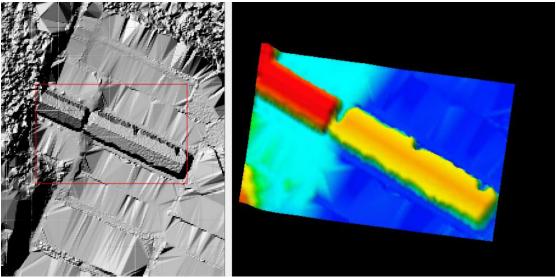
Step 1: Open the DEM Linkage. Click "DEM - > Load DEM " to open;



Step 2: Add DEM data to edit. Click "Add" in the lower left corner of the DEM Linkage dialog box;

Select small DEM blocks in the box for editing. Zoom in and out to view the DEM. When unqualified places are found, start Rectangular Selection, and left-click two points in the position to edit to draw a rectangular box. The DEM inside will be displayed in the main view in 3D form.





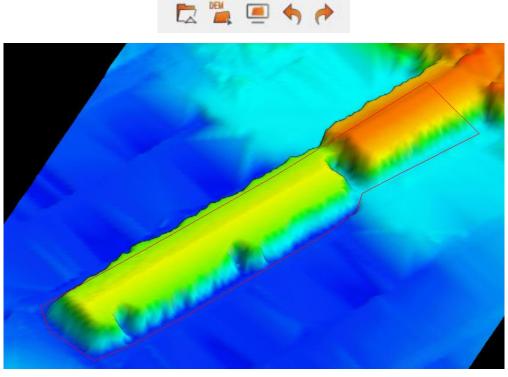


Step 4: DEM selection and editing.

We will combine the two cases to illustrate here, and the operation methods of other editing commands can be referred to these two cases.

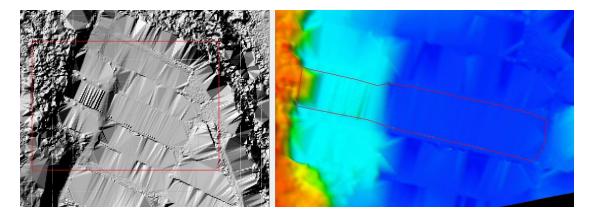
Case 1: As shown in the figure, DEM error caused by building splitting into ground points appears. In this case, we use the "Adaptive Smoothing" function in DEM editing to correct the DEM errors within the scope as follows:

- 1.Click Edit DEM function;
- 2.Click model selection in the processing tools window on the right;
- 3.Drawing in the main view will modify the polygon contained in the region, right-click to end the drawing;
  - 4. Click "Adaptive Smoothing" in DEM edit;
  - 5.Set the Elevation Lift to 0, check the Interpolation edge, and click OK.

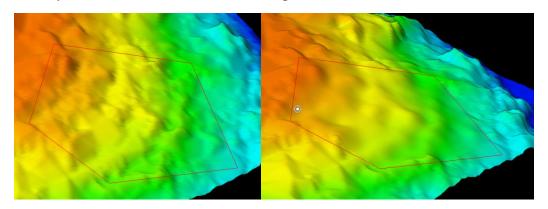


As shown, edits from 3d entities are synchronized to the 2D view in real time.





Case 2: Aiming at the uneven ground caused by some noise points, DEM can be leveled by means of the Elevation Smoothing tool.



The operation method is the same as Adaptive Smoothing, and the editing operation is as follows:

- 1.Using the model selection to delineate the boundaries of the areas to be smoothed on the DEM.
  - 2.Click Elevation Smoothing in Processing Tool;
  - 3.Click OK.

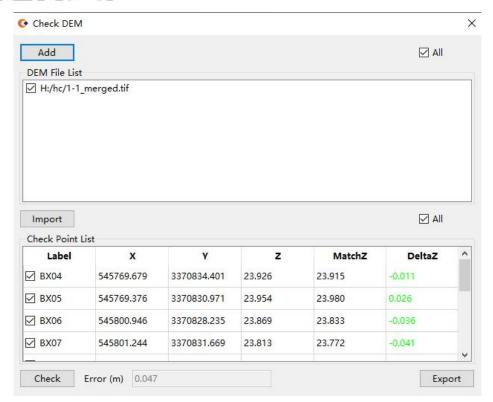
Note: DEM editing results will be saved in real time.

### 2.1.1.4.6. DEM Inspection

After DEM editing is completed, you can use DEM elevation checking tool to check the quality of DEM editing results.

CoProcess provides a way to evaluate DEM quality using elevation checkpoints: Calculate the point median error of elevation points so as to evaluate the quality of DEM.





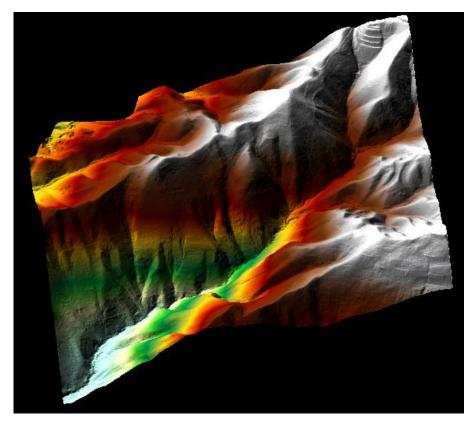
2.1.1.5. Results

The results of DEM inspection are the results that meet the requirements.

# 2.1.2. Contour & Elevation points Generation

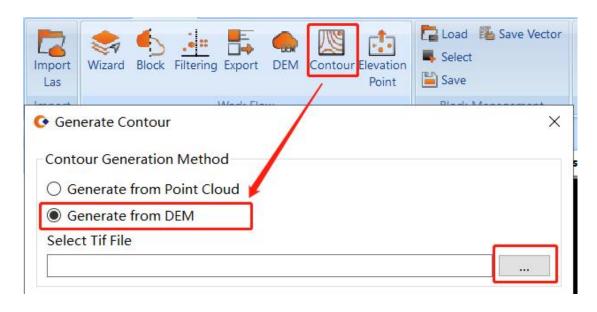
The CoProcess software supports the generation of contours and elevation points from ground points, as well as the direct generation from the existing DEM. Here we use an existing mountain DEM to generate contours.





A mountain scene

Step 1: After the "Block" is loaded, click the "Generate Contour" button, and choose the second option "Generate from DEM" in the "Contour Generation Method" bar. Then choose an already generated or processed DEM.



Choose a generation method



Step 2: Click the "Next" button, then set relevant parameters of "Contour Generation" and "Contour Smooth". The parameter of "Contour Interval" is set to the commonly used 2m, and the "Moving Average" and "Simplify Threshold" of the contour results can be set according to the actual requirements. Here, we choose the default 5 and 0.1.

Contour Interval (m)	2.00			-
Length Filter (m)	10.00			-
Simplify Threshold (m)	0.10			-
	✓ Filter Circle	☐ Filter Short Line	✓ Simplify	
Contour Smooth				
<ul><li>Moving Average</li></ul>		Neighbor points	5	*
○ Tension Spline Fittir	ng			
Spline Fitting				
O Not Fit				

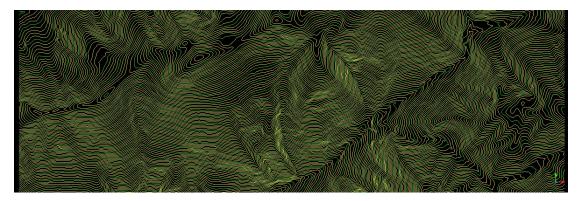
Back	Start

Set contour generation parameters

Step 3: After the above parameters are set, click the "Start" button, and the software will automatically generate the contours of the entire DEM range according to the parameters set.

After the generation of the contours is completed, they will be automatically displayed on the view. The yellow contours are the intermediate curves, and the green contours are the index curves.

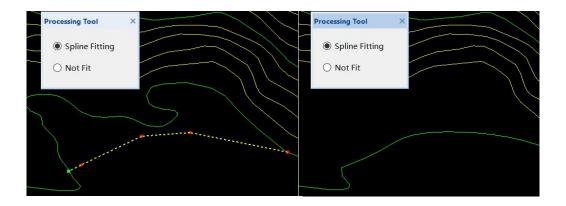




Set contour generation parameters

Step 4: Contour editing: Display the DEM and contours together. If we find there is something wrong with the generated contours, we can use the processing tool to repair them.

Press the shortcut key K or click the "Revise" button, then click a few points on the wrong contours. After double clicking the "OK" button, the newly generated contours will replace the original wrong contours. If we choose the "Spline Fitting" option, the repaired contours will be smoother.

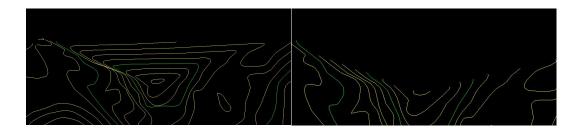


#### Contour revision

For the wrong contours at the boundaries of the DEM due to vertical surfaces, we can use the break function to delete them. The operation is very simple. Just activate the break tool and click two points on the two segments of the wrong contour.

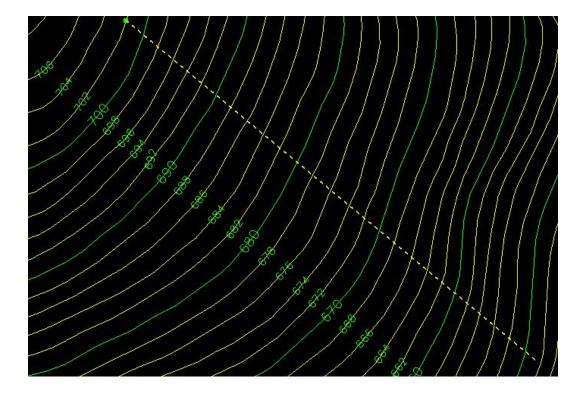


Or we can use a single point to break. What we need to do is clicking at the beginning and end of the wrong contour and then using the line deletion tool to delete the middle part.



#### Contour break/delete

Step 5: Draw contour notes: After a series of editing operations, there is no problem with the contours, and the next step we need to do is displaying the altitude notes of the contours on the contours. We can click the "Mannual Annotation" tool to draw a straight line on the contour, and then an altitude note will be marked where the drawn line intersects the contour line, as shown in the following figure:

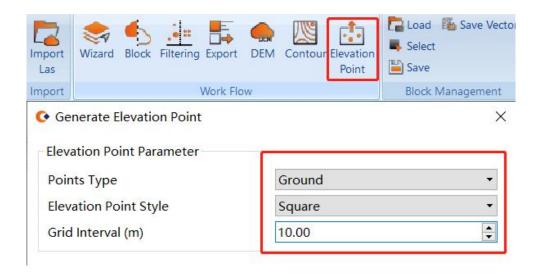


Contour note drawing



Step 6: Generate elevation points: After the notes are drawn on important locations such as ridges by using the above method, it signals that the contour results generation for this area is completed. Next comes the process of generating elevation points. The same as the contours, elevation points can also be generated from both contours and the existing DEM.

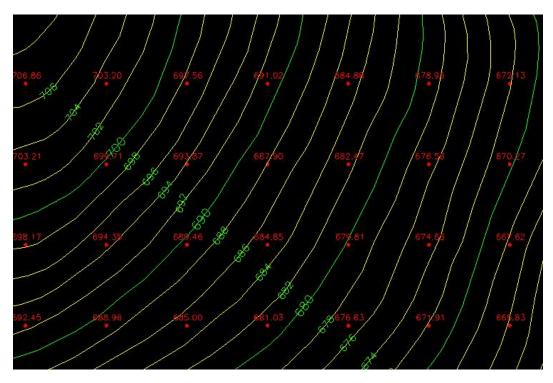
Click the "Generate Elevation Point" function, we can choose "Generate from Point Cloud" or "Generate from DEM" option. Then click "Next" to set elevation point parameters such as grid interval, elevation point style and so on.



Elevation point generation

After the elevation points are generated, we can see the red elevation points on the main view, and after opening the "Text Symbol" option, we can see the text notes of the elevation points.





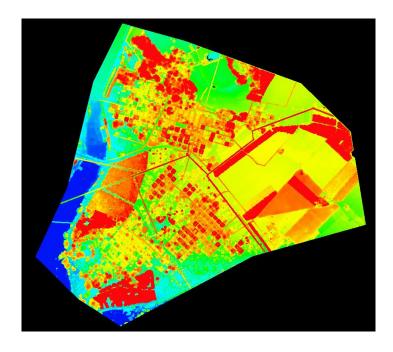
elevation point generation

# 2.1.3. Buildings outline Extraction

### 2.1.3.1. Sample Data

Demo data in this guide is from Chinese village which is ".las" format with 1.94GB.

Demo is fully based on Terrain module.



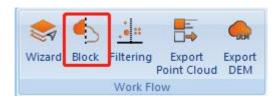


### 2.1.3.2. Steps in Building outline Extraction

- a. Block
- b. Filtering (Including Auto Classification)
- c. Main-View Manual Classification
- d. Buildings outline Extraction
- e. Buildings outline Semi-Edit.

#### **2.1.3.2.1.** Points Block

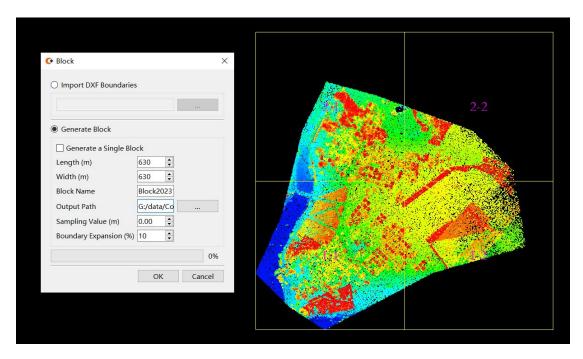
Step 1: Add the point cloud file to be processed in the workspace, click the block button in the terrain module workflow (Figure 5.1), and the block window will pop up.



Step 2: Set the grid size, block name and save path in the pop-up window. When modifying the length and width of a block, the size of the grid in the view changes in real time.

Step 3: Confirm settings and blocking converting.

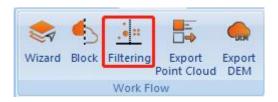




Step 4: Load Grid point cloud.

## 2.1.3.2.2. Filtering

Step 1: Click Filter.



Step 2: Select Filtering block, here we only select left bottom 4 block. If not select, the default is all of blocks.



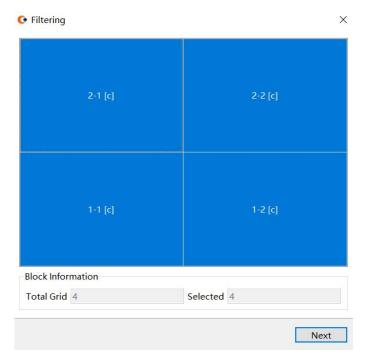


Figure 2-1 select Filtering blocks

Step 3: click next to set filtering configuration. Please set scenarios by actual terrain, here we select plane. If users want to classify buildings and vegetation, it's able to select "buildings" and "vegetation". If users want to select proper configuration, it's recommended to process small block data for configuring setting parameters. For Isolated points filtering please select auto calculation.

Step 4: Click next and generate exported point cloud.



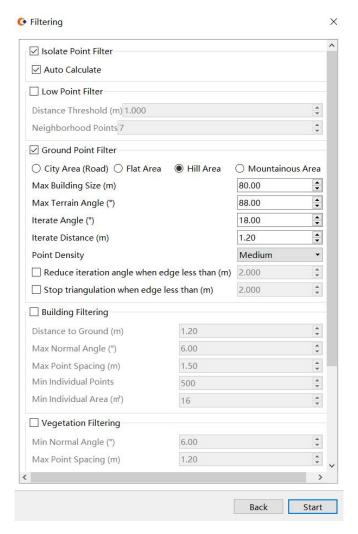


Figure 2-2 Setting Filtering Setting

#### 2.1.3.2.3. Main-View Classification (Manual Edit)

Auto-filtering does not classify points correctly all the time. Depending on the computer configuration, you can choose to load one or a few pieces of data for manual editing. In order to ensure smooth operation, it is recommended to only load one piece of point cloud data at a time.

Step 1: Load grid point cloud, please follow the the preview (default sampled points) point cloud and select blocks to show points cloud.



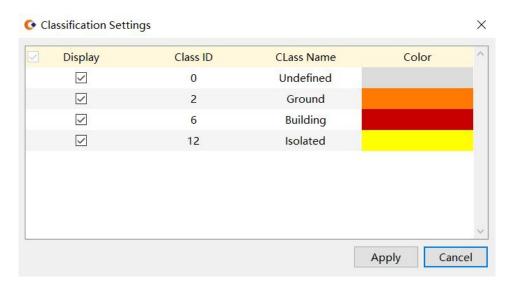


Step 2: Configure colorization mode, using colorization tool to set Elevation/Intensity/Classification to load point cloud. We recommend selecting EDL make texture and shape look better.



Click right icon in shortcut to display different classification. Unselect other and keep buildings to check if classifying is correct.







Step 3: Use Main-View Classification to manually edit point cloud class, please follow the tablet below to input source and target.

Table 2.1 Point Cloud Classification Scenarios

Scenarios	Source	Target
Wrong	Buildings	Non/Vegetation/Ground
Missing	Non	Buildings

# **Main-View Classification Edit Steps:**

a. Click Main-View Classification, set the source and target, here we select them as



Buildings and Non-Classification.

b.Click Rectangular/Polygon/Brush to select region of interest(ROI).

c.There are reverse/forward option for every step. When you finish one ROI, you can close the Main-View Classification, and do next ROI.

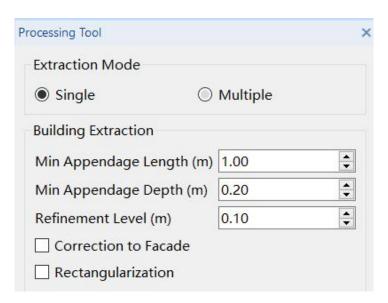


2.1.3.2.4. Buildings outline Extraction

By editing operations, we can obtain finely classified building point clouds. Here we will introduce the process of using these point cloud data to extract building outlines.

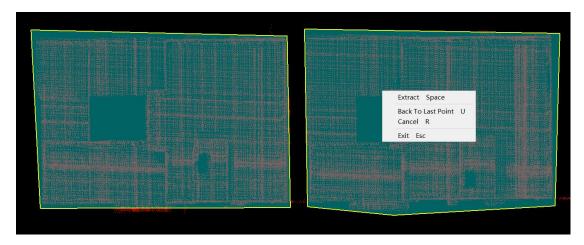
### 2.1.3.2.4.1 Single Building Extraction

Step 1: Open the building outline extraction tool. Click "Building Editing—>Quick Building Extraction" to open the building outline extraction tool, including "Single Building Mode" and "Multiple Building Mode". Here we choose "Single Building Mode".



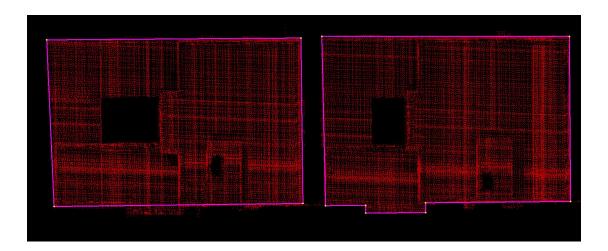
Step 2: In the active view, use the left mouse button to draw polygon areas. After selecting all the building areas to be extracted, click the right mouse button to process these chose building areas.





Select the buildings area

Step 3: Click "Extract" and the software will automatically extract the building outline of the selected polygon area. After the building outline is generated, it will be automatically displayed on the view. The outline color is purple.

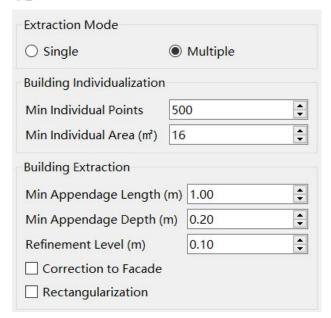


**Buildings Outline** 

# 2.1.3.2.4.2 Multiple Building Extraction

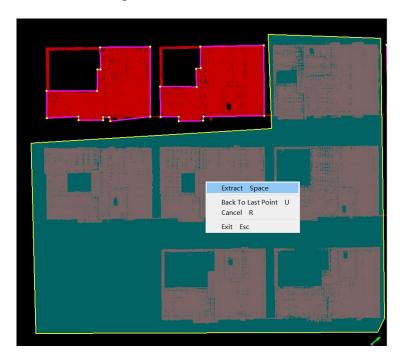
Step1: Open the building outline extraction tool. Click "Building Editing—>Quick Building Extraction" to open the building outline extraction tool, including "Single Building Mode" and "Multiple Building Mode". Here we choose "Multiple Building Mode".





## Multiple Mode

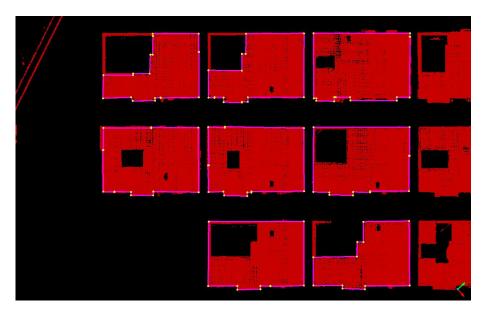
Step2: In the active view, use the left mouse button to draw polygon areas. After selecting all the building areas to be extracted, click the right mouse button to process these selected building areas.



Select the buildings area

Step3: Click "Extract" and the software will automatically extract the building outline of the selected polygon area.





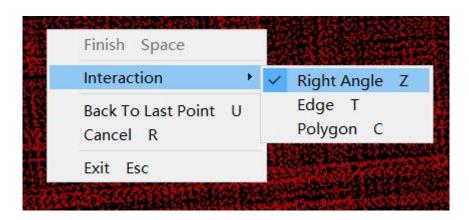
## **Buildings Outline**

#### 2.1.3.2.4.3 Manual Extraction

Step1: Click "Manual Drawing", and right click "Switch Interaction", and you can choose "Right angle", "By edge" and "Polygon" to draw. Here, The default drawing mode is "Right angle".



## **Manual Drawing**

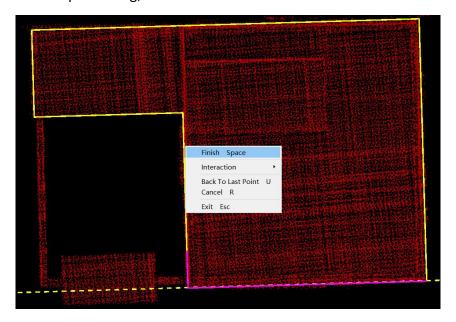


"Right angle" mode

Step2: In the active view, use the left mouse to click two points to determine the

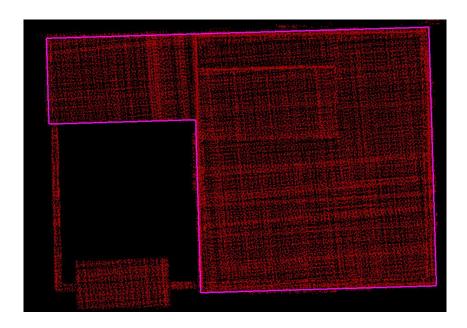


direction of the first edge, and click on other edges in turn, and then click the right mouse button for processing;



Right-angle drawing

Step3: Click the "Finish" button to end the drawing process, and the buildings outline closes automatically.

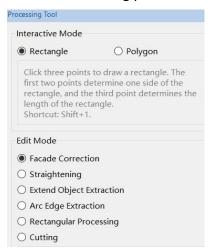


**Buildings** outline



#### 2.1.3.2.5. Buildings Outline Edit

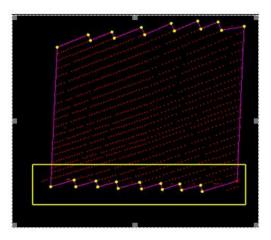
After completing the building outline extraction, manual inspection and detailed editing are often required to output building outline products that comply with specifications. Here we will introduce the editing process of building outlines.

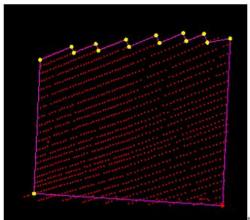


To correct some problematic building outlines more efficiently, the software provides semi-automatic editing tools. The operation process is as follows:

- Step 1: Open Semi-auto edit tool shortcut. Click "Buildings Edit".
- Step 2: Edit as different scenes, here is explained by cases.
- Case 1: Zigzag building outline correction. For this situation, we use the ""Straightening " function in building editing to correct the wrong contours within the range. The editing operation is as follows:
  - a. Click Building Edit.
  - b. Select the polygon interaction mode in the window processing tools on the right.
  - c. Select the ""Straightening " function in editing mode.
  - d. Draw the polygons contained in the area to be modified in the main view, double-click the left mouse button to end the drawing, as shown in the figure the building outline is straightened.



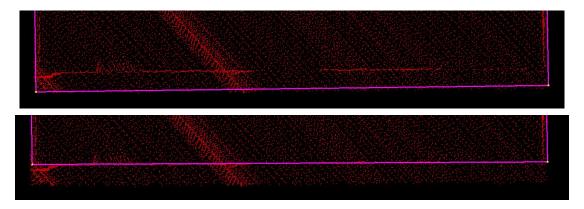




Case 2: Modification of building outline and elevation. For some situations where there is a building facade but the outline does not coincide with the facade, we use the "facade correction" function in the building editor to correct the erroneous outline within the range to a position that coincides with the facade. The editing operation is as follows:

- a. Click Building Edit.
- b. Select the rectangular interaction mode in the window processing tools on the right.
- c. Select the "Facade Correction" function in the edit mode.
- d. Draw the rectangle contained in the area to be modified in the main view.

  After the drawing is completed, the outline is corrected to a position that coincides with the elevation.

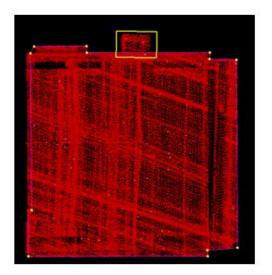


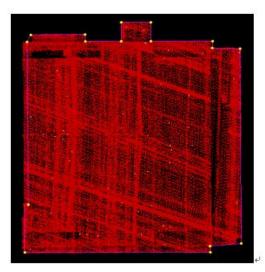
Case 3: Extend Object Extraction. For situations where the outline does not include



some rectangular appendages under the eaves such as balconies, we use the "Extending Extraction" function in the architectural editor to automatically correct the outline to a vector line containing the rectangular extended object. The editing operation is as follows:

- a. Click Building Edit.
- b. Select the rectangular interaction mode in the window processing tools on the right.
- c. Select the "Extend Object Extraction "function in edit mode.
- d. Draw the rectangle contained in the modified area in the main view. When the drawing is completed, the contour line is automatically corrected to a vector line containing the rectangular extension object.



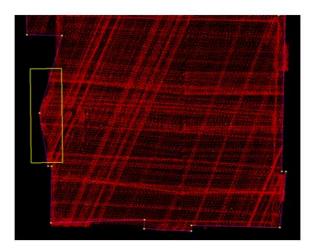


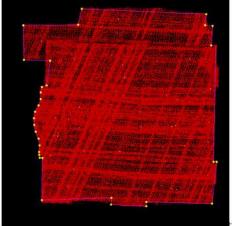
Case 4: Arc Edge extraction. For the case where the contour line of the arc-shaped part of the building is incorrectly extracted, we use the "Arc Edge Extraction" function in the architectural editor to automatically correct the contour line to a position that fits the arc-shaped building point cloud. The editing operation is as follows:

- a. Click Building Edit.
- b. Select the rectangular interaction mode in the window processing tools on the right.



- c. Select the "Arc  $Edge\ Extraction"$  function in the edit mode.
- d. Draw the rectangle contained in the area to be modified in the main view. When the drawing is completed, the contour line is automatically corrected to a position that fits the arc-shaped building point cloud.



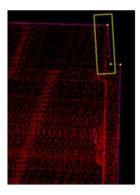


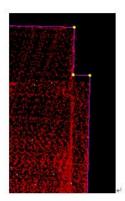
Case 5: Rectangular Processing.

For situations where the corners of building point clouds or the outline of extended objects are not right angles, we use the "Rectangular Processing" function in architectural editing to automatically correct the contours to right angles. The editing operation is as follows:

- a. Click Building Edit.
- b. Select the rectangular interaction mode in the window processing tools on the right.
- c. Select the "Rectangular Processing" function in the editing mode.
- d. Draw the rectangle contained in the area to be modified in the main view. When the drawing is completed, the outline will be automatically corrected to a right angle.

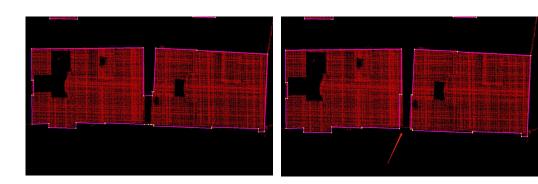






Case6: Cutting. By using the "Cutting" function in architectural editing we can cut the buildings outline into two closed buildings outlines. The editing operation is as follows:

- a. Click "Building Edit";
- b. Select the "Cutting" function in the editing mode;
- c. The default cutting width is 0m, and here we set the cutting width as 1.8m;
- d. Draw a multi-segment line across the buildings outline in the main view, and double click to finish the drawing process, then the buildings outline is cut into two closed buildings outlines.



In addition, if automatic building editing cannot make corrections, you can use vector editing tools such as trimming, adding points, deleting points, and moving points to manually edit the building outline. Please select Dot Symbol to show each point node and edit the outline.





## 2.1.3.3. Building Outline Result

After passing the buildings outline editing check, right click the buildings outline node and select "Export" to export the building outline line as two dimensional multisegment lines, and then the buildings outline production results meeting the requirements can be obtained.



# 2.2. Terra Parameter description

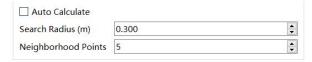
# 2.2.1. Isolated point (noise) filtering

#### Algorithm Principle

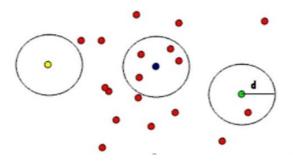
Within a certain radius (Search Radius), if the number of neighborhood points of the point is less than the set threshold, the point is determined as an isolated point.

The parameters in the figure below indicate that within a radius of 0.3m, if the number of neighborhood points is less than 5, the point is determined as an isolated point.





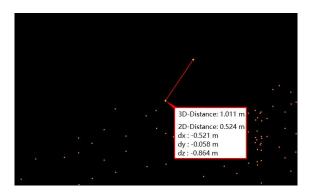
As shown in the figure below: The number of neighborhood points of the green point within radius d is 1, indicating that the green point is an isolated point. The yellow point has no neighborhood points within the d radius, so it's also an isolated point.



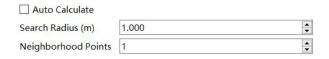
schematic diagram

#### Real Scenario 1

The point in the figure below is 1.011m away from closest point.

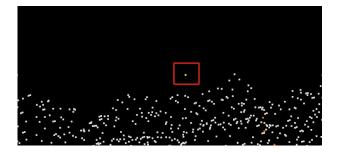


Set the search radius to 1.0m



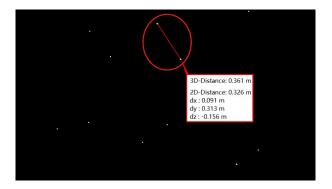
Within the radius of 1.0m, the number of neighborhood points is 0, so this point is marked as an isolated point.





#### Real Scenario 2

Noise is not limited to the isolated single point, for example, isolated point group (2 or more than 2 points), as in the figure below:



If you want to remove the isolated point group formed by these two points, you can set the parameters as below:



#### Auto Calculate

When you check "Auto Calculate" box, the search radius is 2-3 times the average point spacing, and the neighborhood points parameter is set to 5. This rule is suitable for most scenarios.



# 2.2.2. Low point filtering

Algorithm Principle



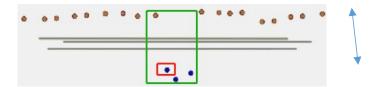
### The parameters in the figure below:



Take a point as the center to search the three points closest to it, including the current check point, where the height difference between the point with the second largest elevation value and the point with the largest elevation value is more than 1m, then this point is determined as the low point.

Due to equipment or working environment, resulting in partial points below the ground point, which is the low point.

As shown in the figure below: Take the point in the red box as the center to search the three points closest to it. The height difference between the point with the second largest elevation value (in the red box) and the point with the largest elevation value (the top point in the green box) is more than 1.5m, then this point (in the red box) is determined as the low point.

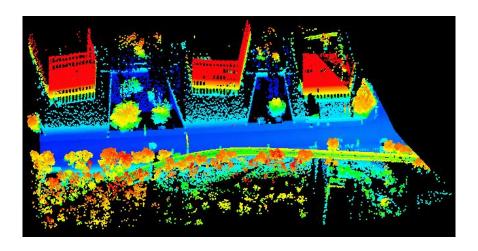


Low point is not limited to the single point but also point group (2 or more than 2 points).It is recommended to set the "Neighborhood Points" to  $\geqslant$  2 , and set the "Distance Threshold (m)" to 1m.



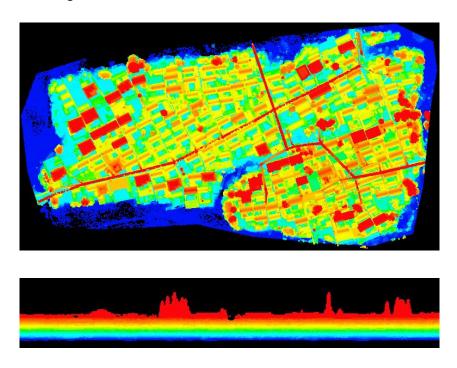
# 2.2.3. Ground point filtering(地面点)

Ground point filtering contains four scenarios:city road, flat ground, hills, mountainous areas. City hardened pavement:



city road

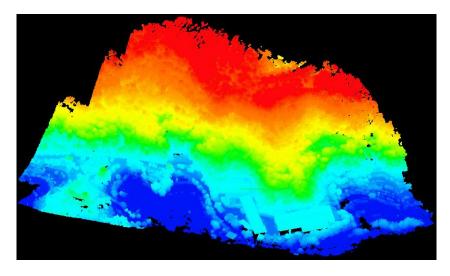
An area with more gentle terrain:

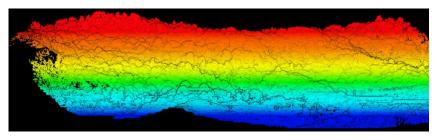


flat ground

An area with highly undulating terrain:







hills
An area of very steep terrain with distinct ridges:







#### mountainous areas

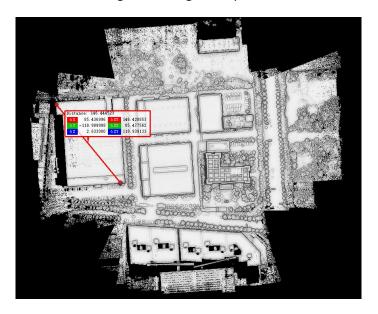
#### 2.2.3.1. Flat Area

Description of parameters

☑ Ground Point Filter		
○ City Area (Road) ○ Flat Area ● Hill Area	O Mountaine	ous Area
Max Building Size (m)	80.00	
Max Terrain Angle (°)	88.00	
Iterate Angle (°)	18.00	
Iterate Distance (m)	1.20	
Point Density	Medium	*
☐ Reduce iteration angle when edge less than (m)	2.000	<b>+</b>
Stop triangulation when edge less than (m)	2.000	<b>‡</b>

# 1) Max Building Size (m)

The diagonal dimensions of the largest building in the point cloud data.



# Parameter setup suggestion:

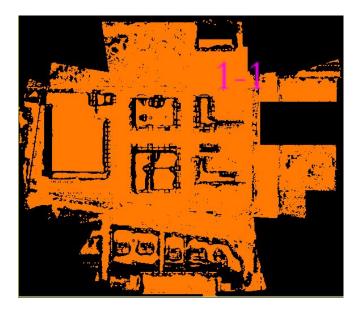
Generally,set according to the diagonal dimensions of the largest building. If the building points are misclassified as the ground points, increase the maximum building size.

# Notes:

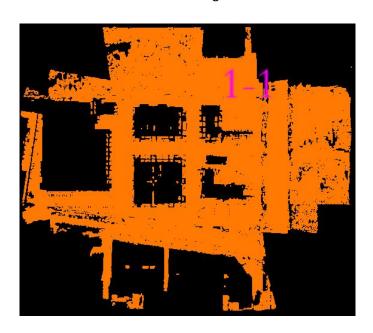
The maximum building size cannot be set too large, the value cannot exceed 1/2 of the side length of the point cloud data outsourcing box. For example, the size of the point cloud data outsourcing box is 60m×70m,the maximum building size can be increased to no more than 30m.



The maximum building size cannot be set too small, the minimum value is generally 10-20 times of the point cloud density.



maximum building size 5m

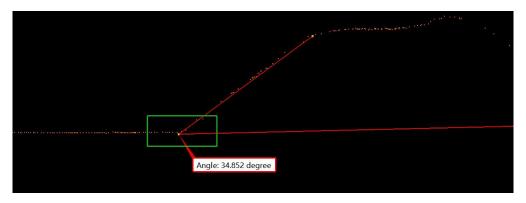


maximum building size 153m

# 2) Max Terrain Angle (°)

The angle between the steepest terrain and the ground.

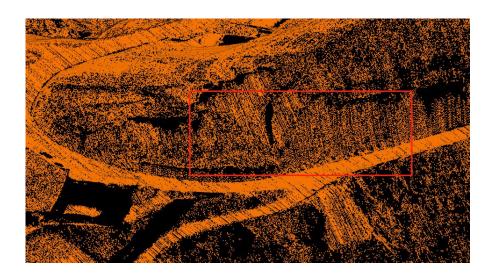




Max Terrain Angle parameter has great influence on the results of complex mountain terrain.



Max Terrain Angle 25  $^{\circ}$ 



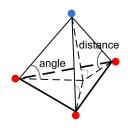
Max Terrain Angle 40  $^{\circ}$ 

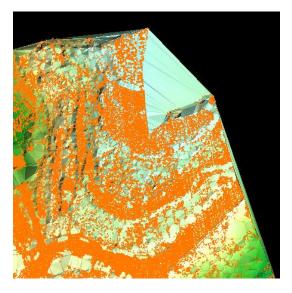
# 3) Iterate Angle (°)

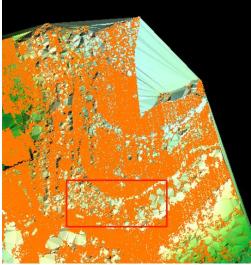
Refers to the largest Angle of the three angles formed by the point to be inserted to the



three vertices of the closest triangle mesh. See the angle in the figure below:





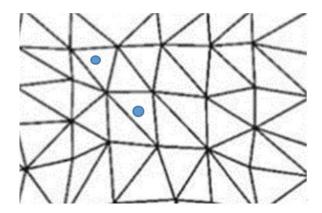


Iterate angle 6°

Iterate angle 30°

# 4) Iterate Distance (m)

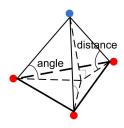
The iterative progressive triangulation algorithm is a process of gradual encryption, as shown in the figure below, the principle is that new points are constantly added to the triangle mesh, we call the triangles in it triangle mesh surface.



Iterate Distance (m): Refers to the distance between the point to be inserted and its closest

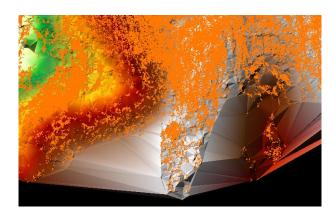


triangle mesh surface. See the distance in the figure below:

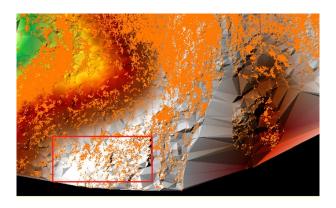


## Parameter setup suggestion:

Iterate distance parameter has little influence on the results, which is generally reasonable between 1.2-2.0m.



Iterate distance 1.2m

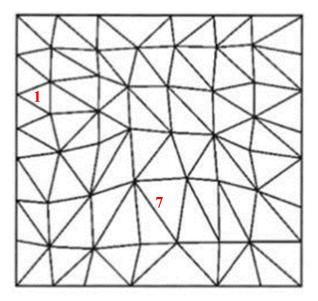


Iterate distance 3m

# 5) Reduce iteration angle when edge less than (m)

When you check "Reduce iteration angle when edge less than (m)" box, ground points whose distance is less than the set threshold will not continue to generate triangle mesh. Can smooth the display of ground points.



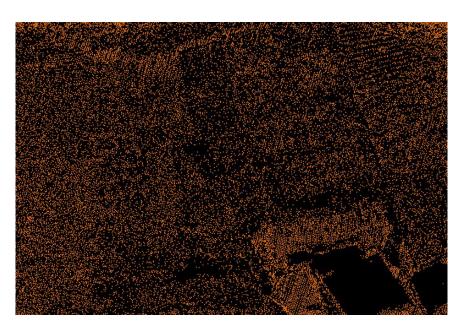


Iteration angle length threshold (m)

# Parameter setup suggestion:

The larger the iteration angle length threshold, the larger the interval between generated ground points, and the smaller the number of ground points.

The default value is 2m, which means that the points whose distance is less than 2m will not continue to generate triangle mesh, so it's not a ground point.

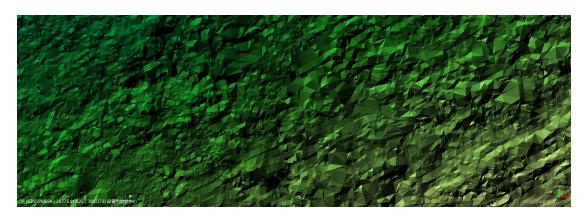


iteration angle length threshold 1 m

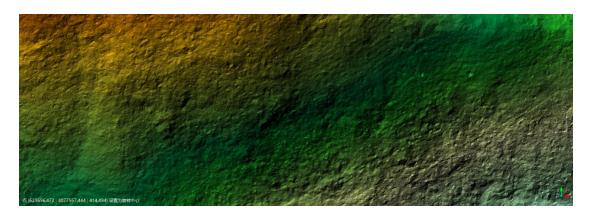




iteration angle length threshold 7 m



not check "Iteration angle length threshold (m)" box



check "Iteration angle length threshold (m)" box

# 6) Stop triangulation when edge less than (m)

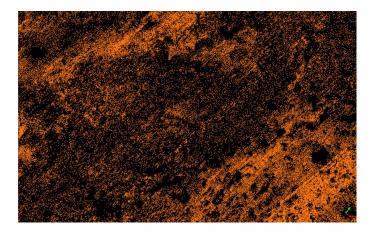
When you check "Stop triangulation when edge less than (m)" box, ground points whose distance is less than the set threshold will not continue to generate triangle mesh. This can prevent that the generated ground points are too dense.



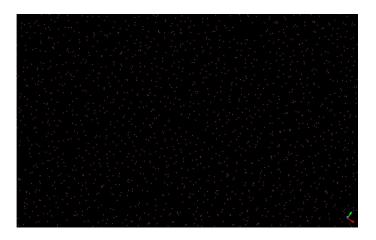
## Parameter setup suggestion:

The larger the iteration angle length threshold, the sparser the generated ground points.

The default value is 2m, which means that the points within the triangle mesh whose three sides are all less than 2m will not constitute the ground points.

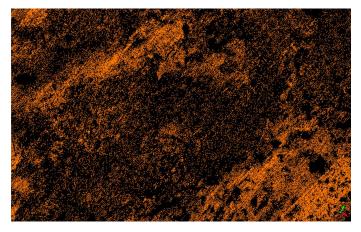


When the "Stop triangulation when edge less than (m)" is not enabled



Stop triangulation when edge less than 1m





Stop triangulation when edge less than 0.05m

**Note**: When this parameter is set too large, the ground points will be too sparse, and some ground feature points will be missing, which will affect the accuracy of the terrain.

Parameter setup suggestion

#### 1.Gentle terrain

- 1) It is suggested that iteration angle be adjusted preferentially, if a large number of other object points are misclassified into ground points, reduce the iteration angle, the reverse is also true, the range of each adjustment is recommended to be 3°.
- 2) For data with larger buildings, also need to consider the maximum building size, the diagonal length of the largest building is measured as the input parameter.
- 3) Because the terrain is relatively gentle, the maximum terrain angle has little influence on the results, leave this as the default.

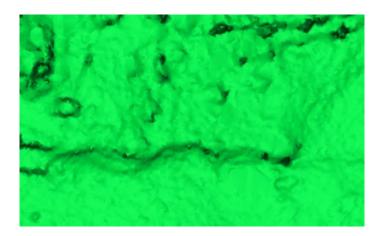
## 2. Non-gentle terrain

1) It is suggested that iteration angle be adjusted preferentially, if a large number of other object points are misclassified into ground points, reduce the iteration



angle, the range of each adjustment is recommended to be 3°. The reverse is also true.

2) The lack of ground points in some steep slope areas is generally due to the small value of max terrain angle, increase the max terrain angle, the range of each adjustment is recommended to be 10°. The reverse is also true(For example, there are a lot of spikes, as shown below).



3) For data with larger buildings, also need to consider the maximum building size, the diagonal length of the largest building is measured as the input parameter.

#### **Notes:**

- 1) In both cases above, the iterate distance parameter has little influence on the results, which is generally reasonable between 1.2-2.0m.
- 2) For point cloud data without buildings, leave the maximum building size as the default. Other parameters should be adjusted according to the rules above.

#### 2.2.3.2. City Area (Road)

City road classification uses the plane detection algorithm, and classifies the ground points through the constraints of the number of points, topographic relief and other parameters. This algorithm is used for local area plane detection, and classifies the inner points of the plane (if the distance from the point to the plane is less than the distance threshold, it is called the inner point of the plane) as the ground points, which is applied to the classification of city hardened pavement of vehicle-mounted point cloud.



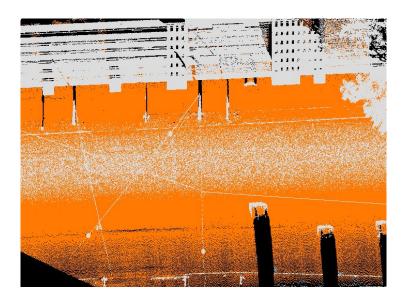
Description of parameters

	O Hill Area	<ul> <li>Mountainous Area</li> </ul>
Near Radius (m)	0.25	<u>*</u>
Points Number Threshold	50	
Plane Points Ratio	0.50	-
Result Sampling (m)	0.05	<b>‡</b>

# 1) Near Radius (m)

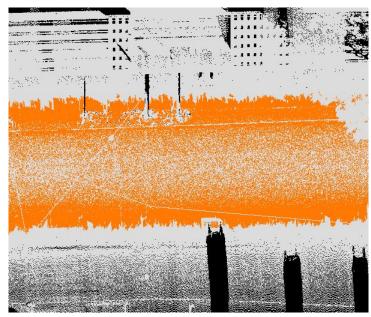
Neighborhood search radius, used for region growth, as input data for plane detection. For normal usage, set to 3-5 times the scan line interval. If the value is too small, the inner points of the plane cannot be correctly detected. If the value is too large, plane detection efficiency is reduced.

As shown below, the scan line interval of the test data is 0.04m, when set the near radius to 0.04m, the pavement on both sides of the road is not properly classified.



near radius 0.15m

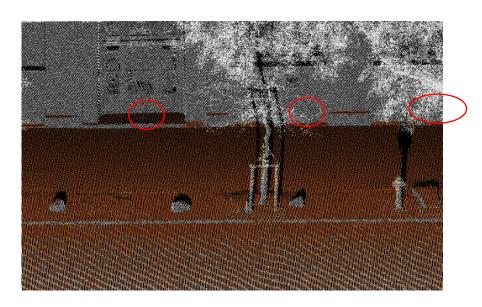




near radius 0.04m

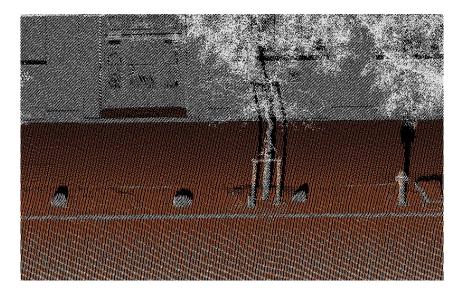
# 2) Points Number Threshold

This parameter is used to constrain the number of points of the local plane being detected. If the number of points is less than this value, the point cloud in this part is discarded. As shown below, if the value is too small, there will be a smaller set of plane points classified as ground points by mistake.



Points number threshold 10

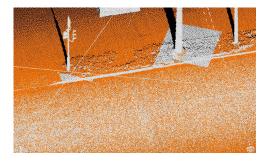




Points number threshold 100

#### 3) Plane Points Ratio

The proportion of points with planar characteristics in a point set is the plane points ratio, used to determine whether the detected local plane is a valid plane. If the calculated value of plane points ratio of the detected local plane is greater than the set threshold, the detected local plane will be classified as the ground points. This parameter is in the range [0,1]. If the value is set large, some rough surfaces will not be classified as ground points, and the classification effect will be more refined.



plane points ratio 0.9

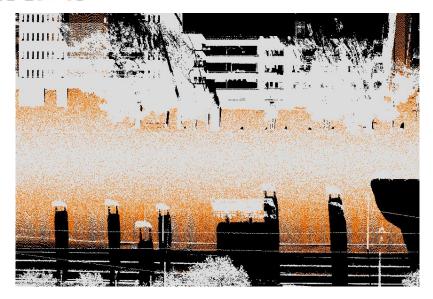


plane points ratio 0.5

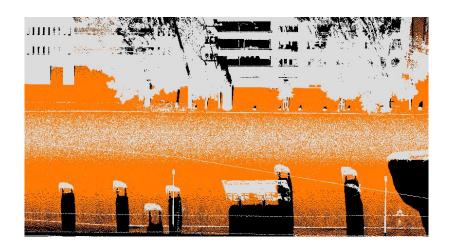
## 4) Result Sampling (m)

The distance threshold used to sample the point cloud. On the premise of meeting the accuracy and density of the result point cloud, thinning the point cloud to improve the efficiency of the algorithm. When you check "Result Sampling (m)" box, can set the value according to the desired result accuracy.





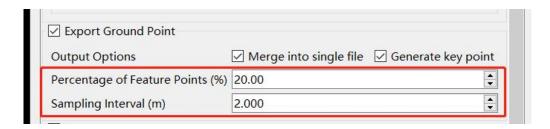
Result sampling 0.05m



Result sampling 0.01m

# 2.2.4. Key Points Parameter Description

When exporting ground points, the ground points are sampled according to the principle of preserving as many features as possible to obtain the feature points that can reflect the fluctuation changes of the terrain.



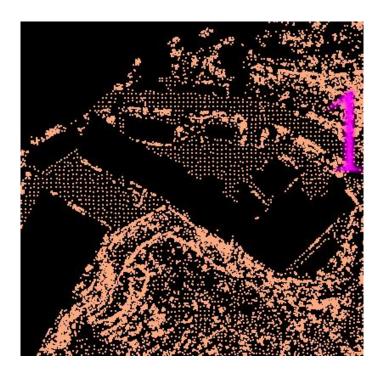


# 1) Percentage of Feature Points (%)

The points are sorted according to the curvature from large to small, and the points with larger curvature are exported in percentage, which are the feature points.

# Parameter setup suggestion:

For normal usage, leave this as the default of 20%, the larger the value, the more points.



Percentage of feature points 20% (613019 feature points)





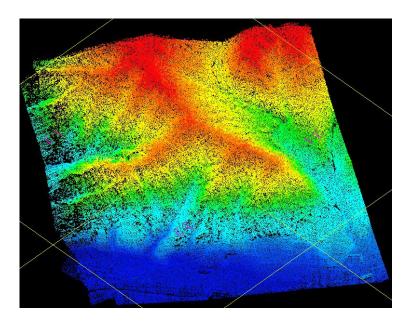
Percentage of feature points 80% (1504292 feature points)

## 2) Sampling Interval (m)

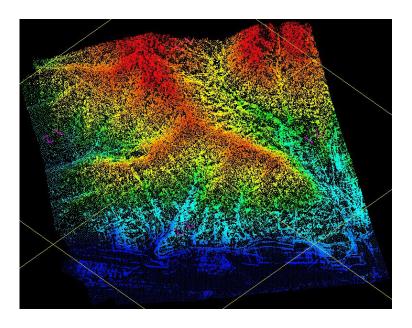
The size of the grid to sample the point cloud.

Parameter setup suggestion:

For normal usage, leave this as the default of 2m, the larger the value, the less points.



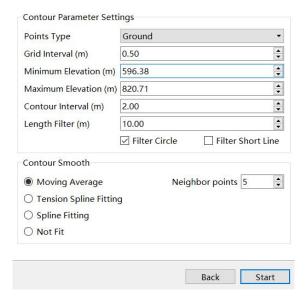
Sampling interval 2m (613019 feature points)



Sampling interval 6m (386666 feature points)



#### 2.2.5. Contours Generation



- Contour Parameter Settings
- 1) Grid Interval (m)

Generate contours using DEM automatically.( Set according to the actual accuracy)

2) Minimum Elevation (m)

Minimum elevation value of point cloud data.( software automatically obtains this value)

3) Maximum Elevation (m)

Maximum elevation value of point cloud data.( software automatically obtains this value)

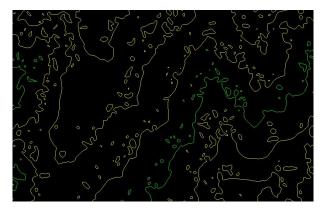
4) Contour Interval (m)

The actual height difference between the two contour surfaces.

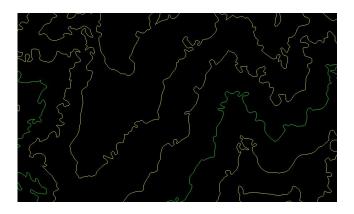
5) Length Filter (m)

When generating contours, there are some little mounds or buildings on the ground, but these are not actual, real terrain, so need to set the this parameter to filter. There are generally two types of abnormal contours that need to be filtered, Circles and short lines, as shown below:





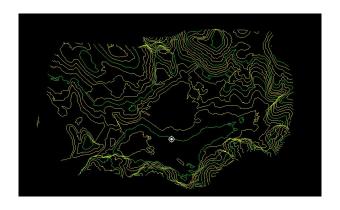
Before filtering



After filtering

Results show:







#### 2.2.6. Contours Smooth

The contour smoothing function is mainly used to generate smooth contours.

Based on the already generated contours, we can use these contour smoothing algorithms shown in the following figure(including moving average, tension spline fitting and spline fitting) to smooth the contours. The moving average algorithm is selected by default.

#### 1) Moving Average

[Description]: The center point is smoothed by several adjacent points.

[Parameter Setting]: The default number of the neighbor points is 5, that is 2 points on each side plus 1 center point. This parameter can only be set to an odd number greater than or equal to 3.

[Applicable scenes]: Contours with dense vertices, such as contours generated based on the DEM with 0.5m resolution.

#### 2) Tension Spline Fitting

[Description]: The center point is smoothed by spline interpolation between two adjacent vertices.

[Parameter Setting]: Not required.

[Applicable scenes]: Contours with sparse vertices, such as contours generated based on the DEM with the resolution greater than or equal to 5m.

#### 3) Spline Fitting

[Description]: Contours are automatically segmented, and for each segment a cubic spline function is used for fitting. The purpose is to smooth the center vertex and to keep the connection of each segment smooth.

[Parameter Setting]: Not required.

[Applicable scenes]: Contours with sparse vertices, such as contours generated based on the DEM with the resolution greater than or equal to 5m.

#### 4) Not Fitting

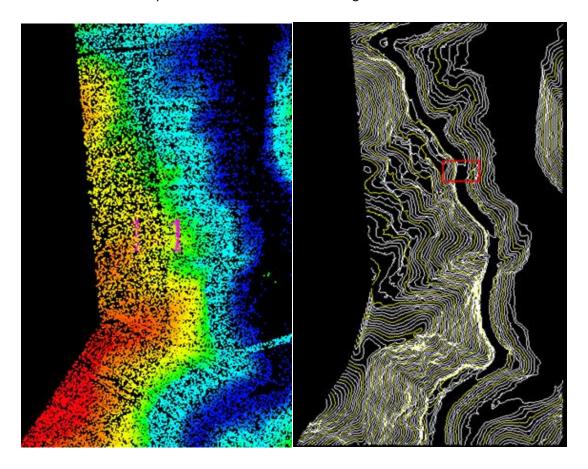
[Description]: The contours result are not smoothed.

[Parameter Setting]: Not required.

[Applicable scenes]: Contours generated based on control points.



We have generated contours based on airborne point clouds, and in the following we will use this as an example to show the effects of each algorithm:



Input point cloud and the effect of not fitting method (Partial display in red box)



Not fitting (Partial display in red box)





Moving Average (Partial display in red box)



Tension Spline Fitting (Partial display in red box)



Spline Fitting (Partial display in red box)

### 2.2.7. Elevation points Generation

[Principle]: Elevation points are obtained by sampling the point cloud at a certain distance, and they can can reflect the elevation change of the scene.



[Parameter Setting]

### 1) Points Type

You can select the ground points or all points, and the ground is selected by default.

#### 2) Elevation Point Style

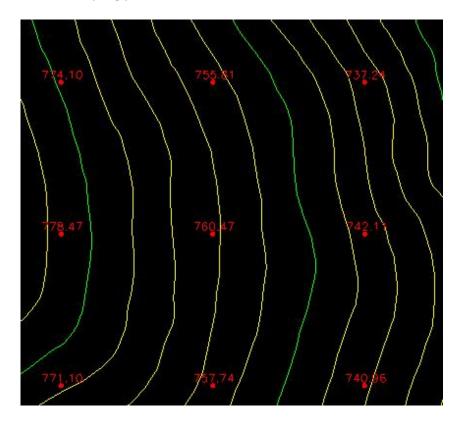
The shape of the sampling points distribution, and now it supports both square and diamond



shapes.

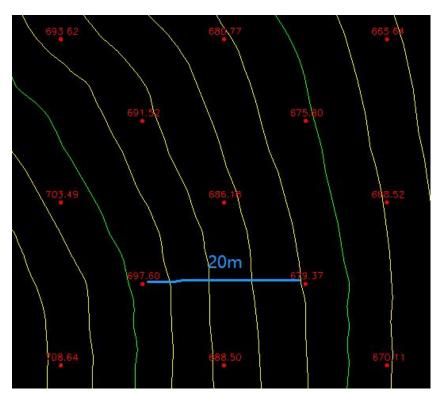
## 3) Grid Interval (m)

The interval of the sampling points distribution.



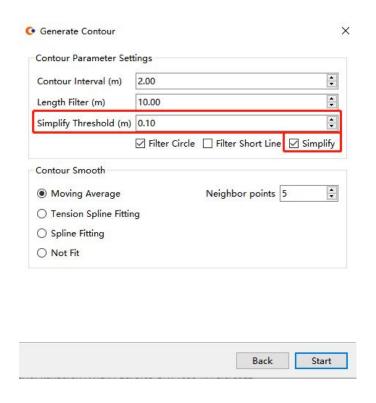
square





Diamond

# 2.2.8. Contours Simplify



Contours simplify setting



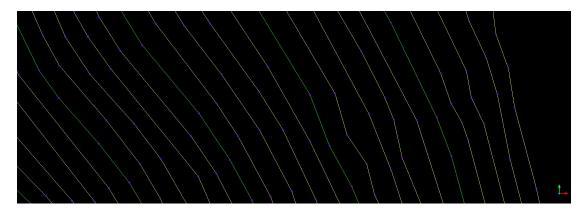
[Simplify Threshold (m)]: The distance between the current point and the two points around it in a straight line.

[Parameter Setting]: The default value is 0.1m, and the recommended value range is [0.05,0.1]. Actually the value can be set from 0.01 to 1.

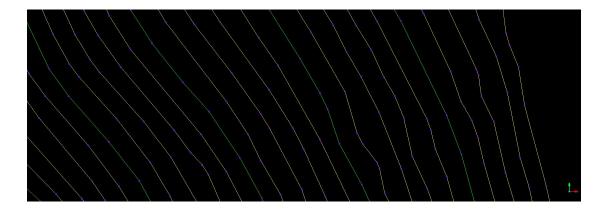
[Applicable scenes]: For contours with dense vertices, this algorithm is used to reduce the number of vertices. The specific performance is as follows. Very few points are retained in the flat region, and if some points form a collinear segment, only the two endpoints of this line segment will be preserved. In the region with large fluctuations, more corner points can be preserved to ensure that the basic shape remains unchanged.

The following is the simplification effect of a dense contour with the default simplify threshold of 0.1m.

The smaller this parameter is set, the more points are retained and the contours are smoother, but at the same time the size of data will be relatively larger.

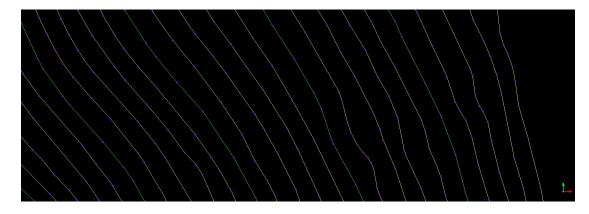


0.2m

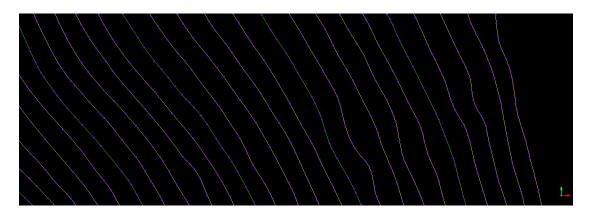




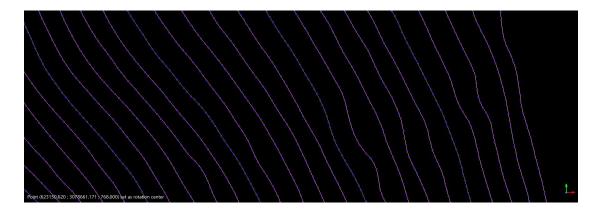
0.1m



0.05m



0.01m

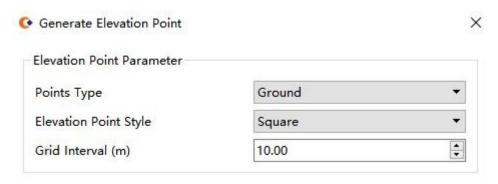


Nonuse simplification

## 2.2.9. Generate elevation points

Elevation points are obtained by sampling the point cloud at a certain distance, which can reflect the elevation changes of the scene.





### 5) Points Type

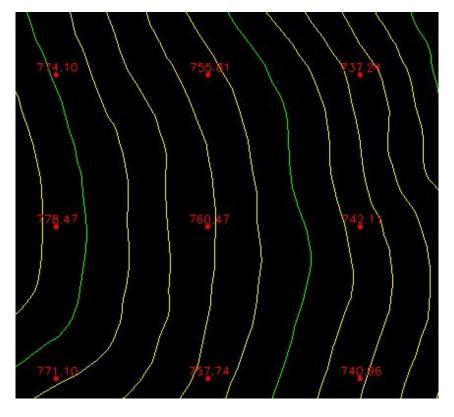
You can select the ground points or all points, the ground is selected by default.

### 6) Elevation Point Style

The shape of the sampling points distribution, now supports both square and diamond shapes.

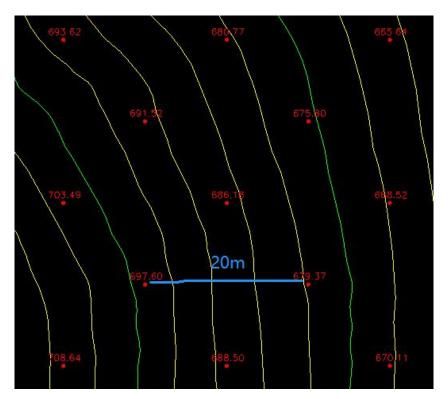
### 7) Grid Interval (m)

The interval of the sampling points distribution.



square





Diamond



### 2.2.10.Building Filtering

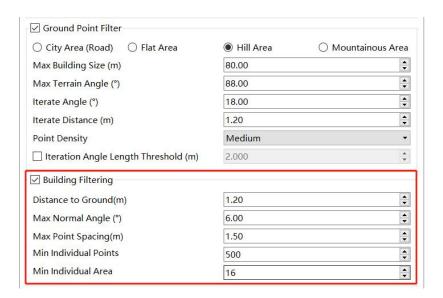


Figure: Buliding Filting - parameter settings

#### 2.2.10.1. Distance from ground

The height of the lowest point of a building from the ground is mainly to avoid the impact of plane points close to the ground (such as cars, small rocks, etc.) on the subsequent extraction results of building contours.

#### Parameter settings:

The default value is 1.2. The value range is [0.1,3]. The recommend value range is [1,2.5]. If the scene is a low rural house, this value can be set to 1.2; The height of urban buildings is generally above 3m, and this value can be set to 2.5 to ensure the accuracy of building contour extraction.

**Note:** Setting this parameter will result in missing points at the bottom edge of the building, but it has little impact on building contour extraction.



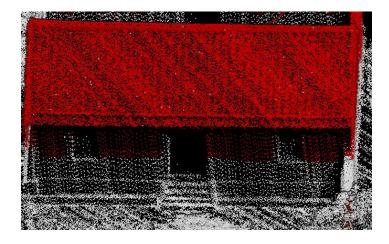


Figure: Distance from ground is 1.2 meters



Figure: Distance from ground is 2.5 meters

#### 2.2.10.2. Maximum normal angle

The angle value between the normals of adjacent points in a single building. The rougher the building surface, the larger the value; the smoother the extracted building surface, the smaller the value. If the value is set too large, some building accessory points will be mistakenly classified into building points. The parameter value needs to be adjusted according to the actual situation.

#### **Parameter settings:**

The default value is 6  $^{\circ}$  . The value range is [1,30], which must be set according to the thickness of the roof point cloud.

The current recommended values are as follows:



When the building surface is rough and uneven and the thickness of the point cloud is about 4-6cm, the value can be set to 6; When the thickness is greater than 6cm, the value can be set to more than 6;

When the thickness of the point cloud is about 2-3cm, the value can be set to 3;

When the building surface is smooth and the thickness of the point cloud is below 2cm, the value can be set to 1 or 2;

The rougher the building surface, the larger the value; the smoother the extracted building surface, the smaller the value. If it is found that there are many points on the roof of the building that are not correctly classified, the value needs to be appropriately increased by  $1-2^{\circ}$ .

The thickness of the roof point cloud can be measured through the section view, as shown in the figure:

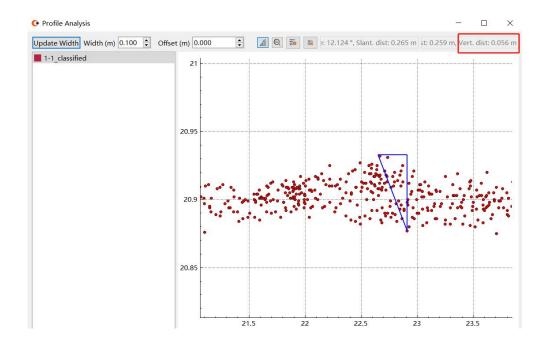


Figure: the thickness of the roof

There are multi-layer point clouds on the roof here, the thickness is also 5-6cm, and the data is relatively rough. At this time, the maximum normal Angle is set to 6°.





Figure: Maximum normal angle is 6°



Figure: Maximum normal angle is 3°



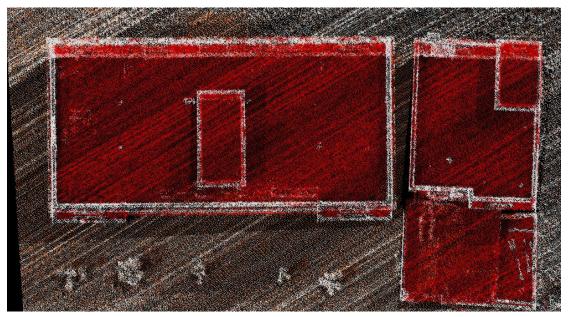


Figure: Maximum normal angle is 1°

#### 2.2.10.3. Maximum point spacing

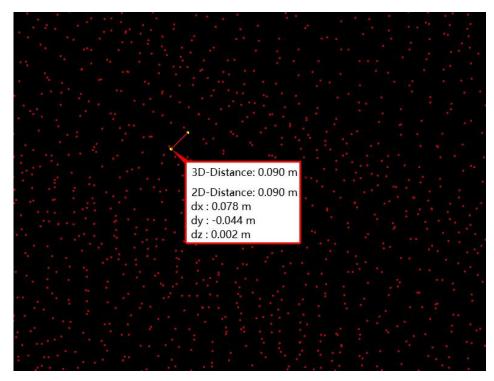
The maximum distance allowed between adjacent building points controls the integrity of building extraction.

#### **Parameter settings:**

The default value is 1.5. The value range is [0.2.6]. The recommende value range is [1,1.5].

The parameter setting should also refer to the point spacing of the building surface, if the setting is too small, the building extraction will be incomplete, and if the setting is too large, the two buildings that are closer together will be mistakenly divided into one.





2.2.10.4. Minimum number of individual points

The number of points occupied by the smallest building in the actual scene, which is the minimum number of points required to form a single building. Buildings with fewer points than the set value will be discarded. By setting this parameter, some ground objects that are mistakenly divided into buildings, such as vehicles with fewer points, shrubs, etc., can be removed.

#### **Parameter settings:**

The default value is 500. The value range is [20,2500].

By measuring the point density and the projected area of the building, the number of points of a single building can be estimated by multiplying the results of the two. The minimum number of points of a single building can be set according to the actual needs.

#### 2.2.10.5. Minimum area of individuals

The number of points occupied by the smallest building in the actual scene, which is the minimum number of points required to form a single building. Buildings with fewer points than the set value will be discarded. By setting this parameter, some ground objects that are mistakenly divided into buildings, such as vehicles with fewer points, shrubs, etc., can be removed.



### Parameter settings:

The default value is 16. The value range is [1,40].

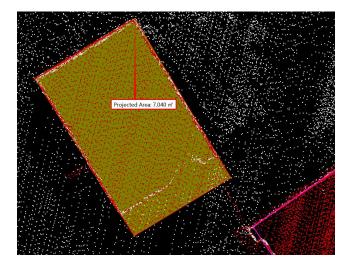


Figure: Actual measured area

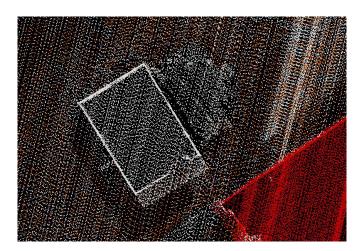


Figure: Minimum area of individuals is 16, not classified as a building



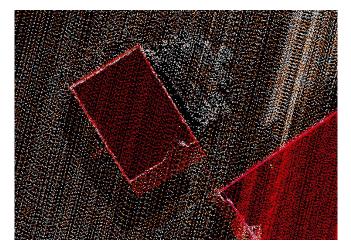


Figure: Minimum area of individuals is 6, classified as a building

### 2.2.11. Vegetation Filtering

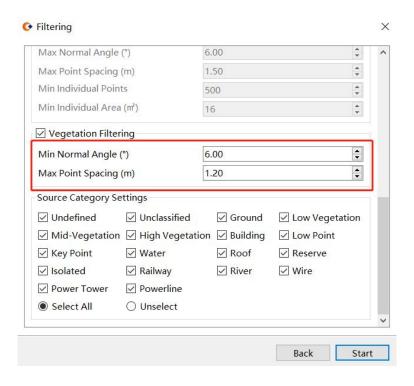


Figure: Vegetation Filtering - parameter settings

#### 2.2.11.1. Minimum normal angle

Unlike building filtering, vegetation filtering is extracted according to surface roughness, so it is necessary to limit the minimum normal angle between adjacent vegetation points. The rougher the surface, the larger the setting value; The smoother the surface, the smaller the

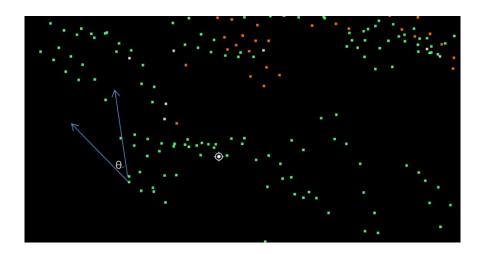


setting value.

#### Parameter settings:

The default value is 6. The value range is [1,30]. The recommend value range is [6,10].

The larger the value is, the more vegetation is to be omitted in classification. The smaller the value, the more surface will be mistaken for vegetation. If it is found that the surface is misdivided into more vegetation, it is appropriate to increase  $1-2^{\circ}$ .



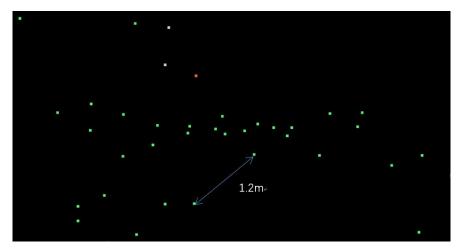
2.2.11.2. Maximum point spacing

The maximum distance allowed between adjacent vegetation points. Mainly affects the integrity of vegetation classification. Setting it too small may result in incomplete extraction of individual vegetation, while setting it too large may cause adjacent unclassified points to be mistakenly divided into vegetation.

#### Parameter settings:

The default value is 1.2. The value range is [0.2,6]. The recommend value range is [1,1.5]. The parameter setting should also refer to the point spacing of the vegetation. Take the maximum distance between the points.





### 2.2.12. Extracting building Outline

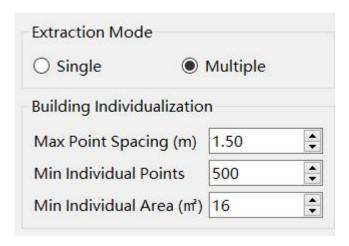


Figure: Extracting building Outline - parameter settings

#### Minimum number of individual points

The number of points occupied by the smallest building in the actual scene. Buildings with fewer points than the set value will not be extracted with outlines.

#### Parameter settings:

The default value is 500. The value range is [20,2500].

By measuring the point density and the projected area of the building, the number of points of a single building can be estimated by multiplying the results of the two. The minimum number of points of a single building can be set according to the actual needs.



Minimum area of individual

The projected area of building in the actual scene, and buildings with an area smaller than this value will not extract outline.

#### **Parameter settings:**

The default value is 40. The value range is [1,40].

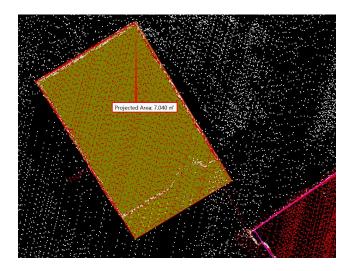


Figure: Actual measured area

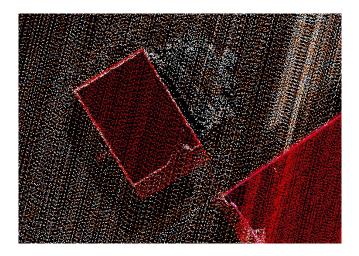


Figure: Minimum area of individuals is 16, no outline was extracted





Figure: Minimum area of individuals is 6, outline was extracted

#### [Buildings outline extraction - Parameter Description]:

建筑轮廓提取		
附属物最小长度 (m)	1.00	•
附属物最小深度 (m)	0.20	•
精细化程度 (m)	0.20	•
□修正到立面		
□矩形化		

Buildings outline extraction parameters

Minimum length of the attachment

It refers the minimum length of the rectangular building attachments in the actual scene. When extracting the buildings outline, the outline of the part of the building attachment whose length is less than this set value will not be extracted.

#### Parameter settings:

The default value is 1.0m, and the value range is [1,2]. In the actual setting, you can set it according to the actual length of the attachment.

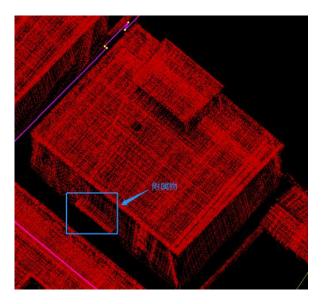
Minimum depth of the attachment



It refers the minimum allowable depth of a rectangular attachment. When extracting the buildings outline, the outline of the part of the building attachment whose depth is less than this set value will not be extracted.

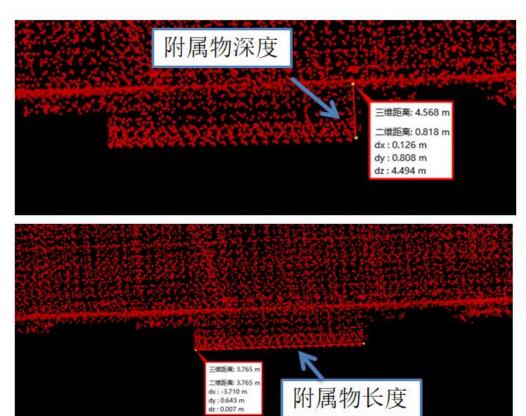
### Parameter settings:

The default value is 0.2m, and the value range is [0.2,0.5]. In the actual setting, you can set it according to the actual depth of the attachment.

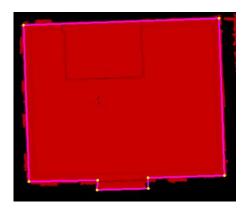


Rectangular attachment



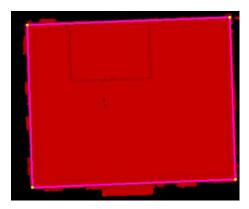


The parameters of the rectangular attachment

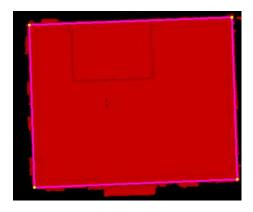


The extraction effect of the default parameter





The minimum length of the attachment is 3.8m



The minimum depth of the attachment is 0.9m

#### Simplification degree

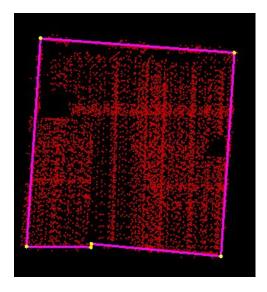
It refers the maximum allowable distance between two adjacent approximately parallel outlines. If the actual distance is bigger than this set value, a short edge will be added between the two parallel lines.

#### Parameter settings:

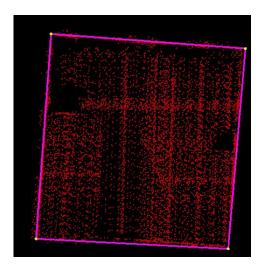
The default value is 0.2m, and the suggested value range is [0.1,0.5].

As the following picture shows, the default simplification degree is 0.2m, but the distance between the two adjacent lines in the picture is 0.3m, which is bigger than 0.2, so at last there is a short edge between the two lines. When the simplification degree is adjusted to 0.3m, then the short edge will disappear.





The simplification degree is 0.2m



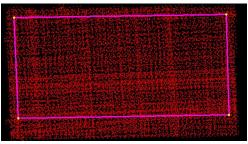
The simplification degree is 0.3m

#### Eave correction

This option is not checked by default, and you can choose whether to check it according to the actual requirement. If not checked, the building outline is extracted to the edge of the point cloud of the building, such as the eave; If checked, when the building has a elevation point cloud, the building outline will be extracted to the building elevation.



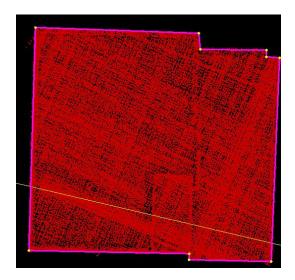




#### Eave correction

#### Rectangularization

This option is not checked by default, and you can choose whether to check it according to the actual requirement. If checked, the extracted building outline is forcibly converted into a rectangle.



Rectangularization

## 2.2.13. Buildings Editing

[Parameter description]:



Facade Correction		
○ Straightening		
O Extend Object Extr	action	
O Arc Edge Extractio	n	
O Rectangular Proce	ssing	
Cutting		
	F. Constitution of	TO THE

parameter

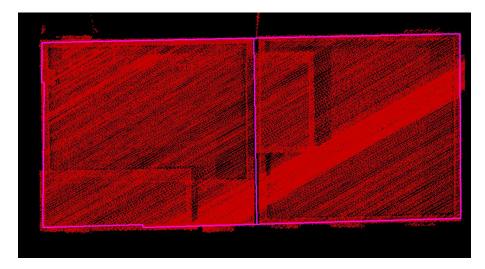
[Cutting width]: It refers to the distance between the outlines of the two adjacent buildings after the cutting operation.

[Cutting width-Parameter setting]: The default value is 0, the range can be set is [0,100);

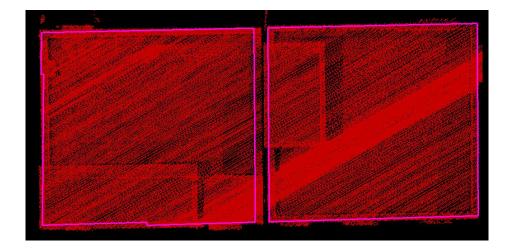
When setting the cutting width, we can set the cutting width according to the actual requirements. When the cutting width is set to 0, the outlines of the two buildings after cutting coincide at the cut line. When the cutting width is set to another value, they do not coincide, and at this time we can edit and revise the outlines of the two buildings respectively.

When the cutting width is set to 0m, after cutting the outlines of the two adjacent buildings coincide at the cut line; When the cutting width is set to 0.7m, the extension width to both sides is 0.7m take the cutting line as the center line, and the outlines of the two buildings are 0.7m apart from the position of the cutting line.





The cutting width is 0m



The cutting width is 0.7m

Note: When setting the cutting width, do not exceed the width of the building outline.

### 2.3. Roads User Guide

#### 2.3.1. Introduction

This document aims to provide technical guidance for users using CoProcess software on the process of road section extraction, and briefly show the whole process of section generation and editing.

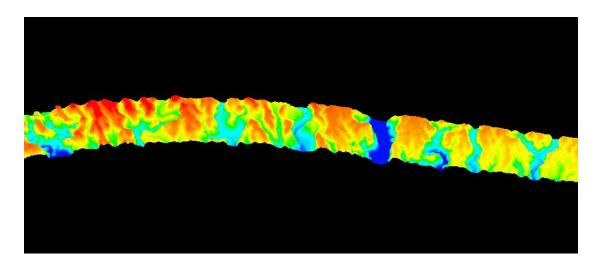
This document does not involve detailed explanation of specific functions. If



necessary, please refer to CoProcess User Manual.

#### 2.3.1.1. Sample Data

The sample data used in this demonstration is the point cloud data of a road planning project in a mountainous area, including piles, straight curve tables, point clouds, chain breaks and other documents in the design. The length of the road is about 16 kilometers.

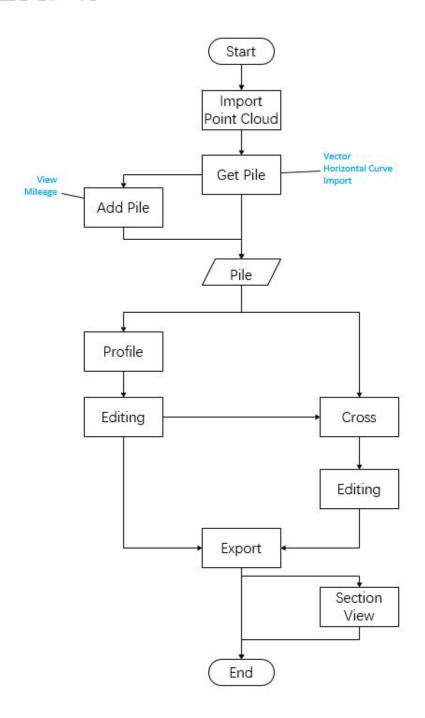


The section production process is mainly divided into the following steps:

- Import point cloud;
- Get pile;
- Generate profile section;
- Profile section editing (optional);
- Generate cross section;
- Cross section editing (optional);
- Export results;
- Generate Section view.

The production flow chart is as follows:





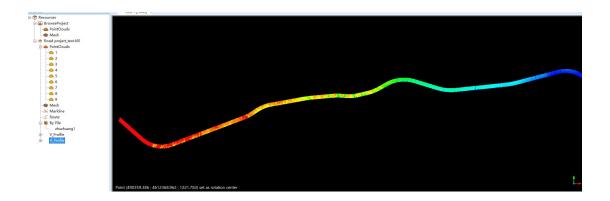
## 2.3.2. Step-by-step

#### 2.3.3. Import Point Cloud

After a new project is created, the cloud must be imported to the ground in the road engineering area. The Road module now supports the simultaneous processing of point clouds with large amounts of data using codata format point clouds. Here we load the sample codata data into the project.



As shown in the figure below, the yellow point cloud icon is the codata point cloud file.



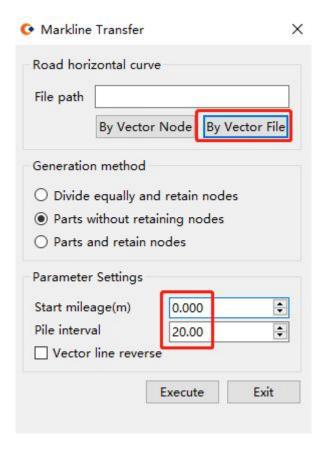
### 2.3.4. Obtain pile by pile

After the point cloud is imported, the road section needs to be extracted by pile. There are three main methods to obtain pile by pile by CoProcess software: vector line transformation, flat curve calculation and direct import. This section briefly describes each of these methods for getting a pile.

#### 2.3.4.1. Vectoring and driving piles

The function of vector line to pile is to generate pile by pile by importing vector line externally or drawing vector line manually when there is no pile or flat curve in the design.





The operation process is as follows:

- 1) Click the function of "vector line to pile" in route analysis;
- 2) Select load from node or load from file. Here we select the local dxf file, so click "Load by File" and then select the road dxf curve;
  - 3) Keep the default values for other parameters and click Execute.

After execution, the pile generated by the local vector line is mounted to the project.

Click the Horizontal Curve in Road Prase, click Load data in the Intersection method, open the corresponding JD data, and then click the Route Calculation, the content of the dialog box will jump to the route page.

Click the Route Create, the corresponding road map and node will appear in the route node under the project.



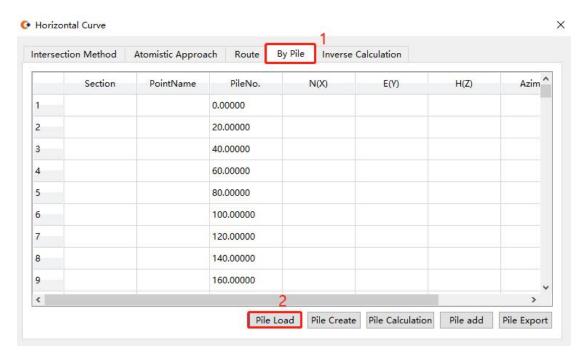
#### 2.3.4.2. Flat curve calculation pile by pile

When a road design document is available, the route calculated by the flat curve can be used to generate pile-by-pile. CoProcess supports the analysis and route calculation of intersection method and element method flat curve data. This paper introduces the method of calculating pile by pile by taking the intersection method as an example.

The operation process is as follows:

- 1) If there is a chain break in the designed road section, click the chain break function in route analysis to import the broken chain;
- 2) Click the flat curve function in route analysis to open the "intersection method" on the first page;
- 3) Click the "Load data" button below to select the local intersection method route resolution file;
- 4) Click Calculation, and the dialog box jumps to "Route" on the third page. At this time, the software has parsed the input file into a route;
- 5) Click "Pile by pile" on the fourth page of the dialog box, and click "Generate pile Number" below;





- 6) After setting the pile distance, click "pile by pile" to calculate "pile by pile";
- 7) Finally, click Draw pile by pile to mount pile by pile to the project.

## 2.3.4.3. Direct import pile by pile

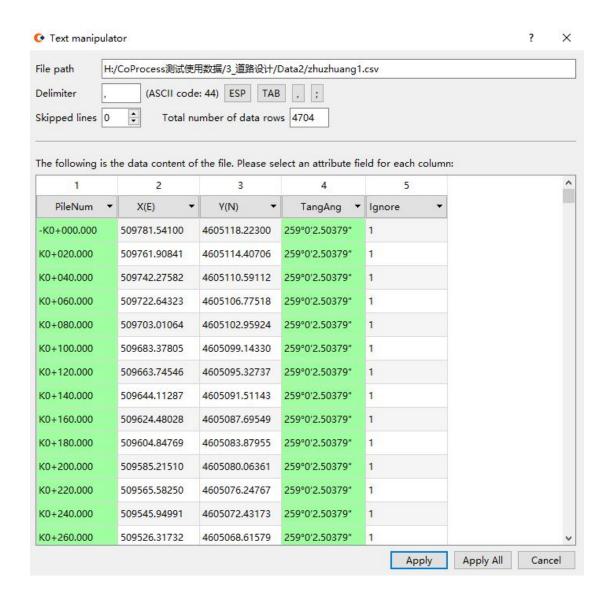
If there is already designed pile by pile, it can be directly imported into the project and used to generate sections.

Right-click the pile-by-pile node of the road project and select the "Import" function in the menu to import the local pile-by-pile file into the project.

### Note:

- (1) When importing the pile-by-pile files in the formats of txt, csv, dat, tmx or route you need to adjust the data type of each column to avoid errors when calculating the sections.
- (2) When importing the pile-by-pile files in the dxf format, the Auto 2000 and/LT2000 version is required and the layer name should be DMX or dmx.





After the pile-by-pile file is obtained by the above three methods, the pile-by-pile file will be mounted to the "pile-by-pile node" of the project, and then the pile-by-pile file will be used to extract the section.





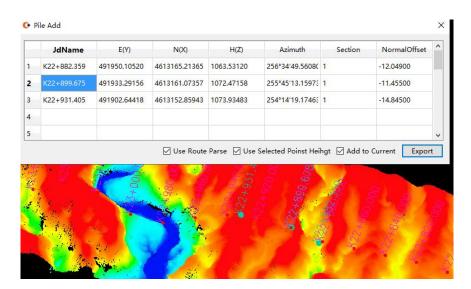
# 2.3.5. Pile Add(Not Necessary)

If some pile points in the automatically generated pile by pile do not meet the requirements of the generated section, we can modify the pile by pile mounted in the project by manual piling method.

The CoProcess software supports two manual piling methods: view piling and mileage piling. Here we take view piling as an example to demonstrate the method of piling.

The method of piling is as follows:

- 1) For the convenience of browsing, we first convert the two-dimensional pile by pile into three-dimensional pile. Use the "Generate pile by pile" in the "longitudinal section" function to generate the 3d version of the current pile by pile;
  - 2) Click "View piling" in the drop-down list of manual piling function;
- 3) Enlarge the view, find the place where the pile needs to be added, and click the left mouse button there to generate a temporary pile point;
- 4) Repeat the above operation until all pile points are added. Click Export to save the pile adding information to the current pile by pile.





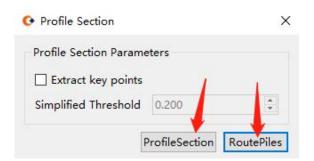
### In addition:

The mileage piling function does not need to click the position of the piling point in the view, and directly input the piling number in the dialog box to add the piling, but it needs the route analysis results to get accurate results.

## 2.3.6. Generate Section

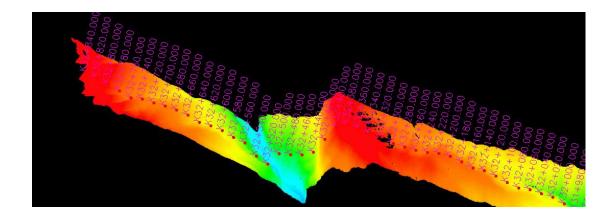
#### **Profile Secion**

- 1.Import route point cloud and 2D pile-by-pile file;
- 2.Click the road module -> Road Survey -> Profile Section;
- 3. Setting whether the features are simplified, click ProfileSection or RoutePiles.



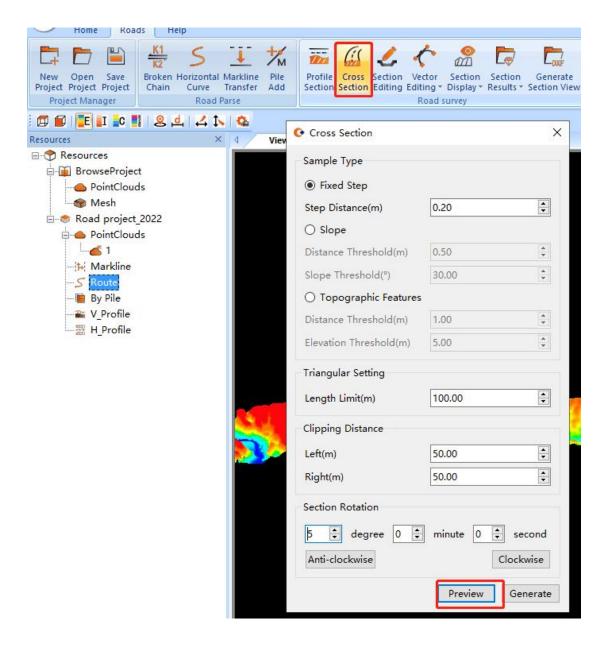
RoutePiles means generating 3D pile-by-pile.

As shown in the figure above, purple dots and yellow tracks represent the resulting longitudinal section on the point cloud.





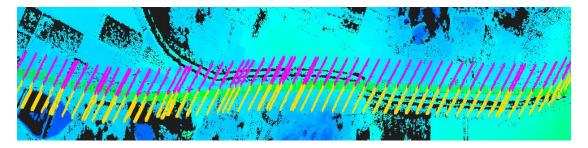
#### **Cross Secion**



Click Road survey -> Cross Section to Setting the section parameters.

After there is no problem in section preview, click Generate to extract and calculate section feature points.



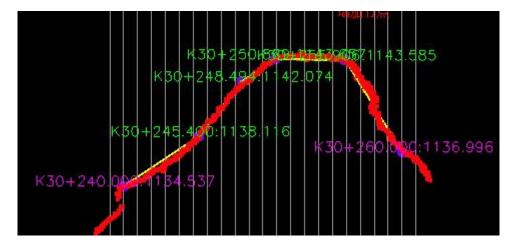


# 2.3.7. Vertical section editing

The current algorithm cannot guarantee the accuracy of the automatically extracted longitudinal section, so we need to check and edit the longitudinal section manually. The following is an introduction to the relevant precautions for vertical section editing:

- 1) Click the "longitudinal section Edit" button, the software will create a longitudinal section edit window, and display the section line and point cloud between the first two pile points in the section window;
- 2) Use the switch button or shortcut key  $\leftarrow$  or  $\rightarrow$  to switch some sections currently displayed;
- 3) When editing the section, you can use the shortcut keys A, D, M, Shift+D to activate the functions of point, delete point, shift point and batch delete point, and you can also use the shortcut key G to hide/show the grid.



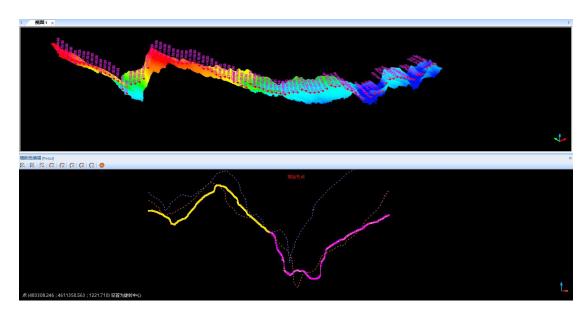


The vertical section edited pile points are automatically saved to the currently loaded pile by pile, and this temporary pile is also written to the pile by pile folder within the project after saving the project.

## 2.3.8. Edit Section

Click the Section Editing function, a cross-sectional editing window will appear.

According to the cross-sectional picture displayed in the cross-sectional editing window, find out the incorrect place and use the shortcut function bar of cross-sectional editing to modify it.





# 2.3.9. Vector Editing

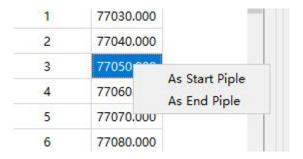
Vector line editing is the editing function of line and section. Click the drop-down box of vector line editing, many editing functions will appear. If you select the desired function, you can edit the line and section.

## 2.3.10.Exporting Survey Results

Click the section to be exported, select the category to be exported, and change the file name to export the section.

## 2.3.11.Generate Section View

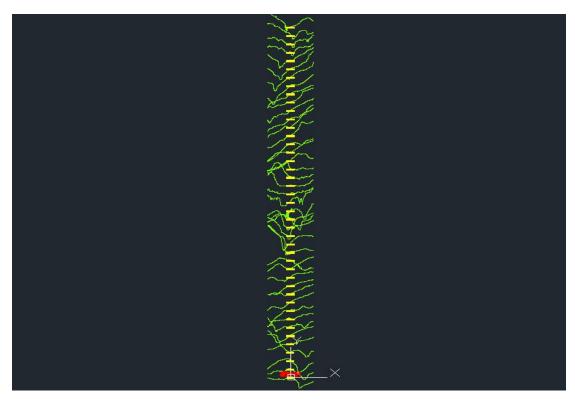
- 1. Click the function of generating sectional view;
- 2. Click Import file and select HDM file;

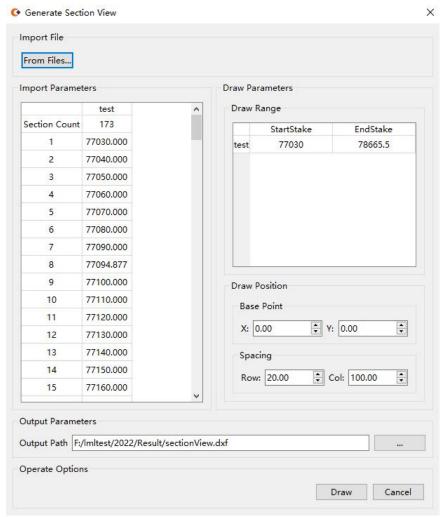


- 3. Modify the drawing base point and spacing, right click on a section can set its pile number as the starting pile number or the end pile number;
- 4. Set the output path;
- 5. Click "Cross section Design Drawing" after setting.

After drawing, you can find the drawn DXF file in the output path.









# 2.4. Volume User Guide

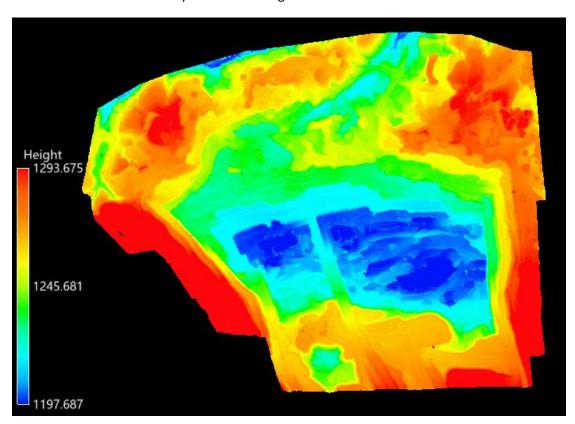
## 2.4.1. Introduction

This document is intended to provide technical guidance for the production process of single-phase volume, Multi volume and Bench line extraction for users using CoProcess. Taking point cloud data in a mining area as an example, this document briefly shows the whole process from data import, volume calculation to output results.

This document does not involve detailed explanation of specific functions. If necessary, please refer to CoProcess User Manual.

# 2.4.2. Sample data

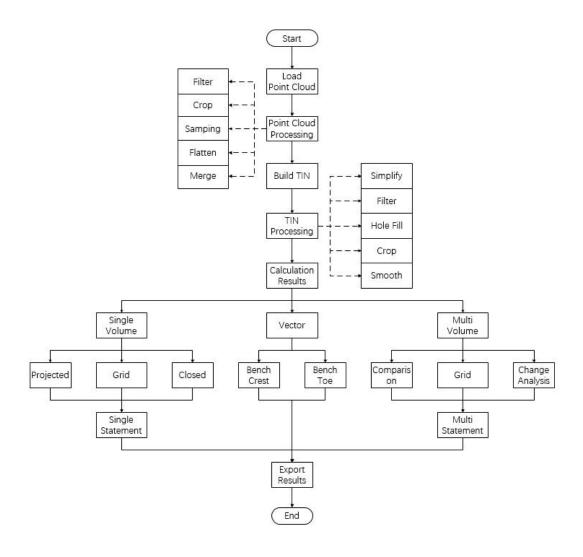
The sample data used in this demonstration is the two-stage LAS point cloud collected by UAV in a mining and mountain area.





## 2.4.3. Work Flow

The production process of volume production using CoProcess is shown as follows:



CoProcess software supports the generation of volume results such as single-phase volume report, multi contrast volume report, and special vector line.

This paper will introduce the processing methods of several classic cases, such as projection volume calculation, two-phase volume comparison report generation, two-phase change analysis and calculation, and slope top and slope bottom line extraction, to guide users to use CoProcess software for production.



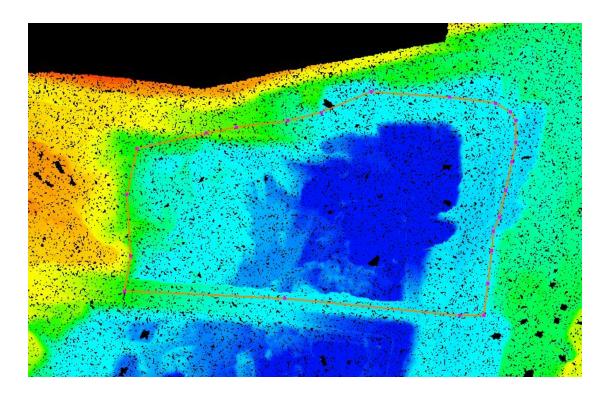
# 2.4.4. Preprocessing

Before the volume calculation, a series of processing of the input point cloud is needed to finally get the triangulation network that meets the requirements, and then the triangulation network can be used to calculate the corresponding results. This section describes how to preprocess data.

Preprocessing is mainly divided into two parts: point cloud processing and triangulation network processing.

### Point cloud data processing and networking

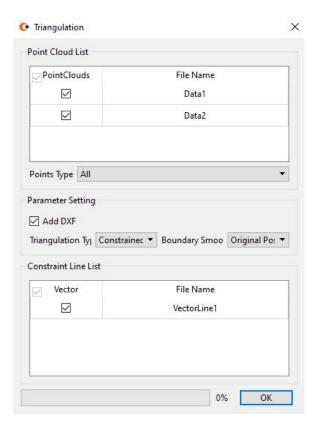
Firstly, the point cloud data to be processed is imported, and the point cloud with qualified quality is obtained through filtering and compression function. Then use the "Vector Line Processing" function to import or draw the boundary vector line of the region of interest;



Using the "TIN" function, select "Add DXF" in the parameter Settings, and select the vector line "VectorLine1" just imported in the second list.

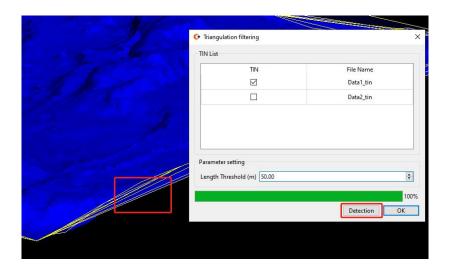


Click OK to generate a triangulated network with vector line 1 range constraints. The next step is to triangulate the triangulation network "Data1\_tin" to do the volume calculation.



## **Triangulation filtering**

Click "Filter" function in "TIN Edit" module, select the triangulation "Data1\_tin" in the pop-up dialog box, and click " Detection";



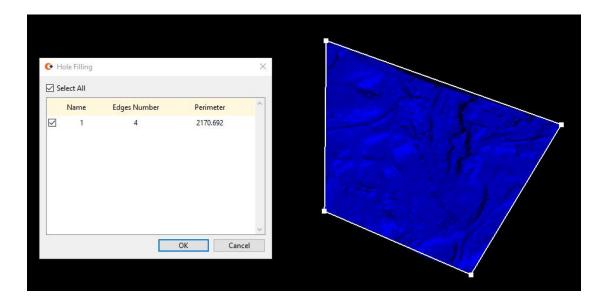


As shown in the figure, the surface that exceeds the edge length threshold of the triangulation network is detected. Click OK. You can delete these large pieces.

### **Hole Filling**

After the triangulation filtering, it is necessary to fill the holes in the triangulation network to repair the false deletion phenomenon that may exist in the previous step of filtering.

Click the "Hole Filling" function in triangulation editing, select the filtered triangulation "Data1\_tin" in the pop-up dialog box, click OK button, the software will detect the boundary in the currently selected triangulation and display it in the second dialog box.



As shown in the figure above, the software detects only one boundary in Data1\_tin, that is, there are no holes inside to fill. From here we can exit the hole filling function and click "Cancel".

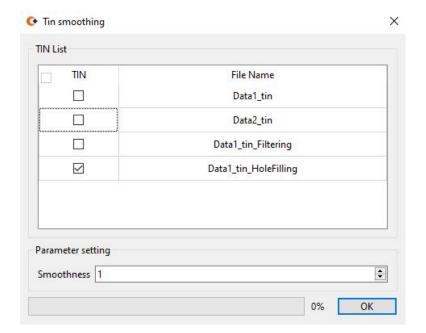
Note: If the software does not detect internal holes, there is no need to perform hole filling on the triangulation. After the processing is completed, click "Finish" to exit Wizard.

### **Smoothing**

Some sharp terrain bumps will affect the result of the volume calculation, so before



calculating the volume, one more smoothing operation is needed.



Click the "Smooth" button in TIN editing, and select the triangulation to be smoothed in the dialog box that pops up, and click OK. The default smoothness parameter is 1. The larger the value, the better the smoothing effect will be.

After smoothing, if the triangulation network does not meet the requirements, you can adjust the parameters and edit the triangulation network again.

At this point, the data preprocessing part is over, and we have obtained the triangulation network that can be used to calculate the volume. The following sections briefly introduce the operation steps of each process based on different data results.

Note: It is recommended to follow the process of filtering - hole filling - smoothing for triangulation network, otherwise it is easy to produce wrong results.

# 2.4.5. Projected Volume Calculation

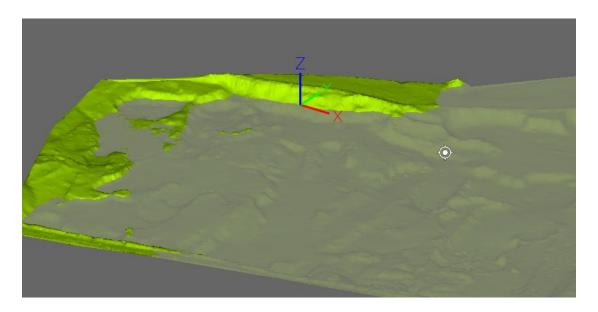
The process of projection volume calculation using CoProcess can be divided into the



following four steps: 1) Setting the reference surface; 2) Projection to the reference plane; 3) Projection volume calculation results; 4) Export reports. The basic operations of this process are described in detail below:

### Step 1: Set the reference surface.

Click the Reference Surface Settings button to open the Reference Surface Settings toolbar. Here we use the first function button "Elevation Reference Surface" and click a point on the triangulation or point cloud to create a reference surface with the elevation of that point.



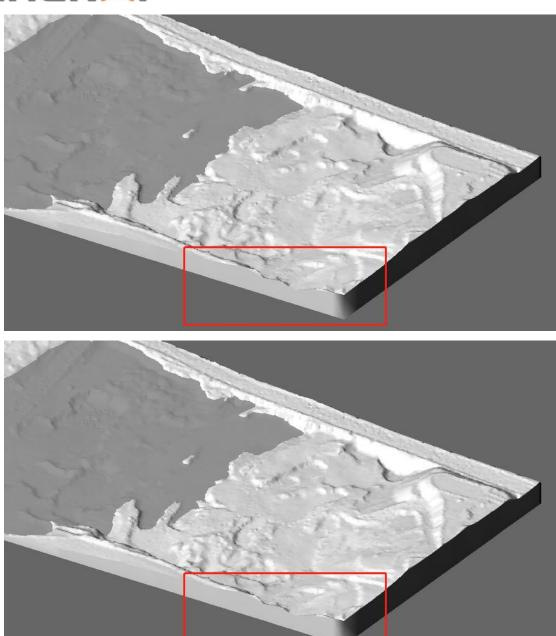
As shown in the figure, the light gray plane is the reference plane.

## Step 2: Project to the reference surface.

To calculate the projection volume, you need to create a projection triangulation based on the current reference surface.

Click the function of "Project to reference plane" and select the triangulation network to be projected in the pop-up dialog box to project it to the reference surface.





The gray boundary as shown in the figure is the projection part of the projection triangulation. Then, the projection volume of the region of interest can be obtained by using the Projection Volume Calculation function.

## Step 3: Calculate the projection volume results.

After the projection triangulation network is successfully acquired, the computation

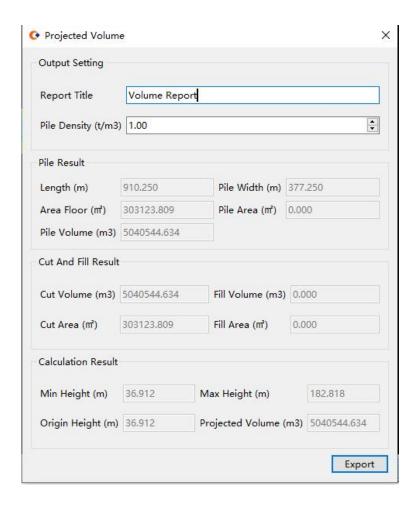


function can be used to calculate the projection volume of the triangulation network

Click the Projection Volume button in the Single Phase Volume, select the "Data1\_tin\_ Projection" triangulation in the pop-up dialog box, and click Calculate.

After the calculation is complete, a results verification window will pop up, where you can confirm the calculation results and modify the title of the report. Here, for example, we changed the report title in the output Settings to "Volume Reoprt."

You can also view the calculated pile, fill and volume calculation results.



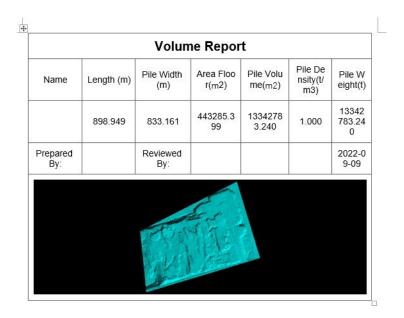
Step 4: Generate reports.

After the results are confirmed, click "Export" on the interface to automatically generate and open the report of projection volume calculation, and mount it on the achievement node in the project.



The volume calculation results can be obtained by filling the output report with the information of the tabulator.

Step 4: After the block conversion is completed, a prompt will pop up whether to load the block point cloud. Click "Yes" here for better subsequent operations.



# 2.4.6. Volume Comparison

Before using the two-phase volume comparison, the triangulation data of the two-phase region of interest should be obtained through the preprocessing function. After processing the triangulation data, the software will automatically calculate the comparison of filling and digging square, average elevation, three-dimensional area and other information of the two stages in the area by using the function of two-stage comparison calculation. Let's briefly introduce the process of operation.

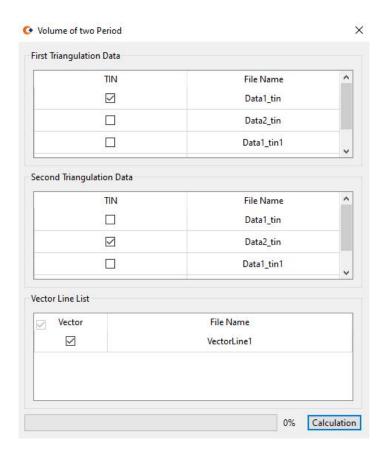
#### Step 1: Preprocess.

According to the preprocessing process in Section 4, the qualified triangulation data of the two periods before and after are obtained.

### Step 2: Data calculation.



Click the volume comparison function in the two-phase comparison, and check the triangulation network in the first two dialog boxes, and the constraint vector line in the third dialog box.

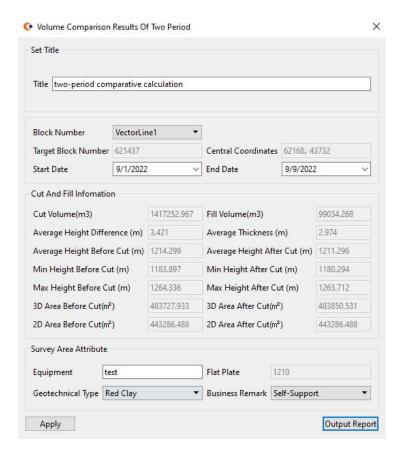


After selecting the corresponding data, click Calculate to generate a data test box just like the projection volume calculation. You can check the calculation results and edit the report information. In addition, when selecting vector lines here, multiple vector lines can be selected at the same time, and the change information in the corresponding region will be counted together.

Here, we changed the title of the report to "two-period comparative calculation", and modified the starting date and the property of the measured area of the calculation data of the area code of the vector line.



After confirming everything, click the "Apply" button.



Step 3: Output the report.

After confirming all the information, click Output Report to generate a two-period comparison report in XLS format. All the areas contained in the selected vector line will be calculated and exported together.

You can find the output report under the project node, right click to view the saving path.

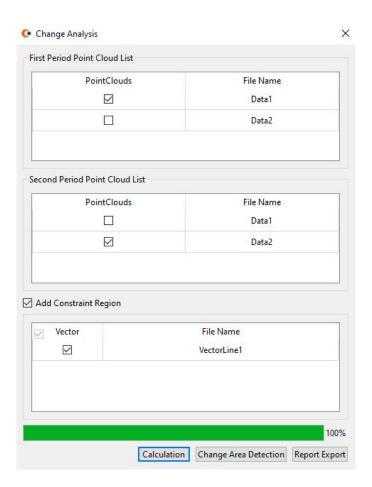
# 2.4.7. Change Analysis

Two-period change analysis was carried out to analyze the point clouds of two periods before and after the same position, and to obtain the information such as the thermal map



of earthwork change and the filling and digging line. Therefore, the method introduced in this section does not require the previous complete data preprocessing process, and only needs to process the point cloud data of the two periods.

The process of generating results of two-stage change analysis mainly includes the following five steps: 1) Selecting data; 2) Calculate the diff map; 3) Report export; 4) Change area detection; 5) Vector export. The following is a brief introduction to these steps and the corresponding operation method.

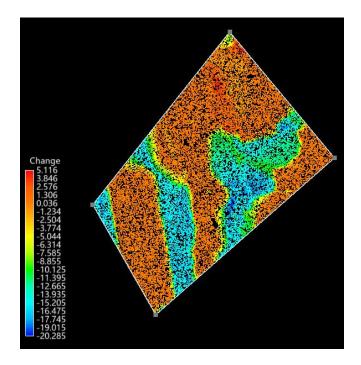


Step 2: Calculate the diff map.

After confirming the selection of the point cloud and the constraint area, click the OK button. After the progress bar is completed, the software will generate a thermal map of the earthwork changes in the working area. The default is the red-blue band change, that is, the parts with large changes are rendered as red and blue, and the parts with small changes are



rendered as green/yellow in the middle.



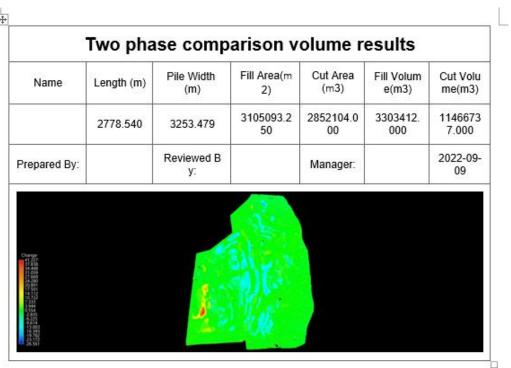
Click the Render adjustment function in the two-phase comparison to modify the color band and range of the thermal map rendering. It can do not pay attention to parts of the region with large and small changes, and focus on the analysis of regional point clouds with changes within the set range.

## Step 3: Export the report.

After adjusting the rendering mode of the thermal map area, click the "Report Export" button in the Change Analysis and calculation dialog box to generate the report of volume contrast change of two periods with one click.

The software will automatically calculate the length of the stack, the width of the stack, the area of the filling square, the volume of the filling square and other information of the comparison between the two periods, and output it in the report.





Right-click the corresponding report to find the saving path of the report.

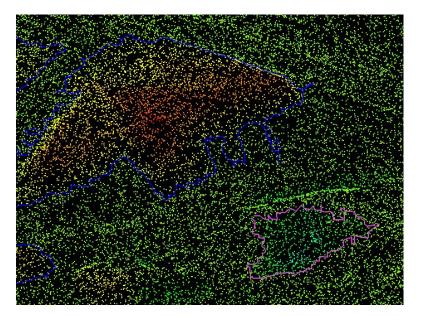
Step 4: Change area detection.

After calculating the volume change of the two periods, the current point cloud area can be detected, and the construction line filling and digging can be extracted.

Click the "Change Area Detection" button under the change analysis dialog box to extract the line filling and digging in the construction area with one click. Meanwhile, manual editing and deletion of these vector lines are supported.

The blue vector line is the add line, and the pink one is the cut line.





Step 5: Vector Export.

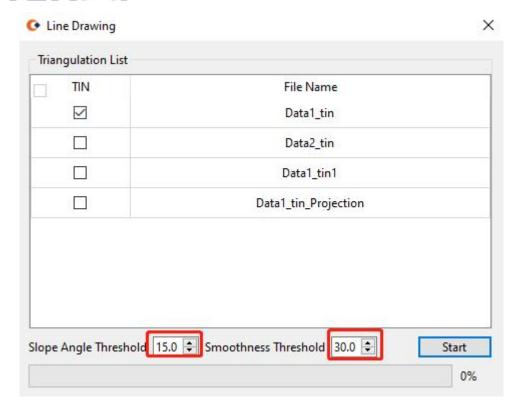
After the vector editing operation for add and cut lines is completed, select the filling and digging lines, click the "Export Vector Line" function in vector line processing, and select the export path to complete the export of vector lines.

## 2.4.8. Bench Lines

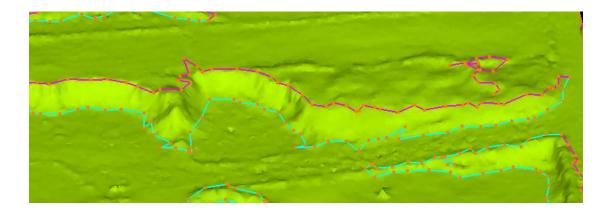
The extraction function of bench crest and bench toe depends on the triangulation network generated by preprocessing. The extraction process is mainly divided into the following steps: check the model, set parameters, perform extraction and vector export. The following is a brief description of these extraction processes:

- Through data preprocessing, the triangulation network of the region of interest is obtained.
- Click the slope top and bottom line extraction in the vector results, and select the triangulation network to be extracted in the pop-up dialog box;
- 3. Set the slope Angle threshold and smoothness threshold, and click Execute.





- 4. Close the extraction dialog box and select the vector line to be exported under the vector node in the workspace.
- 5. Click the function of vector line export in vector line processing;
- 6. Set the export path and click Start to generate the DXF results extracted from slope line.





# 3. Software UserManual

# 3.1. File&Project

File and Project include workspace opening and saving, point cloud and model import, point cloud export, terrain module conversion, point cloud storage conversion and system parameter setting.

## 3.1.1. Open Resources

### Description

Open the saved workspace, and the point cloud or model data in the workspace will be loaded automatically.

### Steps

Click File and Project -> Open Resources, select the CoPrj file, and click Open.

#### 3.1.2. Save Resources

#### Description

After saving the workspace, it is convenient to directly open the workspace next time to load point cloud, model or vector data.

### **Steps**

After importing point cloud or model data, click File and Project -> Save Resources, select the save path, enter the file name, and click Save.

## 3.1.3. Open Point Cloud

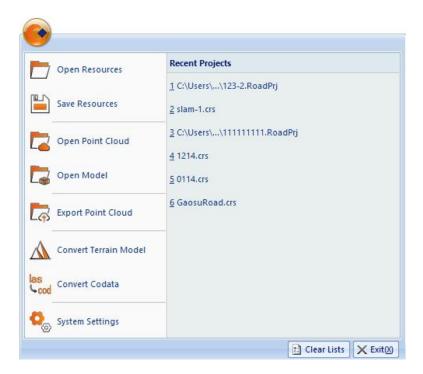
There are three ways to open a point cloud:

- Click File & Project -> Open Point Cloud to import data;
- Right-click the point cloud node in the resources to import the point clouds.
- Drag the point cloud file into the view to load and display.

The icon of the external memory point cloud is yellow, and the icon of the memory point cloud is orange.



# 3.1.3.1. Open by File&Project



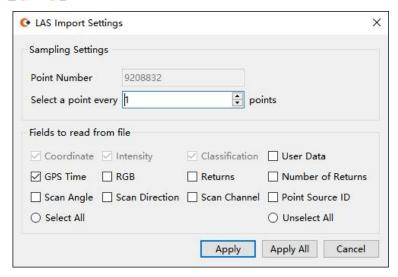
3.1.3.1.1. Open las

#### Description

Open the point cloud data in las format and display it in the view.

- Click File and Project -> Open Point Cloud, select the las format, and click Open.
- The las import setting dialog box pops up, where you can view the number of point clouds and set the thinning option. By reading the las point cloud file information, the coordinate, intensity and category field information is loaded by default, and the user can check other attribute fields that need to be imported.
- Clicking Apply only takes effect for a single las data setting. If you import multiple pieces
  of las data, you need to execute Apply All to take effect for the selected las file, and load
  and display the las point cloud into the 3D view.





#### **Notes**

Only support opening las format point clouds of version 1.4 and below.

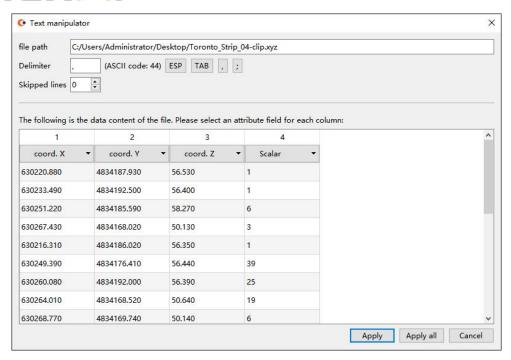
## 3.1.3.1.2. Open ASCII

### Description

Open point clouds by ASCII (\*.txt, \*.pts, \*.csv, \*.asc, \*.xyz) format.

- Click File and Project -> Open Point Cloud, select ASCII format point cloud, and click the
   Open button.
- In the interface of importing ASCII, the default import is point cloud object, and the separator can be freely selected according to the data format.





As shown in the figure above, information such as X coordinate, Y coordinate, Z coordinate, R component, G component, B component, Nx component, Ny component, Nz component and intensity are automatically assigned to each column of ASCII text data. If the data in the first column has other information, you need to ignore it; if you import control vector, you need to select the "target number" attribute in the first column.

To import ASCII text files in batches, click Apply All to import data in batches.

#### 3.1.3.1.3. Open e57

### Description

The E57 format is a file format specified by the ASTM International Standards Organization, and is an agreed transfer format for point clouds, images, and 3D imaging systems acquired by laser scanners and other equipment.

An E57 file is a 3D image file created according to the ASTM E57 format, a format used to store data captured by a 3D imaging system.

The software supports opening \*.e57 laser point cloud files.

- Click File and Project -> Open Point Cloud to pop up the Open Data interface;
- Select the point cloud in \*.e57 format, click the Open button, and the point cloud data will be loaded and displayed in the view.



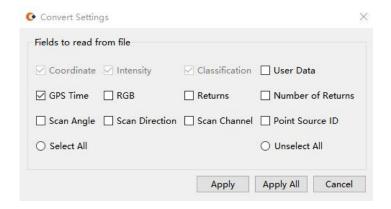
### 3.1.3.1.4. Open codata

## Description

Open the \*.codata point cloud and display it in the view. The codata point cloud is a data type customized by the CoProcess software and supports massive data loading.

### **Steps**

- If there is codata data, click File and Project -> Open Point Cloud, and select single or multiple codata files to load.
- If it does not exist, select the las file, load the conversion settings interface, select different properties, and click the Apply or Apply All buttons. Start the las conversion codata process, and load and display the codata data after the conversion is complete.



### **Notes**

The view currently supports loading a maximum of 1024 codata files, that is, a maximum of 300G external storage point clouds can be loaded and displayed at the same time.

## 3.1.3.2. Open by Resources

### Description

The resources loads point clouds in las, ASCII, e57 and codata formats.

## **Steps**

 Right-click the workspace -> Browse Project -> Point Cloud Node, left-click to select and click to open the point cloud, as shown in the figure.





Select las, ASCII, e57 or codata format to load the point cloud to the view, and at the same time import the data and mount it to the point cloud node.

## 3.1.3.3. Open by Drag

### Description

Drag the las or codata format point cloud into the view to load. When dragging the las point cloud, it will be directly imported into las or converted to codata according to the import point cloud setting in the system settings.

### Steps

Select one or more las or codata point clouds, hold down the left mouse button and drag the data into the view. Pop up the left button to load and display the point cloud.

# 3.1.4. Open Model

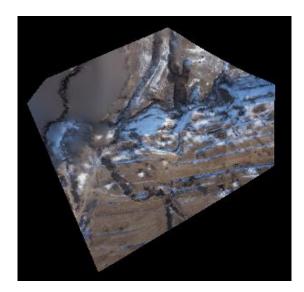
# Description

Load and display the model data such as osgb.

### Steps

 Click File&Project -> Open Model, select the osgb 3D model file, and click Open to display the progress of importing the model. The import is successful and the model is displayed in the view.





Click to open the model, select the xml file, and click Open to load the model data.

# 3.1.5. Export Point Cloud

## 3.1.5.1. **Export las**

### Description

This function support exporting and saving the las or codata point clouds.

## Steps

- Select the point cloud object, click File & Project->Export Point Cloud; or right-click the point cloud object in there sources and select the Export Point Cloud.
- Select the save path, select the save type \*.las, enter the file name, and click the save button.



3.1.5.2. Export ASCII

## Description

Export ASCII( \*.txt, \*.pts, \*.csv, \*.asc, \*.xyz) point clouds .

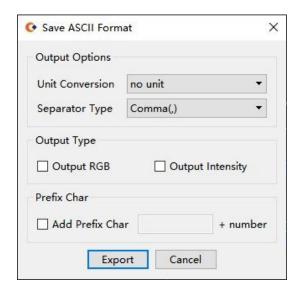
## **Steps**

Select the point cloud, click File and Project -> Export Point Cloud.



Select ASCII (\*.txt, \*.pts, \*.csv, \*.asc, \*.xyz) as the save type, click Save, and the interface for saving ASCII point cloud will pop up.

You can choose three types of separators: comma, space and Tab key. Choose whether to output intensity values, RGB values, and add prefixes according to your needs. After the selection is complete, click Export to save the point cloud in ASCII format to the specified path.



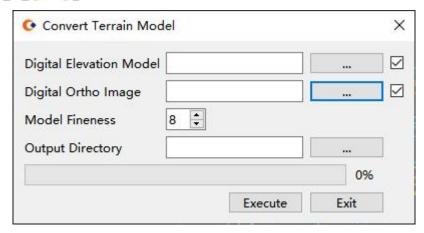
## 3.1.6. Convert Terrain Model

## Description

Digital terrain model with projection information is generated according to the image information of the DOM and DEM.

- Click File and Project -> Terrain Model Conversion to load the conversion interface.
- Select the DEM and DOM images, set the model fineness, select the model output path,
   and click Execute to generate a 3D model.





 Click to open the model, select the generated osga data, and click Open to load and display the model.



### **Settings**

**Model fineness:** The default is 8, and the value range is 3-15. The model fineness is related to the resolution of the DOM and DEM. The higher the resolution, the more fineness levels can be selected. The higher the fineness of the model, the clearer the generated model, the larger the amount of data, and the longer the time-consuming.

## 3.1.7. Convert Codata

### Description

The point cloud external memory conversion function is to convert the las point cloud into a codata format point cloud.

### **Steps**

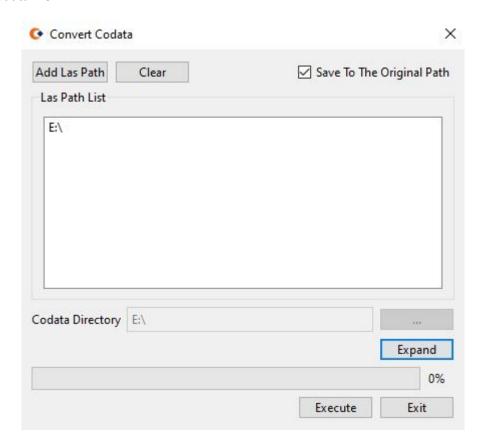
 Click the covent codata, as shown in the figure, pops up the point cloud conversion interface;



- Click "Add Las Path" and select the path of one or more LAS files to be converted.
- Click on Expand Settings to select the properties field to retain when converting. By default, coordinates, intensity, category, and GPS time are reserved.
- Click the Execute button to start the conversion, and the progress bar will display the conversion progress in real time.
- After the conversion is completed, the corresponding prompt message will pop up.

## **Save To the Original Path:**

When selected, the codata generated by the conversion will be stored in the path of the original LAS. If you do not select this parameter, you need to manually set the saving path for the codata file.



#### **Notes**

External memory conversion function, only supports las format point cloud conversion.

# 3.1.8. System Settings

3.1.8.1. Save Settings

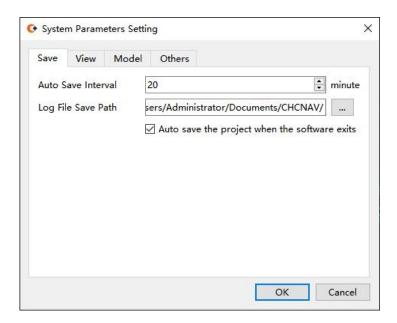
### Description



Set the software automatic saving interval and log saving path.

#### **Steps**

- Set the automatic save time: The default automatic save interval is 20 minutes. If you
  need to modify it, enter the save time and click the OK button to take effect.
- Set log path: select the path and click the OK button to take effect.



3.1.8.2. View Settings

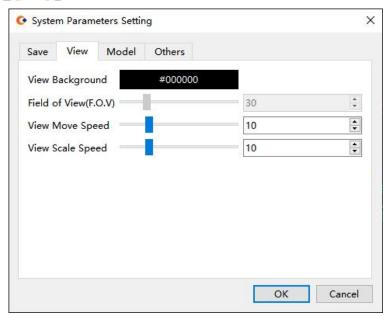
## Description

This function sets the background color of the software view, the moving speed and zoom speed of the mouse in the view, as well as the switch of model lighting.

## **Steps**

 Set the view background: Click the color to be set, the background color is black by default.





- FOV(Field of View): Drag the moving bar, or directly input the value of 15-85, you can adjust the size of the field of view in the perspective projection.
- Moving speed: drag the moving bar, or directly enter a value from 1 to 40, and click OK to save the setting. The faster you move, the more you move the point cloud across the view.
- Scale speed: drag the moving bar, or directly enter a value from 1 to 40, and click OK to save the settings. The higher the zoom speed, the faster the point cloud is zoomed on the view.

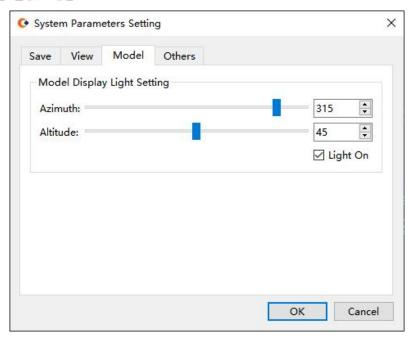
## 3.1.8.3. Model Settings

### Description

Display the azimuth and altitude angle.

- Model display lighting settings: Drag the moving bars of azimuth and elevation angles or directly adjust the angle to change the angle at which the model displays lighting.
- Check "Open Lighting Effect Options" to open the lighting settings.

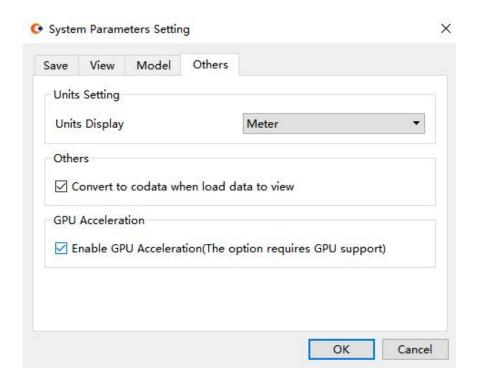




3.1.8.4. Others

# 3.1.8.4.1. Units Setting

Units include Meter, International Feet and US Survey Feet.



3.1.8.4.2. Import Point Cloud

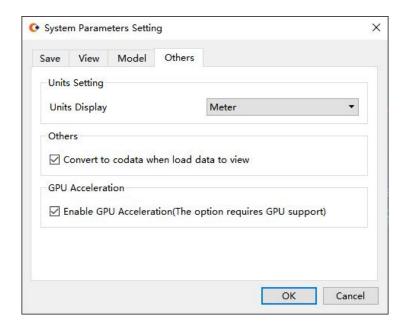
# Description



When dragging the las point cloud to the view or opening the las point cloud, it controls whether to convert the las to codata format. When this option is checked, the external storage point cloud conversion will be performed; when it is not checked, the las format will be loaded directly.

#### Steps

 Check File and Project -> System Settings -> Point Cloud Import Settings -> Drag and drop to import whether to convert codata, and point cloud is determined.



3.1.8.4.3. GPU acceleration

#### Description

The GPU acceleration function is enabled by default. This function requires that the computer hardware has a discrete graphics card to provide GPU support.

#### 3.1.8.4.4.DEM Pixel Center Settings

### Description

Sets whether the pixel drawing is based on the center of the pixel in the upper-left corner or the boundary point in the upper-left corner when generating DEM. The results generated under these two benchmarks will have a horizontal offset of half a pixel in length.

After setting up, you need to restart the software before it will take effect.

### 3.1.9. Theme

Click the theme function in the upper right corner of the home interface of the software to



select different styles of themes, and the software will display the selected theme style as the default.

The software provides four styles of themes, blue, black, silver and aqua.

## **3.2.** Home

### 3.2.1. View Direction

View directions include front view and isometric side view. Front view includes front, rear, left, right, top, and bottom views. Isometric view supports viewing from four directions: southwest, northwest, northeast, and southeast.

## 3.2.2. View Scale

### 3.2.2.1. Full Extent

### Description

Scales the data in the view to the original display scale.

#### **Steps**

- For the data after executing the zoom-in and zoom-out commands, click Home->View
   Scale->Full Extent;
- Point clouds, DEMs, DOMs, models, etc. are scaled according to the view's bounding box.

#### 3.2.2.2. **Zoom in**

To zoom in the data in the view, click Home->View Scale->Zoom in or slide the middle mouse button up to display the zoomed-in data.

#### 3.2.2.3. **Zoom out**

To zoom out the data in the view, click Home->View Scale->Zoom out or slide the middle mouse button down to display the zoomed-out data.

### 3.2.3. View Control

#### 3.2.3.1. Pan

To pan the object in the view, press and hold the middle mouse button to drag.

Execute the lock pan function, the data object will not be panned.

### 3.2.3.2. Rotation

To rotate the object in the view, press and hold the left mouse button to rotate the object in the view.

Execute the lock rotation function, the data object will not be rotated.

### 3.2.3.3. **Zoom**

To zoom the objects in the view, slide the middle mouse button up to zoom in, and slide the middle mouse button down to zoom out.



Execute the lock zoom function, the data object will not be zoomable.

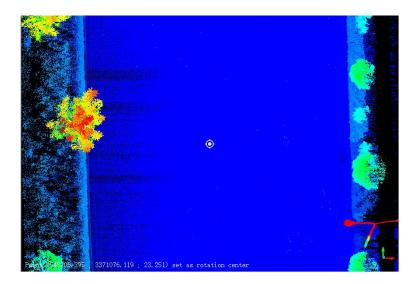
### 3.2.3.4. Set the center of rotation

## Description

Double-click with the left mouse button to set the rotation center of the object.

#### Steps

Double-click the point on the object in the view with the left mouse button, you can view the XYZ coordinate information of the point in the lower left corner, and set the point as the new rotation center. When the object is rotated, the rotation will take this point as the center.



3.2.3.5. Linkage

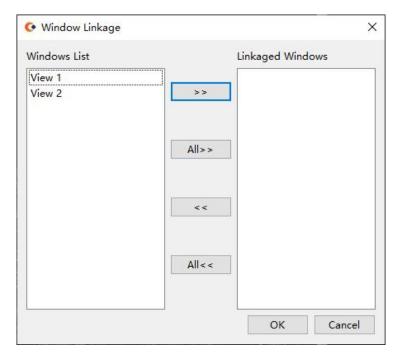
### Description

Display multiple views in linkage to realize synchronous zooming and panning of multi-view data objects.

## **Steps**

• After creating multiple views, click Home->View Control->Linkage to load the window linkage interface.





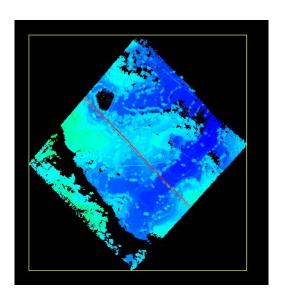
Load the view that needs to be linked in the window list on the left to the linkage window on the right, and click OK. The linkage of multiple views can be realized, and the synchronous zooming and panning of viewing objects can be realized.

#### **Notes**

The view linkage is valid for multiple views. When there is a view, the command is grayed out and cannot be used.

## 3.2.3.6. Orthographic

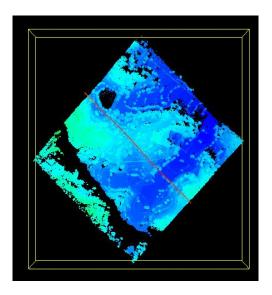
The projection method of objects in the view is orthographic projection by default. Orthographic projection is also called parallel projection. No matter how far the object is from the camera, the size of the object remains unchanged after projection.





## 3.2.3.7. Perspective

Perspective projection uses perspective projection to view objects. The closer to the viewpoint, the larger the object; the farther away from the viewpoint, the smaller the object.



### 3.2.4. Scene Roam

## 3.2.4.1. Viewpoint Roaming

### Description

Navigate through the objects in the view by setting the viewpoint. Roaming according to the current perspective position, manually adding/deleting viewpoints, the camera will roam according to the order of addition; the current roaming information can be saved and imported directly next time.

- Import point cloud, click Scene Roaming to activate Roaming Mode;
- Switch to viewpoint roaming and click "Add viewpoint" to add a viewpoint at the center of the current screen:
- Switch the viewing angle and click multiple times to add viewpoint;
- Click Start to roam the point cloud view in sequence according to the set viewpoints;
- In addition, viewpoint roaming also supports local txt format viewpoint file import and roaming speed adjustment.



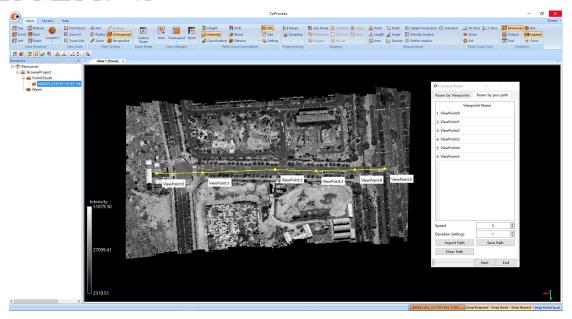


3.2.4.2. Trajectory Roaming

Navigate through view objects by setting the mode of the browse track. Roaming according to the position of the track point: click the left mouse button on the point cloud to add a viewpoint, or import a track file to add a viewpoint;

- Import point cloud, click Camera Roam, activate trajectory roaming;
- Switch to track roaming, left-click on the point cloud to create roaming track point;
- Switch the viewing angle and click multiple times on the point cloud;
- After drawing the roaming track, click Start to roam the view according to the drawn track.
- Likewise, the track view supports the import and export of txt track paths, as well as the
  adjustment of roaming speed. In addition, the elevation of the track can be adjusted
  during the walk.





# 3.2.5. View Management

3.2.5.1. New

### Description

Create and open a new view.

### **Steps**

- Click home -> View Management -> New View to create a new view;
- Drag and drop point cloud or model data from the workspace into a new view to load the data;

### Notes

A maximum of four views can exist simultaneously in a new view.

## 3.2.5.2. **View Layout**

# Description

The window layout is used when there are multiple views. It can be used in tabbed layout, tiled layout, horizontal layout, and vertical layout.

## **Steps**

- Click Home -> View Management -> New View to create at least two view Windows.
- 2Click home -> View Management -> View Layout and select TAB Layout, Tile layout, horizontal layout and vertical layout respectively to see the different layouts.

## Notes

The window layout function is effective for multiple views. If multiple views exist, the title of



the currently active view is marked with Focus.

## 3.2.5.3. Resetting the Layout

## **Steps**

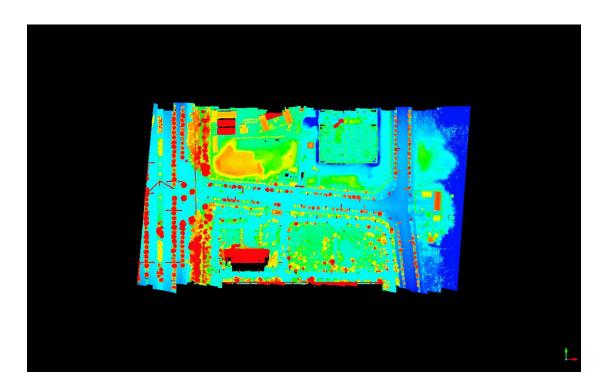
- Click to reset the layout. The system displays Do YOU want to restart the software?
   Select OK to restart the software.
- The software interface workspace and output Windows and views are restored to their initialized state.

## 3.2.6. Colorization

It supports multiple point cloud colorization: height, intensity, classification, single, blend, returns, return number, RGB and EDL etc. User can also configure the settings range and point size.

## 3.2.6.1. Color by Height

According to the elevation value of the point cloud, it is mapped to the specified color interval, which is convenient to observe height change.



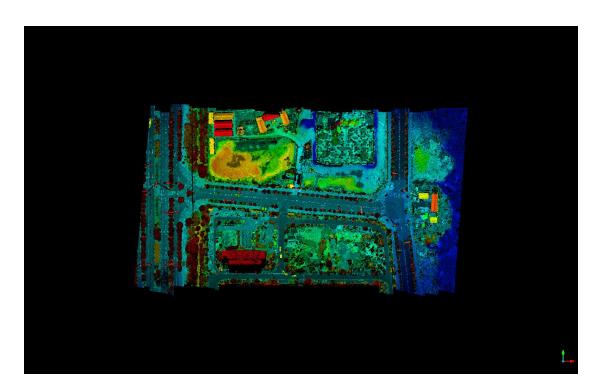
3.2.6.2. Color by Intensity

Map the intensity values of point cloud to uniformly varying color intervals.





3.2.6.3. **Color by Blend**Point clouds will be colored by intensity & height together under this mode.



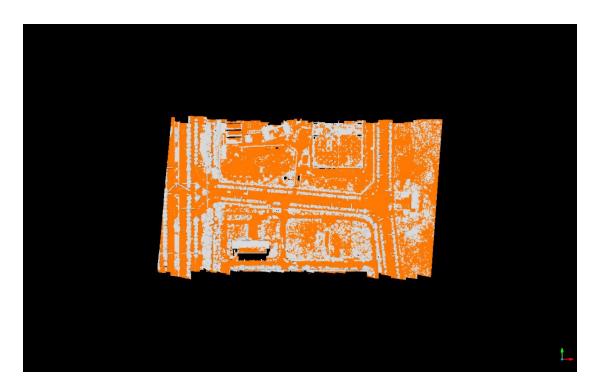
3.2.6.4. **Color by RGB**Point clouds will be colored by real RGB color under this mode.





3.2.6.5. Color by Classification

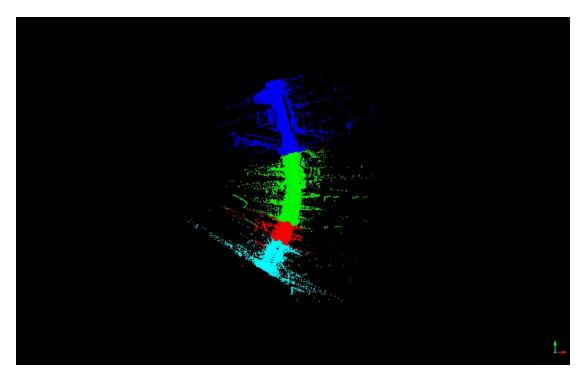
Point clouds will be colored by classification color under this mode.



3.2.6.6. Color by Single

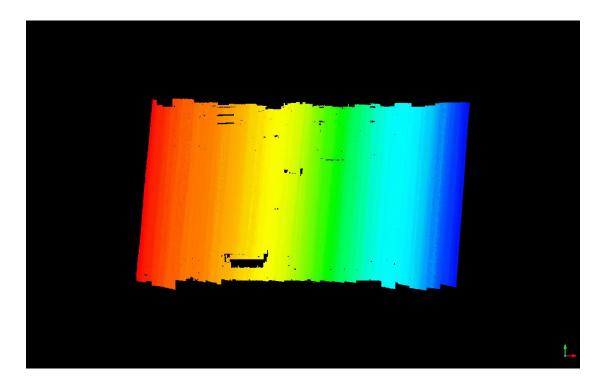
Point clouds will be colored by different colors based on different files under this mode.





3.2.6.7. **Color by Time** 

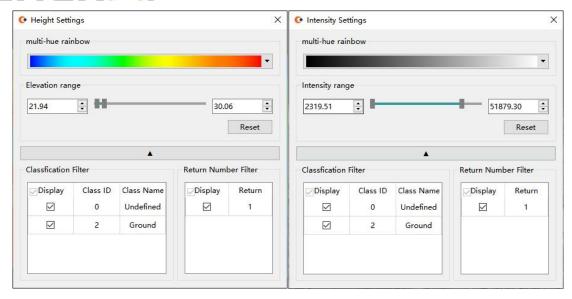
According to the different GPS time values of the point cloud, the time attributes are mapped to uniformly changing color values.



3.2.6.8. Color by Merge

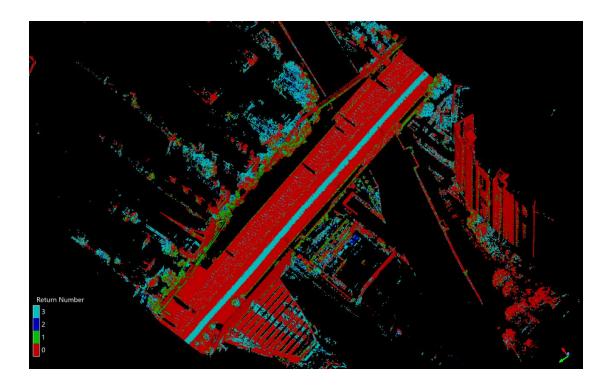
Point clouds will be colored by Height & Classification & Return number, intensity & Classification & Return number together under this mode.





3.2.6.9. Color by Return Number

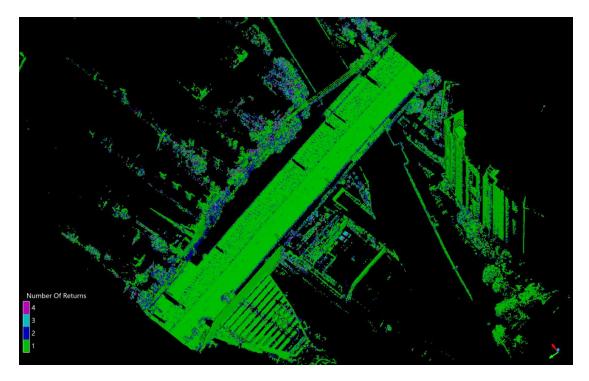
Point clouds will be colored by return number color under this mode.



3.2.6.10. Color by Number of Returns

Point clouds will be colored by number of returns color under this mode.





3.2.6.11. Size Settings

User can configure point cloud size based on this settings.



3.2.6.12. Color Parameter Settings

User can configure parameters on "Settings". Click "Reset" to restore default settings.



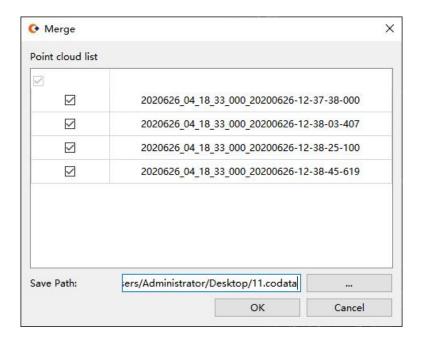


# 3.2.7. Tools

# 3.2.7.1. Merge

### **Steps**

- Import two or more pieces of point cloud data and activate the point cloud merging function;
- Select the point cloud files to be merged in the point cloud list, select the output path,
   and click OK;
- After the merging process is over, it will prompt whether to add the merged point cloud.
   Click OK to load and display the merged point cloud data;



3.2.7.2. **Sampling** 

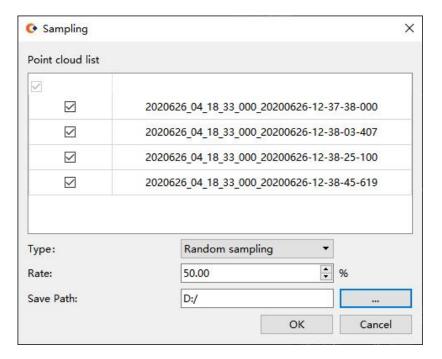
### Description

According to different sampling types, the point cloud data is thinned out.

### **Steps**

Import point cloud, click Home->Point Cloud Processing->Point Cloud Sampling, as shown below:





- Select the point cloud file in the point cloud list, select the random sampling method, enter the sampling rate, and randomly thin the point cloud according to the sampling rate; select the minimum point spacing sampling method, and enter the point spacing to uniformly thin the point cloud according to the sampling interval.
- Select the output path and click OK to thin the point cloud, and output and save the sampling results at the same time.
- After the sampling is completed, it will prompt whether to add the thinned point cloud.
   Click OK to load the data.

#### **Settings**

**Sampling Rate (%):** The thinning rate of random sampling, the default is 50%, and the value range is 0~100%. The reserved points are the product of the total points and the sampling rate. The smaller the sampling rate is, the more points are reserved.

**Point Spacing (m):** the distance between two points when sampling, the default is 0.05m, the smaller the sampling interval, the slower the thinning speed, and the more points are retained; the larger the sampling interval, the faster the thinning speed, and the points reserved few.

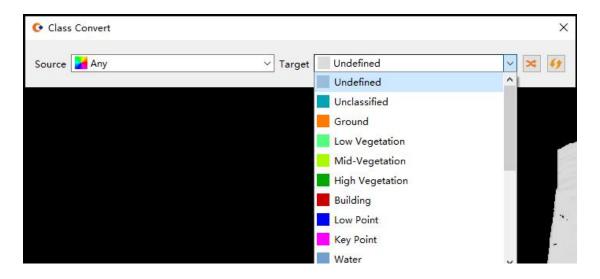
### 3.2.7.3. Class Convert

### Description

Convert point clouds of one class into another class in batches.



- Select the point clouds in las format that you want to convert and click the class conversion function.
- Select the classes of the source and target point clouds.
- Click icorn to do the class conversion.



Click icorn to switch the source and target point clouds.

# 3.2.8. Clipping

## 3.2.8.1. Clip Mode

Controls the on and off of point cloud cropping mode.

## 3.2.8.2. Rectangle Selection

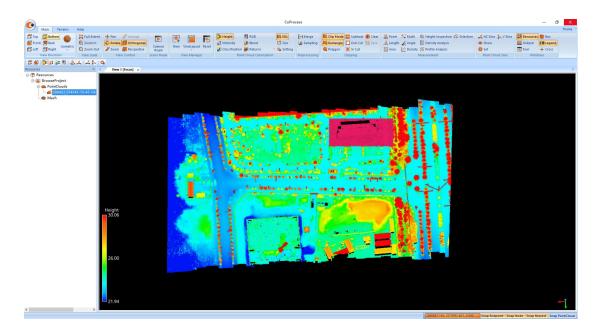
#### Description

Select the point cloud by a rectangle selection.

- Click the clip mode button to enter the crop mode; click the function button to activate the rectangle selection command;
- In the view window, click the left button to select the position of the upper left corner of the rectangle, hold down and drag the mouse until the position of the lower right corner of the rectangle is obtained, and after releasing the mouse, the point cloud in the rectangle area will be highlighted.



After the first selection, you can continue to make a rectangle selection. Selecting an area will add or subtract the selected area according to the state of the subtraction mode.



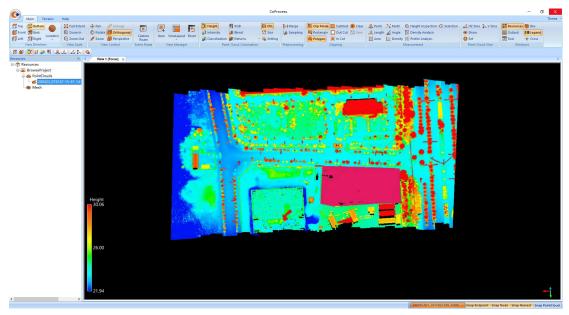
3.2.8.3. Polygon Selection

## **Description**

Select point cloud by polygonal means.

- Click the crop mode button to enter the crop mode; click the function button to activate the polygon selection command;
- Left-click to select the polygon vertex positions in turn, left-click to double-click the last vertex to end the polygon selection; the selection area is highlighted (red) as shown below;
- When the vertex position is wrongly selected, the Ctrl+Z key can cancel the last selected point, which supports multiple cancellations in succession;
- After the first selection, multiple selections can be made on the basis of the first time.
  Selecting an area will add or subtract the selected area according to the state of the subtraction mode.





3.2.8.4. Subtract

This function is used in conjunction with rectangle selection and polygon selection to control whether the currently selected area will be added or subtracted.

### **Steps**

- Subtract mode is automatically activated when a selection command (rectangle selection, polygon selection) is activated;
- By default, the button is in the unselected state, and when it is selected, it is in the selected state, and the selected area will be added;
- When this button is selected, it is in the state of subtraction, and the final result of the selection is to subtract the overlapped part of the current selected area from the selected area.

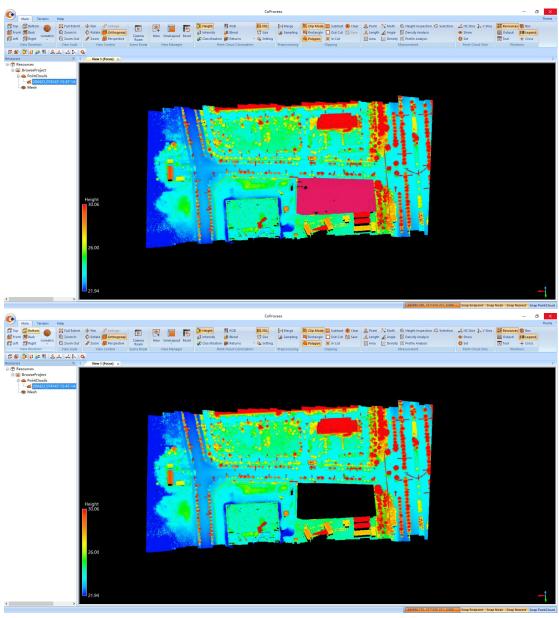
### 3.2.8.5. Out Cut

# Description

Crop the point cloud outside the selection range.

- Activate the selection function to obtain the selection area; perform the outer cropping function, as shown in the figure is the effect of cropping.
- After one cut, you can continue to select and cut multiple times on the cut result.





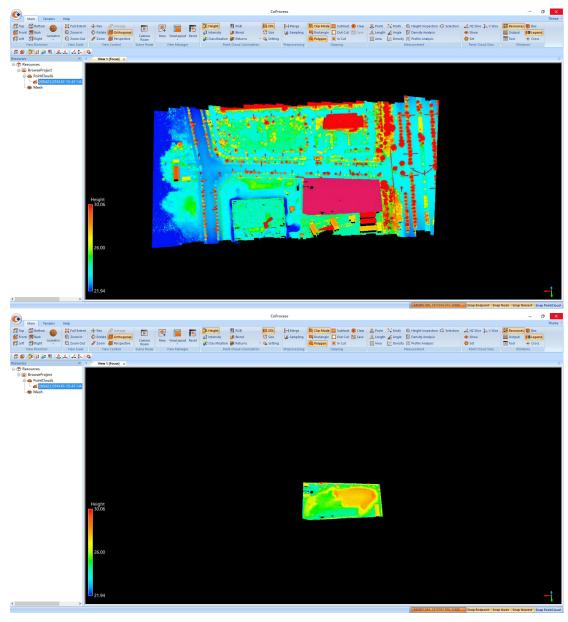
3.2.8.6. In Cut

Crop the point cloud inside the selection range.

# Steps

Activate the selection function to obtain the selection area; Perform the in cut function,
 as shown in the figure is the effect of cropping.





3.2.8.7. Clear Cut

Clears all selection operations and cropping results.

# Steps

- Perform selection or cropping operations, highlight selected areas and create cropping results;
- Execute the Clear command to immediately delete all selections and crop results.

## 3.2.8.8. Save Result

# Description

Save and output the result of point cloud cropping.



#### **Steps**

- Execute the inner trimming or outer trimming command to get the trimming result.
- Activate the save and crop function, pop up the save interface, select the original point cloud file to be cropped and saved, and select the file save directory. The new clip file is named by adding -clip after the original point cloud file name.

### **Parameter Settings:**

**Save Format:** Support to write the clipped point cloud to las or codata format. If you generate codata, you can import it directly into the view.

**Merge Output:** If the number of point clouds participating in clipping exceeds 2, check this option to merge the clipping result into a point cloud file.

# 3.2.9. Measurement

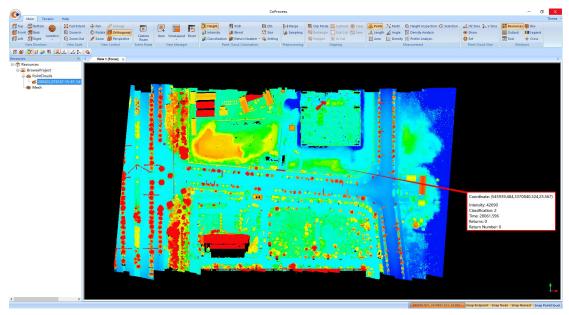
### 3.2.9.1. Point

### Description

Left-click to select a point on the point cloud to query the coordinates, intensity, time, category and other information of the point.

- Click Main->Point Cloud Measurement->Single Point, enter the coordinate measurement command, and the operation method is prompted at the top of the view;
- Select and click the query point with the left mouse button, the XYZ coordinates, RGB color, time, intensity and category of the point will be displayed on the point cloud.



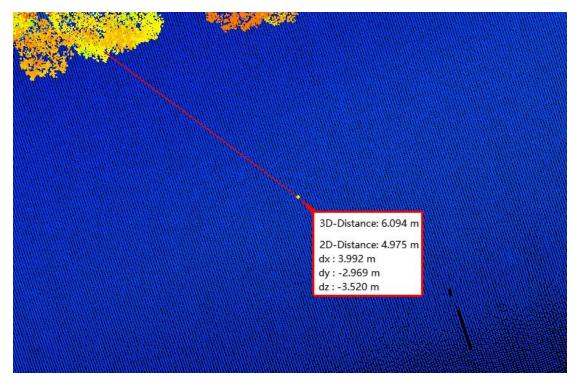


3.2.9.2. Length

The length measurement uses mouse click to query the distance between two points, and also provides the coordinate difference between the last point and the previous point.

- Click Main->Point Cloud Measurement->Length Measurement to enter the length measurement mode;
- The left mouse button selects multiple points on the point cloud in turn, the length of the polyline is displayed, and ctrl+Z can undo the previous point.



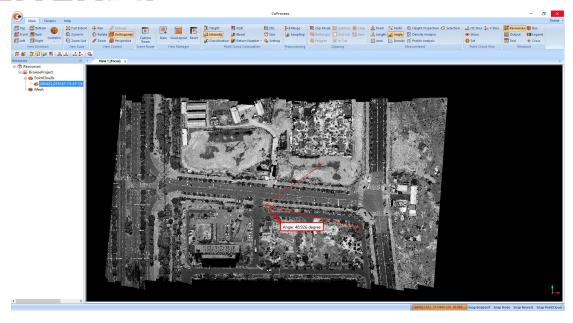


3.2.9.3. Angle

The angle measurement tool selects three measurement points in turn with the left button, and the degree of each angle of the triangle can be obtained.

- Click Home->Point Cloud Measurement->Angle Measurement to enter the angle measurement mode;
- The left-click selects three points in turn, and the angle measurement information is displayed in the prompt box.

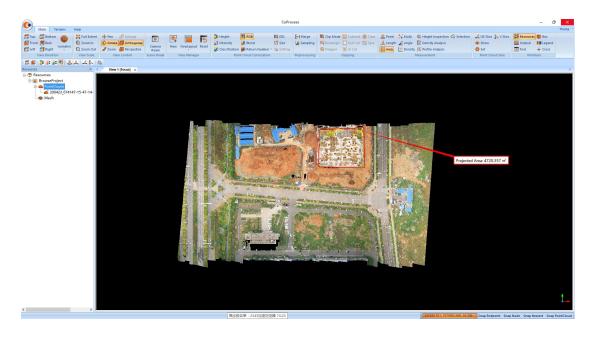




3.2.9.4. Area

The area measurement function calculates the projected area within the drawn area by drawing a polygon.

- Click Main->Point Cloud Measurement->Area Measurement;
- Select the polygon area with the left mouse button, and display the projected area in the area in real time. ctrl+Z can undo the previous point.





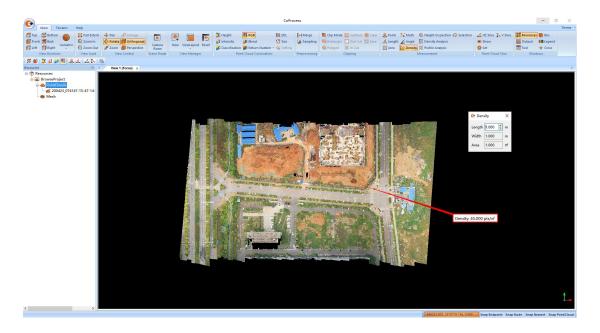
## 3.2.9.5. **Density**

### Description

The point density is obtained by calculating the average number of points in a unit square meter, which can be used as an important indicator to measure the quality of point cloud data.

#### **Steps**

- Click Main->Point Cloud Measurement->Density Measurement to enter the point density measurement command.
- By default, the point density in a unit square meter can be calculated and displayed, or you can enter the length of the square by yourself, and left-click a point to calculate the point density within the selected range.
- Click the close button in the upper right corner of the point density to end and exit the point density measurement command.



3.2.9.6. Multi-point

## Description

Select and export multiple 3D point coordinates, intensity, classification, RGB and time and other information, and support to draw two lines to obtain the coordinates of the intersection.

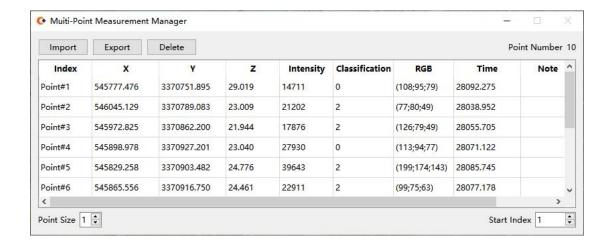
### **Steps**

Click point cloud measurement -> multi-point selection to load the multi-point



measurement panel shown in the figure below;

Left-click to select and click a point on the point cloud, a 3D sphere and a 2D label will be drawn at each selected point, and the index, XYZ, intensity, category, time and RGB information of the selected point will be displayed in the measurement panel.



- After selecting the point, click the Export, enter the name of the save file, and click Save to export the data table in csv format.
- Hold down the shift key, and the mouse draws two intersecting lines in turn to fit the position of the intersection.

#### **Settings**

Point size: control the display size of the yellow 3D sphere drawn when selecting a point;

**Number of points**: Display the number of points in the measurement panel;

Start Index: the default start sequence number value when selecting a point;

**Import:** Import the multi-point measurement values in csv format, and display the imported point label index in the view;

**Delete:** Select a row in the multi-point measurement panel to delete the information of the point;

### 3.2.9.7. Height Inspection

### Description

Calculates the elevation error for imported control points and point clouds.

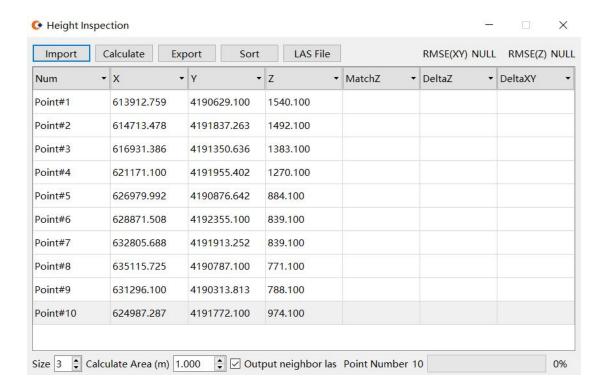
#### **Steps**

 Enable the Height Inspection, click the Import button to load and display the control points, right-click to delete the selected points, and position to center the points in the

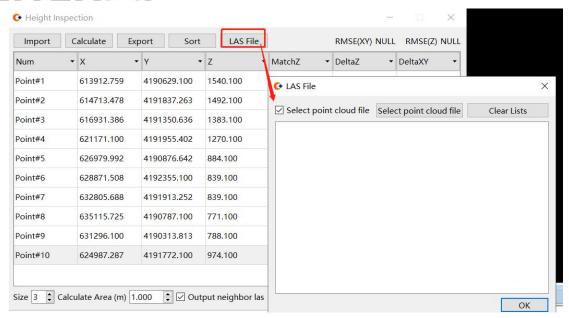


view and highlight the name label.

- Click Calculate to display the matched points on the point cloud along with the elevation error.
- Click Sort to sort according to the field.
- Export report, you can export and save the elevation report in .csv format.
- This tool also supports batch height inspection of point cloud files in the las format. Click "Point cloud file", check the "Select point cloud file" option, and click OK. Then repeat Step2 and Step3, you can obtain the height inspection result of the point cloud file in the las format and output the neighboring point cloud file in the las format in the file directory where the control points are.







### **Elevation check method:**

Several points with lower elevation values and ground points are found as matching points in a certain neighborhood range of each control point, and the software obtains the nearest two-dimensional distance and three-dimensional distance of these points.

The software first uses the point closest to 2D for matching, and if the elevation difference exceeds the built-in threshold, the point closest to 3D distance is used as the matching point to calculate the elevation difference.

If no ground points are found, select the points with the lowest elevation values in the range.

### 3.2.9.8. Density Analysis

#### Description

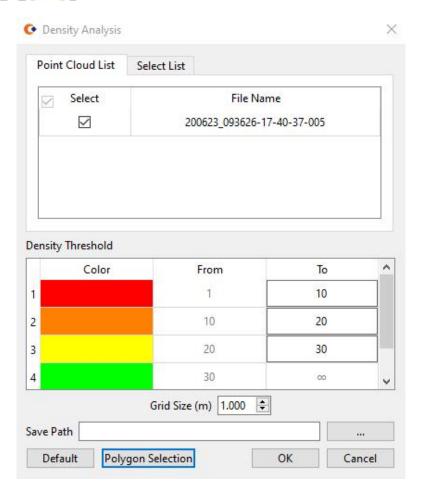
This function analyzes the density of point cloud data and outputs a html report for density quality check.

## 3.2.9.8.1. According to Point Cloud

#### **Steps**

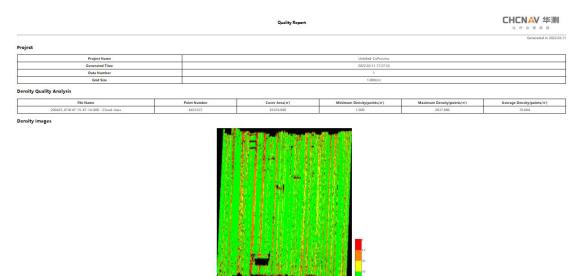
Click Point Cloud Measurement -> Density Quality Inspection, enter the density interval
 value and grid size, as shown in the following figure:





Select the output path and click OK to start the density analysis calculation. After the processing is completed, save the quality inspection results and html report in the folder.





# **Settings**

**Density Threshold:** The density of point in different threshold ranges is displayed with different colors.

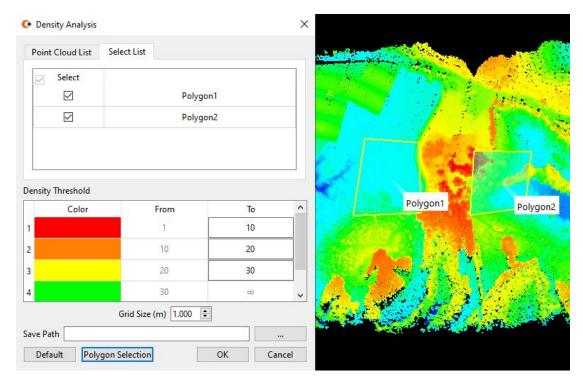
**Grid Size (m):** When the point density statistics are carried out based on the grid size as a unit, the size of the grid side length in the process of point cloud grid.

## 3.2.9.8.2. Customize Regional

## **Operation Procedure:**

- Click the density inspection, input density interval value and grid size;
- Click the polygon box and select the area to be inspected on the point cloud.
- Set the saving path and click OK.
- After processing, the selected folder will save the results of the quality check and the HTML report.



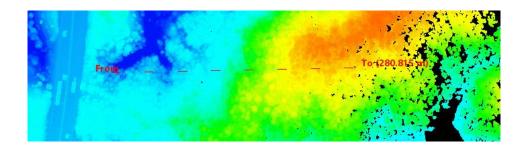


3.2.9.9. Profile Analysis

By drawing and displaying section lines, the moving and measuring functions of slices can be realized.

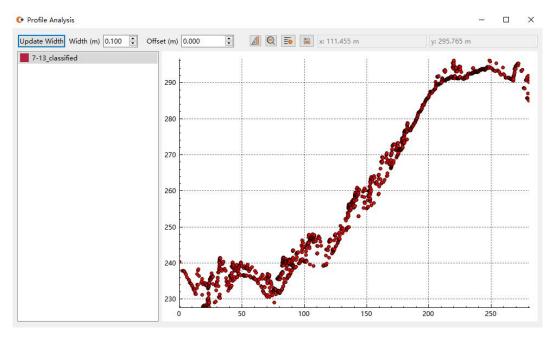
### **Steps**

 Activate the section analysis function, left-click to select a point to draw a section line, as shown in the figure below, and double-click the left-click at the last point to end the drawing.

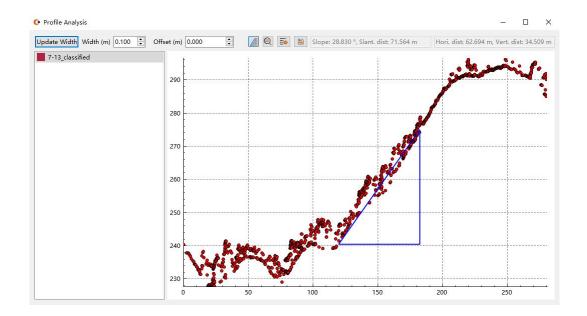


Loads the profile measurement analysis window, which provides the function of slice movement and width adjustment. Enter the slice width and click the Update Width button to display the adjusted slice. Slice movement can be controlled by modifying the offset value.





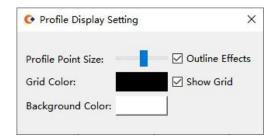
Activate the profile measurement button, drag the blue triangle with the left mouse button in the section area until the end position, the upper right side of the window displays the horizontal, vertical, hypotenuse distance and slope. Slope is the angle between the horizontal and the hypotenuse.



- By default, the section is displayed in proportion. Activate the button to display the actual size, as shown in the following figure:
- Section display settings, you can modify point size, control outline effects and grid



display, adjust section background color and grid color.



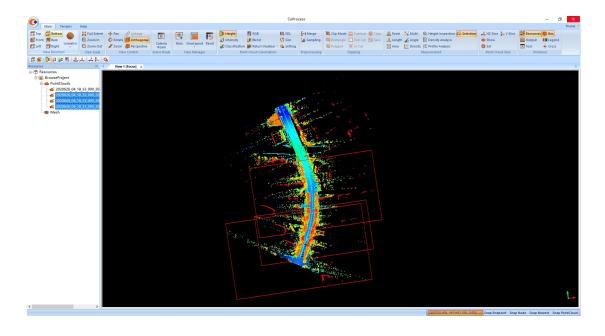
Click the Save button it to save the scene in the current profile view to the specified path.

#### 3.2.9.10. Selection

### Description

Box-select the point cloud in the view to realize the linkage between the view point cloud and the specific point cloud nodes in the resource.

- Activate the point cloud selection function and select any point cloud in the view;
- The left mouse button selects the upper left and lower right corner points in turn, and by drawing a frame selection area, you can quickly locate the point cloud list node in the workspace, and realize the linkage between view selection and point cloud nodes.





# 3.2.10.Slice

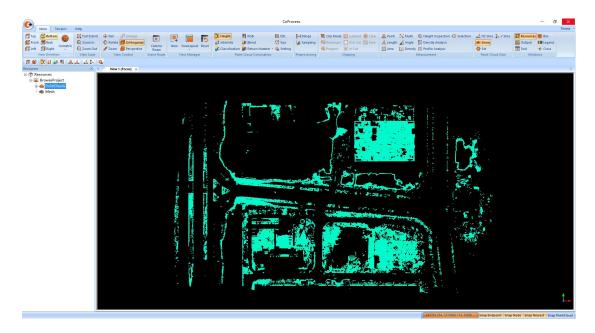
# 3.2.10.1. Horizontal Slice

# Description

Select a point to create a new horizontal slice.

#### Steps

- Click Main -> Point Cloud Slice -> Horizontal Slice;
- Left-click to select a point on the point cloud to create a new horizontal slice, hold down the ctrl key and scroll the mouse wheel to move the slice, and hold down shift to scroll the mouse wheel to adjust the display thickness of the slice.



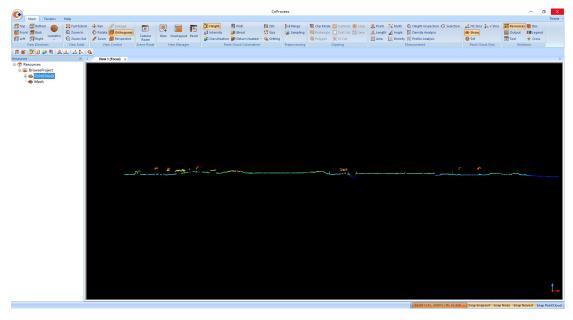
3.2.10.2. Vertical Slice

## Description

Create vertical slice.

- Click Main->Point Cloud Slice->Vertical Slice to enter the vertical slice command;
- Select two points with the left mouse button to create a new vertical slice, hold down the ctrl key and scroll the mouse wheel to move the slice, and hold down the shift key to scroll the mouse wheel to adjust the display thickness of the slice.



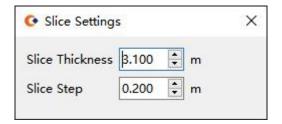


3.2.10.3. Settings

Set the slice thickness and step size, the step size indicates the stride of moving the slice.

## **Steps**

- After creating a new slice, click Home->View Slices->Slice Settings.
- The horizontal slice and vertical parameters can be set separately. After modifying the slice parameters, click Close, and the modification will take effect.



# **Settings**

Slice thickness: The default is 0.2m. By modifying the slice thickness, the vertical slice thickness can be changed. The minimum slice thickness is 0.1m.

Slice step: The default is 0.2m, which controls the sliding speed of the slice. The larger the value, the faster the scroll wheel slides up and down. The minimum slice step size is set to 0.1m.

## 3.2.10.4. Show

## **Description**

Toggles between point cloud browsing mode and slice display mode.

#### **Steps**

After creating a new slice, click Home -> Point Cloud Slice -> Show to exit the slice mode,



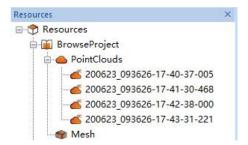
and click again to enter the slice mode. The shortcut key is Q.

# **3.2.11.Windows**

## 3.2.11.1. Resources

# **Description**

The Resources are managed according to BrowseProject and BlockProject. BrowseProject can load and display point cloud and model data.



#### **Steps**

 The workspace is checked by default. Click Home -> Windows and cancel the Resources check box to hide the Resources. Check to duplicate the Resources

## **Shortcut keys:**

Middle click any node in the Resources to display or hide the project or data.

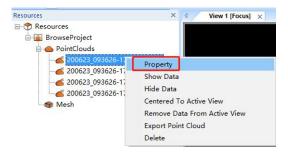
# 3.2.11.1.1.Property

## Description

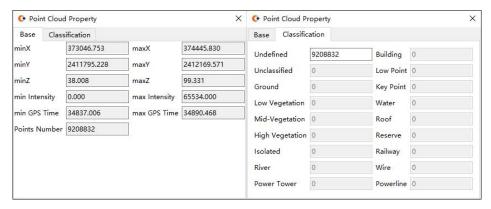
View the basic attributes and classification of the point cloud. The basic attributes include the bounding box size, time, intensity and number of points. The classification displays the number of point clouds in each category.

## **Steps**

Right click the point cloud and select "Property" to view the point cloud properties.







3.2.11.1.2.Show Data/Hide Data

# Description

Can show or hide the point cloud or mesh.



3.2.11.1.3.Centered To Active View

# Description

Center the point cloud and model data to the active view **Steps** 

When the software opens multiple views, click a view with the left mouse button to activate it. Right-click the added point cloud or model data, select "Center To Active View", it will display the data in the center view.





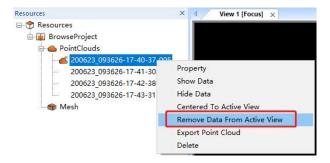
#### 3.2.11.1.4. Remove Data From Active View

# **Description**

This function is to delete the data in the current view and retain the data nodes in the Resources.

#### **Steps**

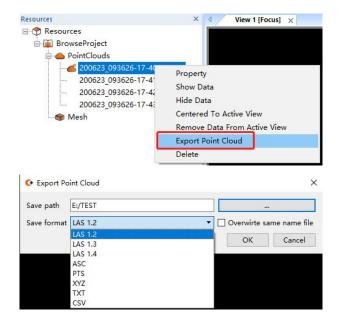
• Right click the data node and select "Remove Data From Active View" to remove data in the view. Drag data node to the view, the data will be loaded in the view again.



3.2.11.1.5.Export Point Cloud

# **Description**

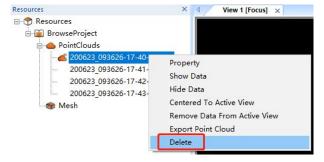
Export and save multi segment point cloud data.



3.2.11.1.6.Delete

Delet the data from Resources nodes and views.





# 3.2.11.2. Output

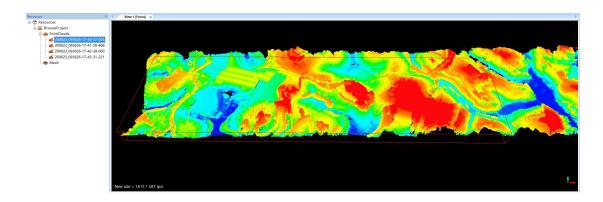
Display relevant operation information and tips. Click Home -> Windows- > Output to show or hide the output window.

3.2.11.3.**Tool** 

Show or hide the processing tool.

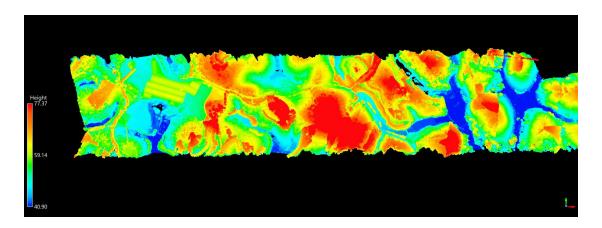
3.2.11.4.Box

Show or hide the bounding box of point cloud or mesh.



3.2.11.5. Legend

Show or hide the color bar of current render settings.



3.2.11.6. Cross

Show or hide the cross in the center of view.



# 3.3. Terrains

# 3.3.1. Import Las

## Description

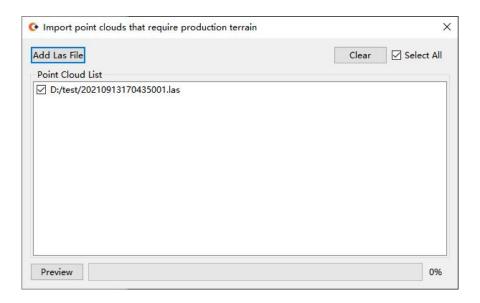
The import LAS function enables users to use las data to complete the main process of DEM production.

## **Steps**

- Click terrain Module-> Import LAS, and the dialog box for importing point clouds that need to produce terrain will pop up after clicking.
- Click the add LAS file button to select the LAS to be imported, and click Open to import LAS.
- Click the preview button to add to the workspace node tree, and subsequent users can carry out the main process of DEM production.

#### Note:

The main process of DEM production can be found in the "4.2Workflow" section of this document.



# **Parameter Description**

Add LAS file: Import point clouds that require production terrain.

Clear: Clear the imported point cloud.

Preview: Loads the selected point cloud into the workspace node tree.



# **3.3.2. Work Flow**

Include five functions: Wizard, Block, Filtering, Export Point Cloud and Export DEM.



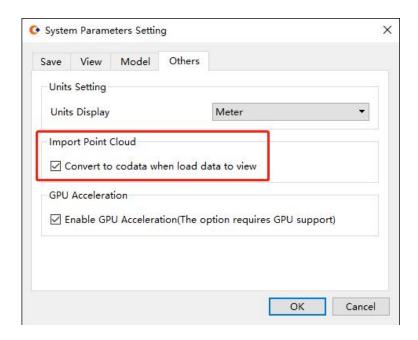
3.3.2.1. Wizard

# Description

Create a wizard to generate results of ground point or DEM.

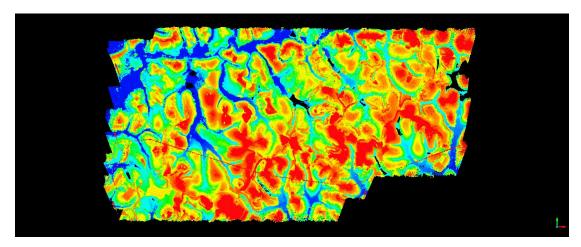
## **Steps**

 Open "System Parameters Setting" and confirm "Convert to codata when load date to view" is checked.

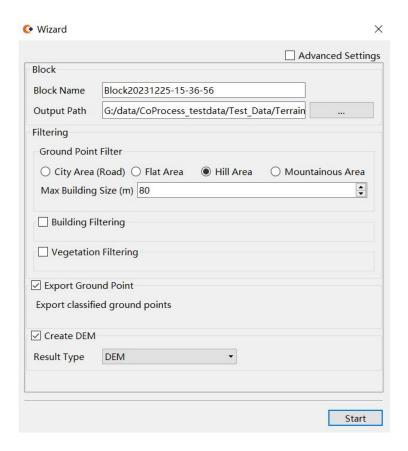


Import \*. LAS or \*.codata point cloud data need to be processed.





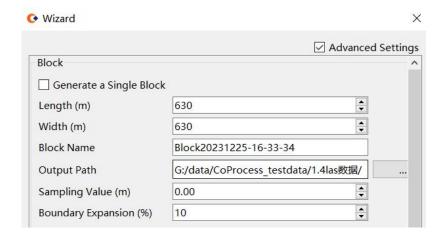
 Click Terrains -> Wizard, the processing settings dialog will poll up, which includes the settings of Block, Filtering and Export. Set parameters and click "Next".



# **Parameter Description**

**Advanced Settings:** Click Advanced settings and all settable items will be displayed. **Block Settings:** 





**Length and width:** The length and width of a single grid block.

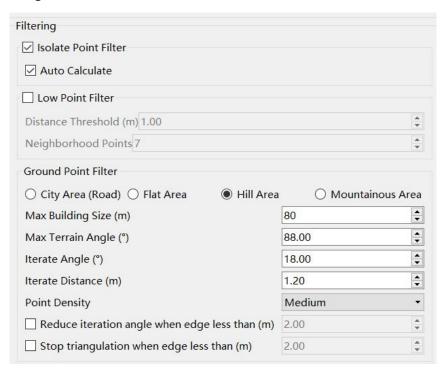
Block Name: The name of the block folder. The default is the current date.

Output Path: Data saving path.

Generate a single block: All imported data is divided into a whole block.

PointCloud Sample: After checking, set the sample interval to dilute the point cloud.

## **Filtering Settings:**



**Isolate Point Filter:** Points with less than the specified value in a certain neighborhood of point cloud data are identified as outliers. This function is mainly used to remove outliers in the air or below the ground. The isolated point deviates far from the main point cloud and has the characteristics of small number of points and discrete distribution.

Auto Calculate: When checked, the software will automatically calculate the "Search Radius"



and "Neighborhood Points" of isolate point filter.

**Ground point filter:** According to the scene settings, the software will automatically adjust the filtering parameters.

**Maximum Building Size:** The default value is 80m. Generally, it can be kept as the default, or it can be set according to the longest hypotenuse of the building.

Maximum Terrain Angle: Select the section with the steepest slope for measurement.

**Iteration Angle:** The maximum value of the angle from the point to the three vertices of the triangulation. The larger the iteration angle is, the farther the point is from the triangulation. During the iteration, the points that are far away and meet the iteration angle will be added to the ground points. The default value is  $25^{\circ}$ , which should not be set too large.

**Iterate Distance:** The vertical distance from the point to the tin plane.

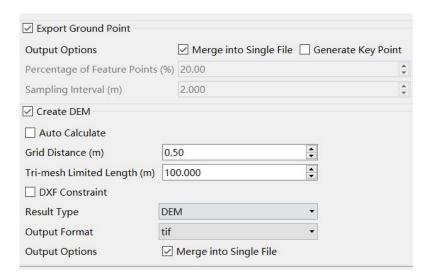
Point Density: The thinning degree of filtering results.

**Reduce the iteration angle when edge less than (m):** When the edge length of the filter triangulation network is less than the set value, the number of iterations at this point is reduced. It can effectively reduce the roughness of ground points.

**Stop triangulation when edge less than (m):** ground points whose distance is less than the set threshold will not continue to generate triangle mesh. This can prevent that the generated ground points are too dense.

#### **Notes**

It is recommended to adjust the iteration Angle first. If a large number of feature points are mistakenly classified as ground points, the iteration Angle is reduced, and vice versa. The range of each adjustment is recommended to be 3°.



#### **Export Settings:**

**Merge into single file:** When this option is checked, the output results of different blocks will be merged into one file.



**Generate Key Point:** When checked, you can set the percentage of selected feature points to export ground points.

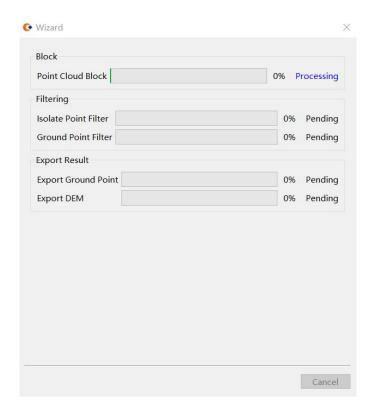
**Auto Calculate:** When this option is checked, the software will automatically calculate the grid distance.

**DXF Constrain:** When this item is checked, you can select a DXF file for constraints.

**Result Type:** You can choose DEM and DSM result type output.

Output Format: Output data in .tif format.

After setting the parameters, click Next. After confirmation, click the "Start" button, and the software will automatically process the ground points and DEM under the block project path.



 After processing, click "Finish" button, the software will automatically load block data for subsequent manual editing operations.

# 3.3.2.2. Block

# Description

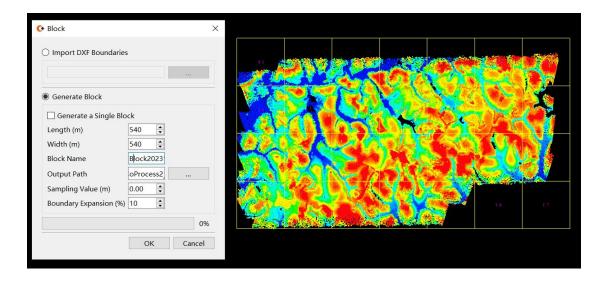
Divide point cloud data into blocks according to user-defined length and width.

# Steps

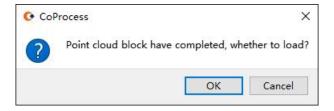
Import LAS data need to be processed.



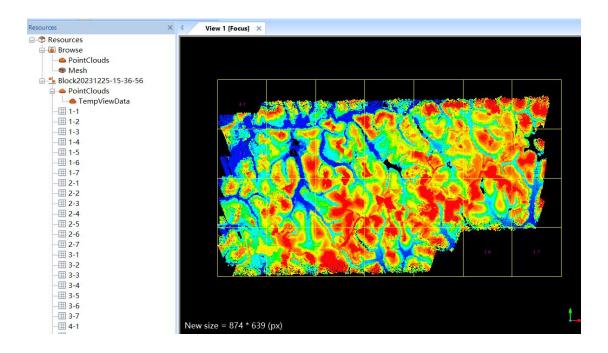
Click Terrains -> Work Flow -> Block, set the length, width, block name and output path.



 Click "OK", the block data will be generated. After generation, the software will prompt whether to load the block data.



Click "OK", the block data will be added to the software.

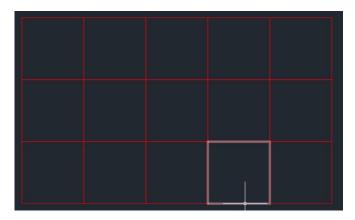




#### **Filtering Settings:**

**Import DXF Boundary:** when checked, the imported standard split grid will be used for blocking.

Requirements for the boundary files in the dxf format: Version Auto 2000/LT2000, and rectangular grids should be arranged in an adjacent manner and can not overlap.



**Length and width:** the length and width of a single grid block.

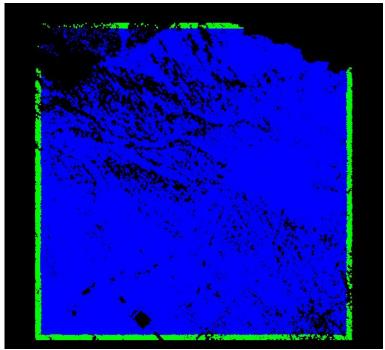
Chunk name: the name of the chunk folder. The default is the date of the day on which the chunk was built.

**Generate a single block:** when checked, the whole point cloud will be divided into the same piece when the software is divided into blocks.

**Sampling Value:** the thinning interval can be set to dilute the point cloud.

Point cloud Sampling: after checking, set the thinning interval to dilute the point cloud.

**Boundary Extension:** when checked, an extended border las file is generated under the path of each block, which is used to assist ground point classification.





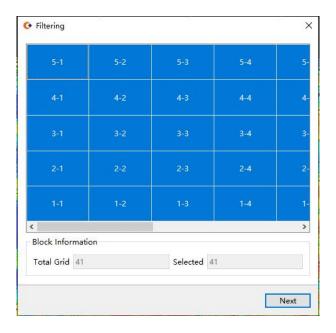
# 3.3.2.3. **Filtering**

# Description

Filtering is performed for the currently loaded block data.

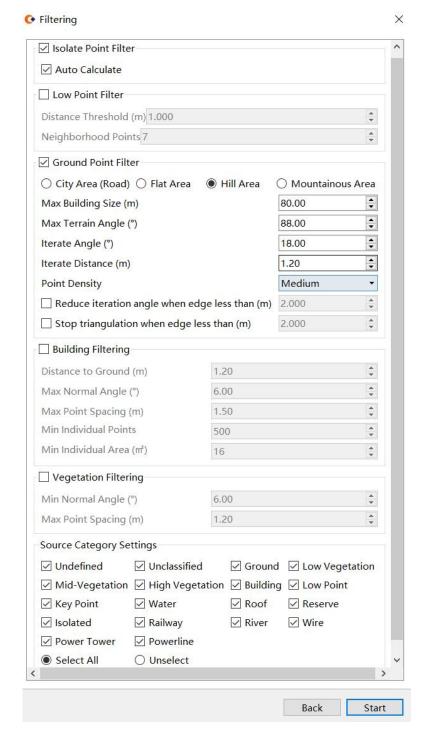
Filtering of isolated points, low points, ground points, buildings, or vegetation on all partitioned files or several files at once.

- After the block data loaded. Click Terrains -> Work Flow -> Filtering.
- Select the grid to be filtered. Support Ctrl+A for select all and other functions. Click "Next".



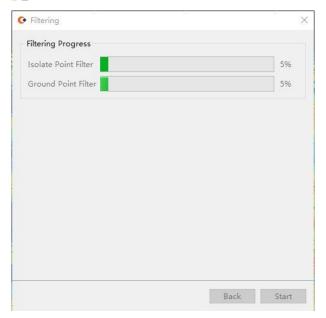
- Set parameter of isolate point filter and ground point filter. Click "Next".
- Set filtering parameters. For isolated point filtering, after selecting automatic calculation, the software will calculate the filtering parameters automatically; When unselected, you can manually set the search radius and number of neighborhood points.



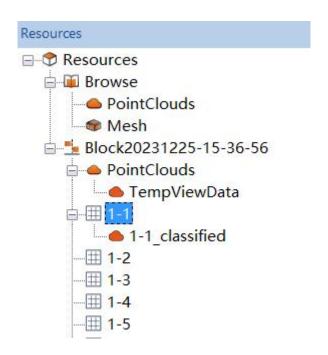


Select ground filtering, select whether to check building filtering or vegetation filtering, and click Start to proceed with filtering processing. The processing results will be saved in the block node.





As shown in the above figure, the filtered point cloud has\_ Classified suffix



# **Filtering Settings:**

**Isolate Point Filter:** Points with less than the specified value in a certain neighborhood of point cloud data are identified as outliers. This function is mainly used to remove outliers in the air or below the ground. The isolated point deviates far from the main point cloud and has the characteristics of small number of points and discrete distribution.

**Auto Calculate:** When checked, the software will automatically calculate the "Search Radius" and "Neighborhood Points" of isolate point filter.

**Ground point filter:** According to the scene settings, the software will automatically adjust the filtering parameters.

Maximum Building Size: The default value is 80m. Generally, it can be kept as the default, or



it can be set according to the longest hypotenuse of the building.

**Maximum Terrain Angle:** Select the section with the steepest slope for measurement.

**Iteration Angle:** The maximum value of the angle from the point to the three vertices of the triangulation. The larger the iteration angle is, the farther the point is from the triangulation. During the iteration, the points that are far away and meet the iteration angle will be added to the ground points. The default value is  $25^{\circ}$ , which should not be set too large.

**Iterate Distance:** The vertical distance from the point to the tin plane.

**Point Density:** The thinning degree of filtering results.

**Reduce the iteration angle when edge less than (m):** When the edge length of the filter triangulation network is less than the set value, the number of iterations at this point is reduced. It can effectively reduce the roughness of ground points.

**Stop triangulation when edge less than (m):** ground points whose distance is less than the set threshold will not continue to generate triangle mesh. This can prevent that the generated ground points are too dense.

#### **Building Filtering (Optional)**

It is used to classify building points and non-building points. By default, building filtering is not selected.

## Distance from ground:

The height of the lowest point of a building from the ground is mainly to avoid the impact of plane points close to the ground (such as cars, small rocks, etc.) on the subsequent extraction results of building contours.

It is recommended to set the range between 1 and 2.5 meters. If the scene is a low rural house, this value can be set to 1.2; The height of urban buildings is generally above 3m, and this value can be set to 2.5 to ensure the accuracy of building contour extraction.

**Note:** Setting this parameter will result in missing points at the bottom edge of the building, but it has little impact on building contour extraction.

# Maximum normal angle:

The angle value between the normals of adjacent points in a single building. The larger the value, the rougher the extracted building surface. The smaller the value, the smoother the extracted building surface. The default value is 6, and it is recommended to set the range from 2 to 6°. The larger the parameter, the more complete the building. If it is too large, some building attachment points may also be mistakenly classified as building points. The specific setting needs to be based on the actual situation of the point cloud, which has a relatively sparse density, When the thickness is relatively thin, small points can be set, such as 3. When the thickness is large and the point density is high, it can be set to 6 or above.

## Maximum point spacing:



The maximum distance allowed between adjacent building points controls the integrity of building extraction. Suggest setting the range to [1,1.5].

## Minimum number of individual points:

The number of points occupied by the smallest building in the actual scene, which is the minimum number of points required to form a single building. Buildings with fewer points than the set value will be discarded. By setting this parameter, some ground objects that are mistakenly divided into buildings, such as vehicles with fewer points, shrubs, etc., can be removed.

## Minimum area of individuals:

The number of points occupied by the smallest building in the actual scene, which is the minimum number of points required to form a single building. Buildings with fewer points than the set value will be discarded. By setting this parameter, some ground objects that are mistakenly divided into buildings, such as vehicles with fewer points, shrubs, etc., can be removed.

## **Vegetation Filtering (Optional)**

Used to classify vegetation points and non-vegetation points. When selecting vegetation filtering, ground point filtering must be synchronously selected. By default, vegetation filtering is not selected.

## Minimum normal angle:

The angle value between the normals of adjacent vegetation points, with a default value of 6, is recommended to be set within a range of  $6-10^{\circ}$ . The larger the value, the easier it is for the vegetation layer to be missed, while the smaller the setting, some surface objects may also be mistakenly classified as vegetation points. In actual settings, if there is a significant mismatch in vegetation, it is necessary to increase this parameter value appropriately, with an adjustment range of  $1-2^{\circ}$ .

# Maximum point spacing:

The maximum distance allowed between adjacent vegetation points. Mainly affects the integrity of vegetation classification. Setting it too small may result in incomplete extraction of individual vegetation, while setting it too large may cause adjacent unclassified points to be mistakenly divided into vegetation. It is recommended to set it within the range of 1-1.5m. In actual settings, adjustments may need to be made depending on the specific situation.

## **Source Classification Settings:**



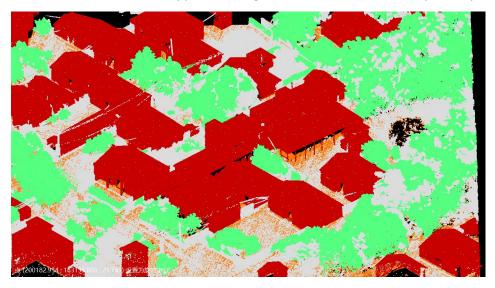
# **Function description:**

The source category refers to the point cloud category that participates in filtering. Only the selected category will participate in filtering, and the unchecked category will not participate in filtering processing.

Source Category Settings			
✓ Undefined	✓ Unclassified	✓ Ground	✓ Low Vegetation
✓ Mid-Vegetation	✓ High Vegetation	✓ Building	✓ Low Point
✓ Key Point	✓ Water	✓ Roof	✓ Reserve
✓ Isolated	✓ Railway	✓ River	✓ Wire
✓ Power Tower	✓ Powerline		
Select All	○ Unselect		

Figure: Source Classification Settings:

This setting can be used to preserve point cloud categories that already exist in the original data, but the software does not support filtering. But it needs to be manually classify.



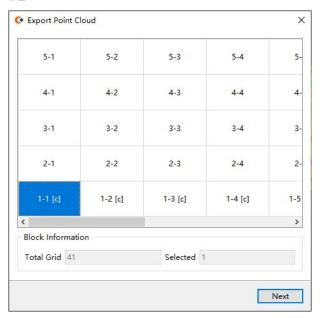
3.3.2.4. Export

# **Description**

This function can export point data after classification.

- Load the block file and click Terrains -> Work Flow -> Export Point Cloud.
- Select the gird to be exported. Click "Next".



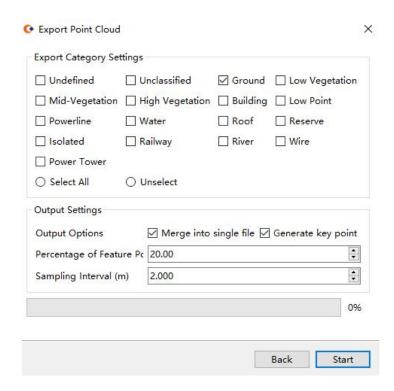


## **Notes**

If the grid data has been classified, there will be a "[c]" after the grid name.



Set the category need to be exported (Default is ground). Click "Export".





You will get the point cloud exported in the block folder path.



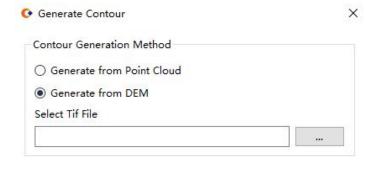
3.3.2.5. **DEM** 

# **Description**

This function generates DEM in TIF format from point cloud data.

# **Steps**

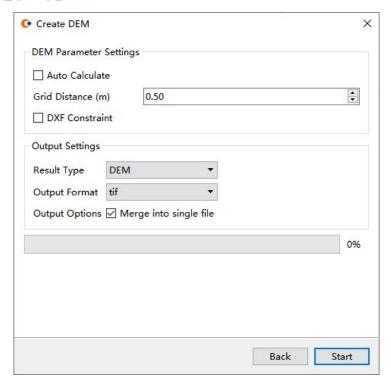
- Load the block data, Click Terrains -> Work Flow -> Create DEM.
- Select the gird to be exported. Click "Next".





Set DEM parameter settings and output settings. Click "Start".





#### **Notes**

DEM is generated based on classified ground points and DSM is generated based on all points.

You will get the DEM/DSM exported in the block folder path.



# **DXF Constraint:**

When DXF constraint is checked, users need to import several range lines in DXF format.

When generating DEM, the software will use the imported scope line to trim the result DEM according to the user's settings, or at the same time, the points on the scope line participate in the generation of DEM'.

## 3.3.2.6. Contour

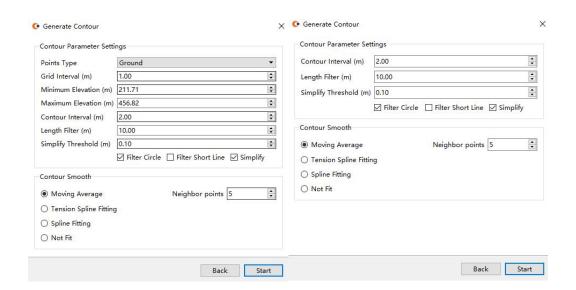
## **Description**

This function can generate contour data in dxf format based on point cloud data or DEM.

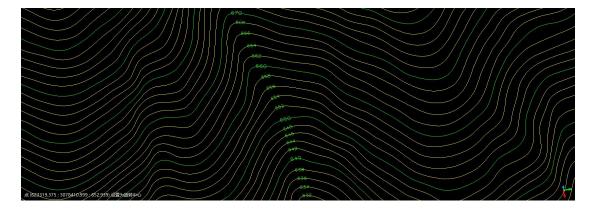
- Load the block data, Click Terrains -> Work Flow -> Contour.
- Choose to generate contours based on point cloud or DEM.



 Set contour generation parameters, and select whether to enable contour smoothing and simplification. (left is the setting interface of generating contours based on point cloud, right is the setting interface of generating contours based on DEM)



 Click "Start", after the processing is completed, the contours will be output to the contour folder under the block project folder.



# Parameter setting:

**Points type:** You can choose ground points or all points. If non-lidar data is used to generate contours, it is recommended to use all points to generate contours.

**Grid Interval:** The spacing of the network during the generation of contour lines, which affects the accuracy of contour lines.

**Contour Interval:** The height difference of contour lines during the generation of contour lines, which affects the density of contour lines.

**Minimum/maximum Elevation:** The elevation range in which contour lines are generated. The software automatically obtains the minimum and maximum elevations of the point cloud in the selected scene.



**Length Filter:** After generating contour lines, lines whose length is less than the set length are automatically deleted from the results.

**Simplify:** If the distance between the current point and the two points around it in a straight line is less than the set value, the current point will be deleted, can reduce the number of points without changing their shape.

**Contour smooth:** The software provides three contour smoothing methods: sliding mean, spline fitting and tension spline fitting. Generally, sliding means can be used for data.

**Neighbor points:** The number of points around which each point is smoothed. The parameter must be odd.

**Moving Average:** The position of the point to be measured is obtained based on the weighted fitting of adjacent points, which is suitable for smoothing contour lines generated by high-density point cloud data.

**Spline Fitting:** By constructing piecewise polynomial fitting data, combined with second-order differentiability, local smoothing processing is realized, which is suitable for smoothing low-density contour data.

**Tension spline Fitting:** Curve interpolation is carried out through adjacent points, and vertex correction is achieved by using a certain proportion coefficient, which is suitable for smoothing low-density contour data.

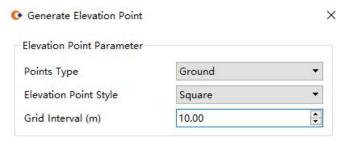
#### 3.3.2.7. Elevation Point

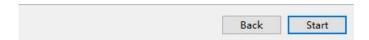
# Description

Based on the point cloud data, this function can generate the elevation points in DAT format according to the square grid or diamond grid.

- Load the block data, Click Terrains -> Work Flow -> Elevation Point.
- Select the gird to be exported.
- Click Next to set elevation point generation parameters;
- After confirming the parameters, click Start;
- After the processing progress is completed, the elevation point data will be output successfully.





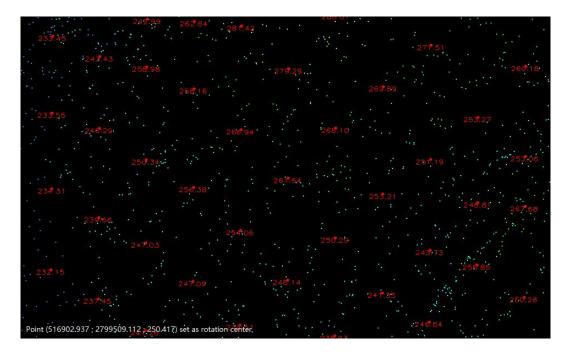


# Parameter setting:

**Points Type:** You can choose ground points or all points. If non-lidar data is used to generate contours, it is recommended to use all points to generate contours.

**Elevation Point Style:** Square or diamond can be selected, and the resulting elevation points will be arranged according to the set shape.

**Grid Interval:** The side length of the grid used to fit the elevation when generating elevation points.





# 3.3.3. Block Management

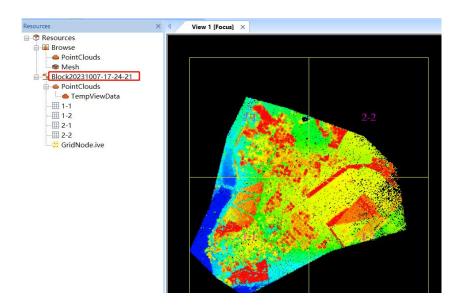
# 3.3.3.1. Load

## Description

Load and display block data.

#### **Steps**

 Click Terrains -> Block Management -> Load, select block data, and click open to load and display block data.



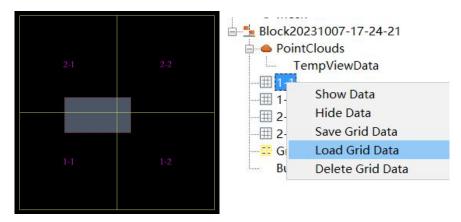
3.3.3.2. Select

## Description

Load and display the point cloud data in the block grid.

- After loading the block data, click Terrains -> Block Management -> Select. Draw a rectangular box to select the grid, and then the point cloud data corresponding to the grid will be loaded.
- Right click on a grid in the resources node tree, and then click "Load grid data" in the right menu can also display the point cloud data in the block grid.





3.3.3.3. Save

## Description

Manually save the data such as point cloud, TIN, contour line and elevation point in the current display block grid.

#### Steps

 After the block project is loaded, click the block Save function in Block Management to manually write the current edit result to the local block project.

## **3.3.3.4. Save Contour**

## Description

Manually save the currently displayed contour data.

#### Steps

 After the block project is loaded, click the block Save function in Block Management to manually write the current edit result to the local block project.

# 3.3.4. Category Editing

## 3.3.4.1. Fast Filter

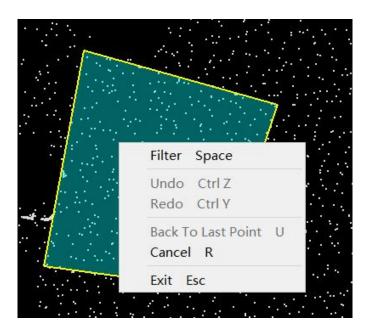
#### Description

Filtering processing is carried out for the point cloud in the selected area.

- Select the area of the polygon with the left mouse button on the active view and right-click to end the selection. After box selecting all the filter areas, right click to carry out the filtering processing.
- Click "Filter" to filter the selected polygon region, and the shortcut key is Space.



- (1) "Undo" can undo the previous filtering result, and the shortcut key is Ctrl + Z.
- (2) "Redo" can restore the filtering result before the last undo operation, and the shortcut key is Ctrl + Y.
  - (3) "Back To Last Point" can cancel the last points drawn, and the shortcut key is U.
  - (4) "Cancel" can cancel all the points being drawn, and the shortcut key is R.
  - (5) "Exit": Exit the fast filter command, and the shortcut key is Esc.



The "Fast Filter" function also supports the boundary expansion.

3.3.4.2. TIN

3.3.4.2.1. TIN

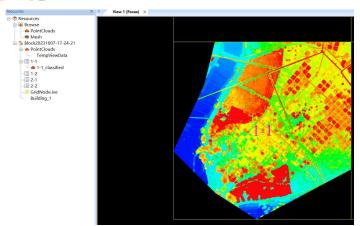
# Description

The TIN function is only for ground points in the point cloud that are currently displayed. Can realize TIN browsing and point cloud and TIN synchronous editing.

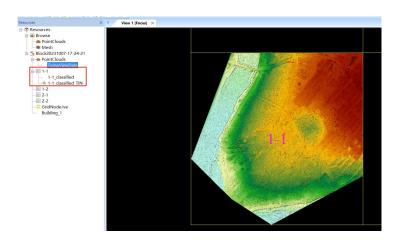
## **Steps**

Load block data and select blocks to be constructed for TIN;





 Click Terrains -> Category Editing -> TIN, the TIN will be constructed from the displayed grid data. And TIN data will be automatically loaded to the corresponding block node.



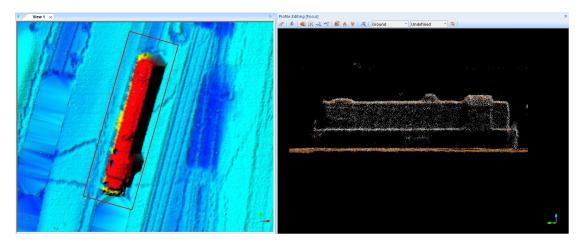
3.3.4.2.2. Profile Editing and TIN refresh

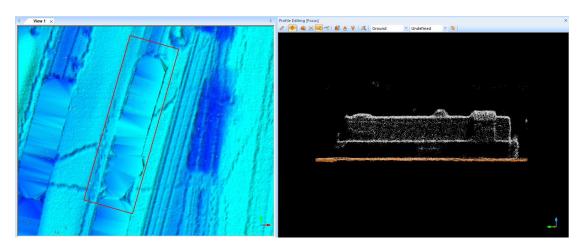
# Description

When ground points are edited in the profile editing, the TIN in the main view will update synchronously.

- Load block data with ground points, and construct TIN.
- Open the profile editing. Select the area you want to edit. Edit ground points in the
   Profile editing window, TIN will update synchronously.







As shown in the figure above, we painted the misdivided ground points in the profile editing window on the right as undefined points, and the TIN on the left was also modified simultaneously.

# 3.3.4.3. Main-View Classify

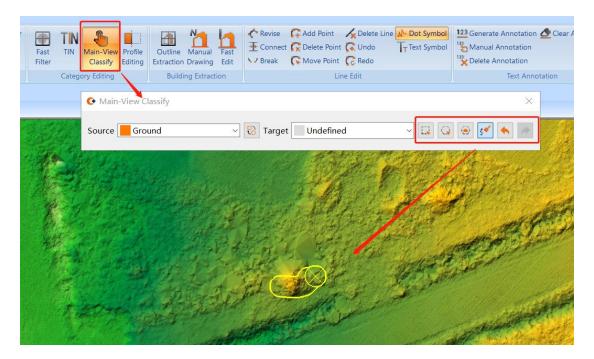
## Description

If it is obvious that some vegetation and noise points are not correctly classified, you can use the main view editing function to directly batch classify without drawing the section.

# Steps:

Click Main-View Classify, and the dialog box will appear.



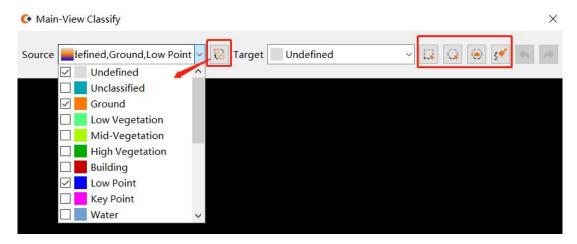


- Set the source class and target class. You can set more than ten different categories. In addition, the source category can select multiple tags;
- Click the Rectangle Tool, Polygon Tool or Brush Tool to mark the areas that need to be reclassified on the main view;
- After drawing, the category of points belonging to the source class in the region will be modified into the target category.

# 3.3.4.3.1. Source Category Multiple Selection Mode

Click the button next to the source category to switch to the source category multiple selection mode. This editing mode enables multiple types of source category selection.





3.3.4.3.2. Editing Tool

The editing tool has four types: Rectangle, Polygon classification- Screen Select, Polygon classification- Model Select and Brush. The operation methods are as follows.

## (1) Rectangle

After the "Rectangle" tool is activated, long press and drag on the main view to draw a rectangular box, release the left mouse button, then the attribute category of the points in the drawn rectangle area will be changed.

#### (2)Polygon classification- Screen Select

Left click multiple points in turn in the main view to draw a closed polygon, double click the left button on the last point to end the drawing, and then you can modify the category information of the points inside the polygon;

The size and location of the polygon based on the "Screen Select" method are fixed on the screen and do not vary as the model zooms in, zooms out and moves.

## (3) Polygon classification- Model Select

Left click multiple points in turn on the point cloud to draw a closed polygon, double click the left button on the last point to end the drawing, and then you can modify the category information of the points inside the polygon;

The polygon drawn based on the "Model Select" method varies as the model zooms in, zooms out and moves.

#### (4) Brush



After the "Brush" tool is activated, long press and drag on the main view to draw an area, release the left mouse button, then the attribute category of the points in the drawn area will be changed.

Besides, before using the "Brush" tool, you can adjust the brush size with Ctrl+Wheel.

## 3.3.4.3.3. Undo/Redo

By clicking "Undo" and "Redo", you can undo the last edit, or roll back the undo operation.

And the "Undo" operation and "Redo" operation can be carried out for a maximum of 10 times.

# 3.3.4.4. Profile Editing

## 3.3.4.4.1. Profile select

## Description

Draw the section in the main view by selecting points, and display the section in the section editing view.

## **Steps**

- Click Terrains -> Profile Editing to open the profile editing view.
- On the main view, left-click two points to determine the position of the section.
- The point cloud of section will display in the profile editing view.



## 3.3.4.4.2. View linkage

Activate the view linkage icon . When you move or scale the profile editing view, the main view moves with it.



#### 3.3.4.4.3. Lock rotation

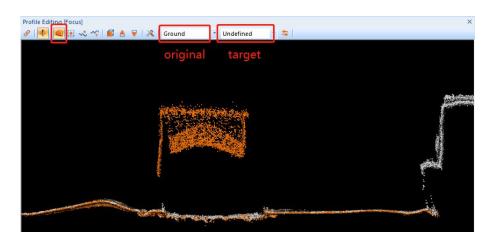
Activate the lock rotation icon  $\stackrel{4}{=}$ . The point cloud of the profile editing view cannot be rotated.

# 3.3.4.4.4. Polygon classify

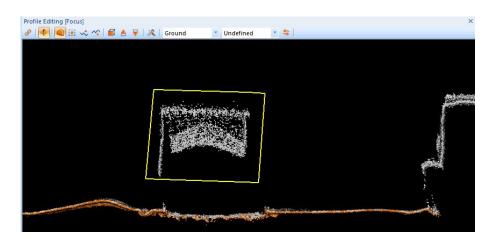
# Description

Draw a polygon in the profile editing view to classify the original category to target category. **Steps** 

After loading the profile data, active polygon classify icon , and then set the original category and target category in the toolbar above the profile editing view.



Draw polygon area in the profile editing view, left-click and select points to draw polygon. Double left click to finish drawing. The points of original category will be classified to target category.





# 3.3.4.4.5. Line above/below classify (shortkey:W/S)

## Description

Draw a polyline and classify the original category above/below the polyline to the target category.

## Steps

- After loading the profile data, active line above/below classify icon <sup>∞</sup>/<sup>∞</sup>, and then set the original category and target category in the toolbar above the profile editing view.
- Draw a polyline in the profile editing view, left-click and select points to draw polyline.
  Double left click to finish drawing. The points of original category above/below the polyline will be classified to target category.

# 3.3.4.4.6. Frame selection classify

## Description

Draw a rectangle according to the center line to classify the original class into the target class.

## **Steps**

- After loading the profile data, active frame selection classify icon \_\_\_\_\_, and then set the
   original category and target category in the toolbar above the profile editing view.
- Left click at least two points respectively to determine the center line of the rectangle,
   move the mouse to determine the width of the rectangle, double-click the end

## 3.3.4.4.7. Brush classify (shortkey:X)

# Description

Drag and drop in the Profile Edit view to classify the original category in the area into the target category.

- After creating a new profile, Click the Brush Categorization , set the source category and target category in the toolbar above the Profile Edit view.
- drag and drop in Profile Edit view to draw an area, and set the selected area category to



the target category.

#### 3.3.4.4.8. Front View



Resets the perspective of the section view to front view.

# 3.3.4.4.9. Profile up

# Description

This feature allows you to move up a section area of the same size over a section area that has been drawn.

Shortcut key:

Press † on the keyboard to quickly move up the profile.

#### 3.3.4.4.10.Profile down

# Description

This feature allows you to move down a section area of the same size over a section area that has been drawn.

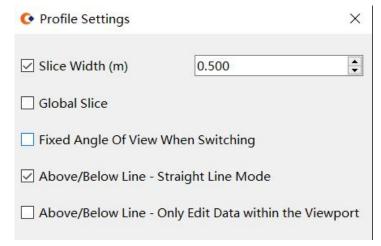
Shortcut key:

Press the keyboard  $\downarrow$ , can quickly move down the profile.

# 3.3.4.4.11. Profile settings

# **Description**

Set the rotation angle and expansion width of the drawn profile, and the interaction mode of the area above/below the line.





### **Profile settings**

#### **Steps**

Click the profile settings icon , set the profile parameters.

### **Parameter Description**

**Slice width:** when checked, the slice width is fixed at the set value. If not selected, the width can be manually adjusted when drawing the profile.

Global Slice: when checked, the entire point cloud will be profiled.

**Fixed view when switching:** When this setting is checked, the view will no longer scale to the entire section after switching the profile.

Above/Below Line - Straight Line Mode: After checking this option, activate the "Above/Below the line" tool, left click two points to draw a straight line to determine the area above the line and the area below the line, and then modify the categories of the point clouds in the area above/below the straight line; If this option is not checked, activate the "Above/Below the line" tool, left click at least two points respectively to determine the area above the line and the area below the line, double-click the end the operation, and then modify the categories of the point clouds in the area above/below the line.

Above/Below Line - Only Edit Data Within the Viewport: After checking this option, activate the "Above/Below the line" tool to modify only the categories of the point clouds in the area above/below the line that have been drawn in the viewport; If this option is not checked, activate the "Above/Below the line" tool to modify the categories of all point clouds in the area above/below the line that have been drawn.

#### 3.3.4.4.12.Switchover



Toggle source category and target category.

#### 3.3.4.4.13.Undo/Redo



Click Undo and Redo to undo the last edit or to roll back the undo action. Undo and redo supports a maximum of 10 times.

# 3.3.5. Building Extraction

#### 3.3.5.1.Outline Extraction

3.3.5.1.1. Single Mode

Single building mode



### description:

Extract individual building outlines for the selected area.

#### Steps:

- 1. Click on Outline Extraction, activate the processing tool, and select "Single Building Mode".
- 2. Left click on the active view to draw a polygonal area, select all the building areas to be extracted, and then right-click to extract.
- 3. Click "Extract" to extract the building outline of the selected polygon area.

During the process of drawing a polygon, right-click and select 'undo' to cancel the previous point drawn, select 'cancel' to cancel all currently drawn points, and double-click to end the drawing.

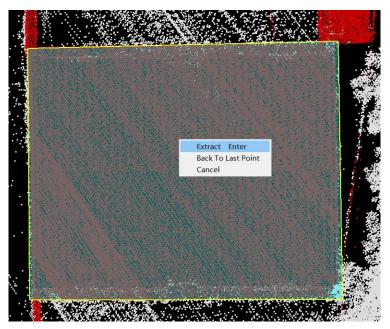


Figure: Single building mode

# 3.3.5.1.2. Multi building mode

# **Function description:**

Extract multiple building outlines for the selected area.

# Steps:

1. Click on Quick Building Extraction, activate the processing tool, select "Multi Building Mode", and set building extraction parameters as needed;

Left click on the active view to draw a polygonal area, select all filtering areas in the box, and then right-click to process.

3. Click "Extract" to extract the building outline of the selected polygon area.

During the process of drawing a polygon, right-click and select 'undo' to cancel the previous point drawn, select 'cancel' to cancel all currently drawn points, and double-click to end the drawing.



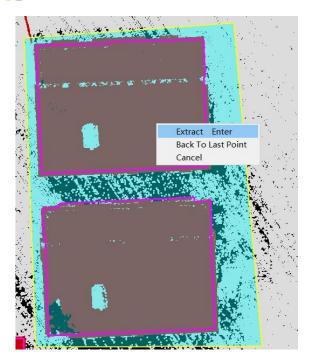


Figure: Multi building mode

Parameter settings:

**Maximum point spacing:** The maximum distance between adjacent points of a single building affects the accuracy of extracting building outlines.

**Minimum number of individual points**: The number of point clouds for a single building and building contours with points less than this threshold are not extracted.

**Minimum area of individual:** The minimum projected area of a single building, and building contours below this threshold are not extracted.

#### 3.3.5.2. Manual Drawing

#### **Function description:**

Draw the buildings outline manually. This function supports three interactive mode including right angle, by edge and polygon, and the short key is N.

Right angle: The short key is Z;

By edge: The shortcut key is T; Polygon: The shortcut key is C;

Take a step back: Back to the previous point, and the shortcut key is U;

Cancel: Cancel this drawing process, and the shortcut key is R;

Exit: Exit the manual drawing command, and the shortcut key is Esc.



### 3.3.5.2.1. Right angle drawing

#### Steps:

- 1. Click "Manual Drawing", right click "Switch Interaction" and choose "Right angle";
- 2. In the main view, use the left mouse to click two points to determine the direction of the first edge, then click on the other edges in turn;
- 3. Press Enter or double click, or right click to finish the drawing, and the buildings outline will automatically close.

# 3.3.5.2.2. Line drawing

### Steps:

- 1. Click "Manual Drawing", right click "Switch Interaction" and choose "By edge";
- 2. In the main view, use the left mouse to click two points to determine the direction of the first edge, then click two points on the other edges in turn;
- 3. Press Enter or double click, or right click to finish the drawing, and the buildings outline will automatically close.

# 3.3.5.2.3. Polygon drawing

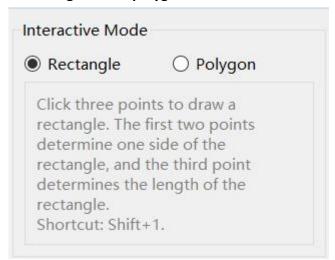
# Steps:

- 1. Click "Manual Drawing", right click "Switch Interaction" and choose "Polygon";
- 2. In the main view, click multiple points to draw a polygon;
- 3. Press Enter or double click, or right click to finish the drawing, and the buildings outline will automatically close.

**3.3.5.3. Fast Editing** 

#### 3.3.5.3.1. Interaction mode

Two interaction modes: rectangular and polygonal.



Rectangle: After selecting the rectangular interaction mode, left click two points on the main



view to determine an edge of the rectangle. Move the mouse and click the left mouse button to draw a rectangular box. Release the left mouse button to correct the building outline. The shortcut key is shift + 1.



**Polygons:** After selecting the polygon interaction mode, left click on multiple points in the main view to draw a polygon. Double click on the left mouse button to end the drawing and correct the building outline. The shortcut key is shift + 2.

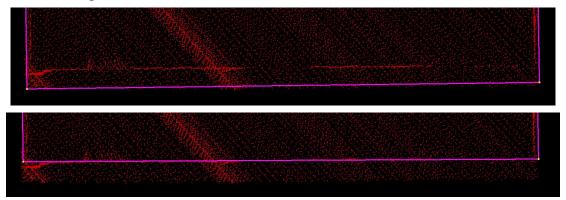
# 3.3.5.3.2. Editing mode

#### 3.3.5.3.2.1. Facade Correction

Function description: Box select the outlines and building elevation point clouds that do not coincide with the building elevation, and automatically correct the outlines to the position that coincides with the elevation. The shortcut key is ctrl + 2.

#### Steps:

- 1. Click on Building Edit to activate the processing tool dialog box.
- 2. Select the rectangular interaction mode in the right window processing tool.
- 3. Select the "Facade Correction" function in editing mode.
- 4. Draw the rectangle that will be included in the modified area in the main view. After drawing, correct the outline to coincide with the elevation.



# 3.3.5.3.2.2. Straightening

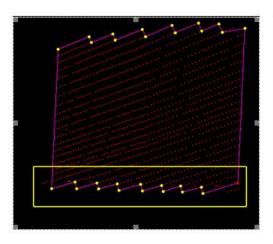
#### **Function description:**

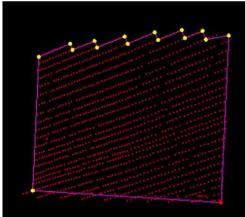
Box select a jagged building outline and building point cloud to straighten the outline. The shortcut key is ctrl + 1.

- 1. Click on Building Edit to activate the processing tool dialog box.
- 2. Select the polygon interaction mode in the right window processing tool.



- 3. Select the "Straightening" function in editing mode.
- 4. Draw the polygons included in the modified area in the main view, double-click the left mouse button to end the drawing, and the building outline is straightened as shown in the figure.



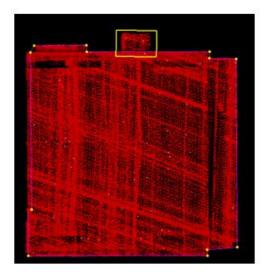


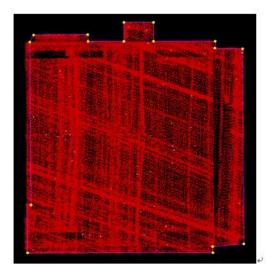
# 3.3.5.3.2.3. Extend Object Extraction

Function description: Select the building outline and the rectangular attachment point cloud extending below the eaves, and automatically correct the outline to a vector line containing the rectangular extension object. The shortcut key is ctrl + 3.

Steps:

- 1. Click on Building Edit to activate the processing tool dialog box.
- 2. Select the rectangular interaction mode in the right window processing tool.
- 3. Select the "Extend Object Extraction" function in editing mode.
- 4. Draw the rectangle included in the modified area in the main view. After drawing, the outline is automatically corrected to a vector line containing the rectangular extension object.





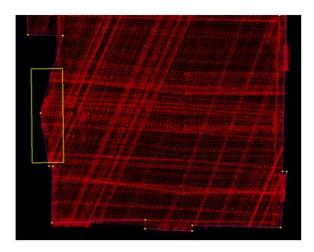


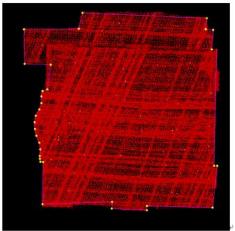
# 3.3.5.3.2.4. Arc Edge Extraction

Function description: Box select the building outline and arc building point cloud, and automatically correct the outline to the position that fits with the arc building point cloud. The shortcut key is ctrl + 4.

# Steps:

- 1. Click on Building Edit to activate the processing tool dialog box.
- 2. Select the rectangular interaction mode in the right window processing tool.
- 3. Select the "Arc Edge Extraction" function in the editing mode.
- 4. Draw the rectangle included in the modified area in the main view, and when the drawing is completed, the outline will automatically be corrected to the position that fits with the arc building point cloud.





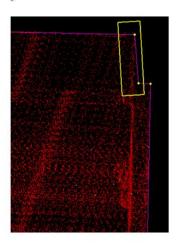
# 3.3.5.3.2.5. Rectangular Processing

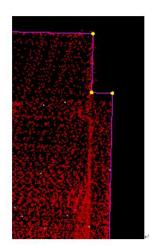
### Function description:

Make rough edge of buildings vertical. The shortcut key is ctrl +5.

- 1. Click on Building Edit to activate the processing tool dialog box
- 2. Select the "Rectangular Interaction" mode in the right window processing tool
- 3. Select the "Rectangular Processing" function in the editing mode.
- 4. Draw the rectangle in modified area in the main view. After drawing, the outline angle will be automatically corrected to a vertical.





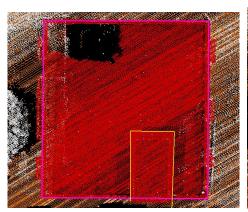


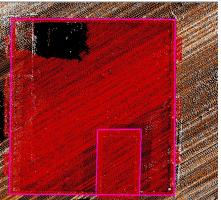
# 3.3.5.3.2.6. Cutting

# Function description:

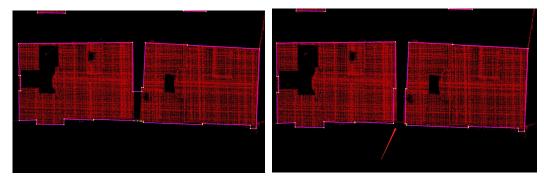
Draw a multi-segment line across the buildings outline, cut the buildings outline into two closed buildings outlines according to the multi-segment line and the cutting width. The shortcut key is ctrl+6.

- 1. Click on Building Edit to activate the processing tool dialog box;
- 2. Select the "Cut" function in the editing mode;
- 3. Set a value for the cutting width parameter, and the default value is 0m;
- 4. Left mouse button click multiple points, draw a multi-segment line across the buildings, and double click to finish the drawing process, then the buildings outline is cut into two closed buildings outlines.









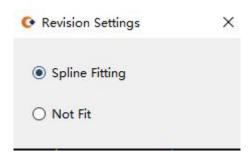
# 3.3.6. Contour Edit

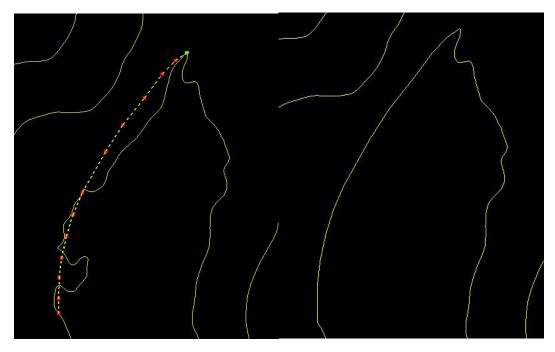
Use various tools to edit the automatically generated contour lines. After editing, click Save Block to write the changes to the local file.

# 3.3.6.1. Revise

# Description

Manually click several points to modify a section of a contour line and change its shape. The shortcut key is K.







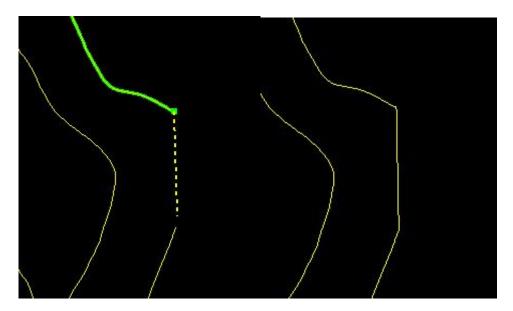
- Click the button or press the shortcut key K to activate the repair function;
- Click on a point on the contour you want to edit, then click on multiple points in succession according to the actual terrain until the color of the point turns green;
- After all the drawing is completed, the single mouse right click and select "Finish";
- Once this is done, a portion of the original contour line will be replaced with a manually drawn shape.

In addition, the contour repair function can connect two different contour lines, but only for the same elevation of the line.

#### 3.3.6.2. Connect

# Description

Manually click to select two contours to merge the two lines into one. The shortcut key is L.



- Click the button or press the shortcut key L to activate the connection function;
- Click the left mouse button on the first contour line to be connected, and the contour line will become highlighted;
- Click the left mouse button on another contour line with the same elevation;
- When this is done, the two contours will be combined into one.



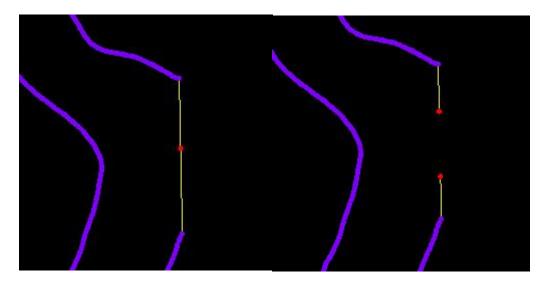
#### Notes

Contour merging is only effective for contours with the same elevation

3.3.6.3. Break

#### Description

Manually click a point or two points on the contour line to form a line segment, breaking one contour line into two. Single break shortcut key: 1, two break shortcut key: 2



### **Steps**

- Click the button or press the shortcut key 1 or 2 to activate the interrupt function. By default, the two-point interrupt mode is activated;
- In the case of single point interruption, press shortcut key 1, and then click the left button on the contour line to break one line into two;
- When two points break, press the shortcut key 2, click the left button on the contour line, and then click the left button again on the same contour line
- When the operation is complete, one contour line will be broken into two.

### 3.3.6.4. Add Point

### Description

Add A node to the contour line, shortcut is A.

- Click the button or press the shortcut key A to activate the dot function;
- Click the left button on the contour line to add a node to the selected location.



You are advised to enable the dot symbol function when using the dot function.

#### **3.3.6.5. Delete Point**

# Description

Delete a node on the contour line, shortcut is D.

#### Steps

- Click the button or press the shortcut key D to activate the deletion point function;
- To delete a node in the selected location, click the left button on a node in the contour line.

You are advised to enable the dot symbol function when using the dot deletion function.

#### 3.3.6.6. **Move Point**

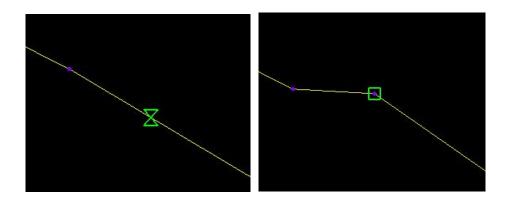
# Description

Move a node on the contour line, shortcut is M.

#### **Steps**

- Click the button or press the shortcut key M to activate the shift function;
- Long press the left mouse button on the contour line, drag the point to the target position, and release the left mouse button to move the node.

If there is no node on the vector line of the mouse capture position, then a new node will be added to the mouse capture position.



Move point

#### 3.3.6.7. **Delete Line**

#### Description

To delete a contour line, shortcut is E.



- Click the button or press the shortcut key E to activate the line deletion function;
- Click two points on the view to draw a rectangular box. If drawn from left to right, contours where the entire line is within the area are deleted; If drawn from right to left, this contour will be deleted if it is part of the right contour within the region.

#### 3.3.6.8. Undo

#### Description

Undo the last edit to the contour.

#### **Steps**

Click the button or use Ctrl+Z to undo the contour editing.

#### 3.3.6.9. Redo

### Description

Rollback of the last contour undo operation.

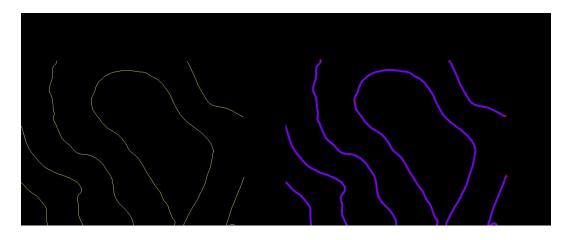
#### **Steps**

Click the button or use Ctrl+Y to undo the contour back and forth.

# 3.3.6.10. **Dot Symbol**

# Description

With the dot symbol display enabled, the nodes of the contour line will be displayed in the view. The first and last points are shown in red and the middle nodes are shown in purple.

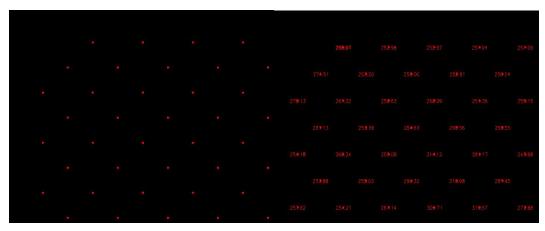


3.3.6.11. **Text Symbol** 

#### Description

After text symbol display is enabled, elevation information about elevation points is displayed in the view.

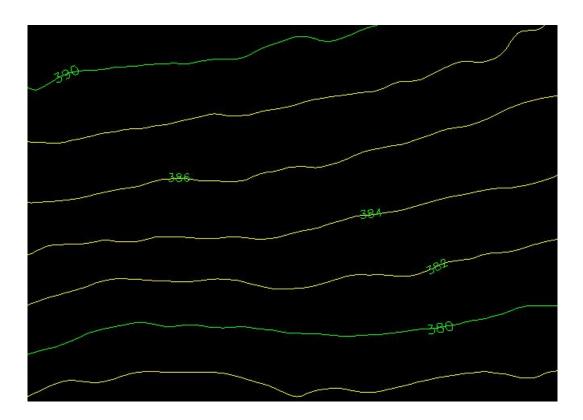




# 3.3.7. Text Annoation

# Description

Batch generation and deletion of elevation notes on contour lines, elevation notes will be saved with contour dxf.



3.3.7.1. Generate Annoation

#### Description

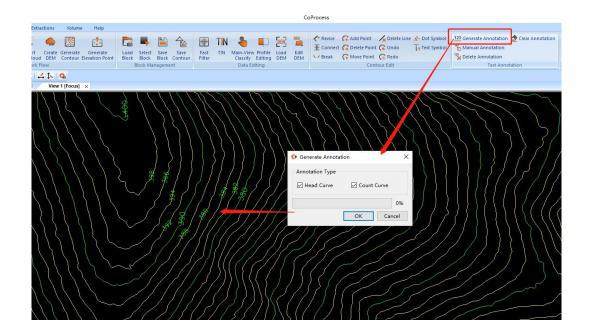
Generate an elevation note at the middle of each contour line in the scene.

# **Steps**

After the contour is generated, click the Generate Note button;



- In the pop-up dialog box, check the type of contour line to generate notes, including the first curve and the count curve;
- Click OK to generate contour notes.

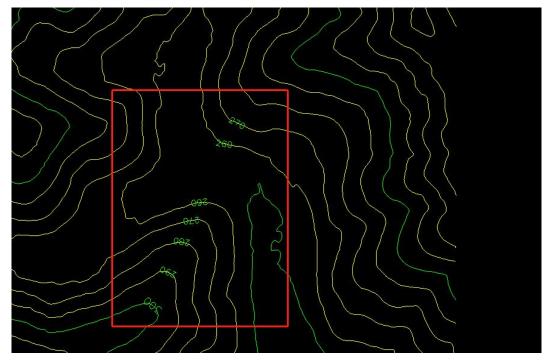


3.3.7.2. Manual Annotation

Manually draw a line in the view and then generate a note at the intersection of the drawn line and the contour line.

- After the contour is generated, click the manual note button;
- Click two points on the view to draw a line segment that intersects the contour line;
- After the second point is drawn, contour notes are extracted from all the intersections.





#### **Notes**

If there is no elevation difference between the contour line intersected with the drawn line, the extraction direction may be wrong.

If the drawn line and the contour line have only one intersection, the note will be generated in the direction of the drawn line.

#### 3.3.7.3. Delete Annotation

#### Description

Click on two points to draw a rectangular box with all the notes inside removed.

#### **Steps**

- After the contour line and text annotation are generated, click Delete annotation button;
- Click two points on the view to draw a rectangular box;
- After the second point is drawn, all the notes in the rectangle will be deleted.

### 3.3.7.4. Clear Annotation

# Description

Delete all comments from the view in batches.

- After generating the contour line and text note, click the Clear note button;
- All text annotations in the view are deleted.



# 3.3.8. DEM

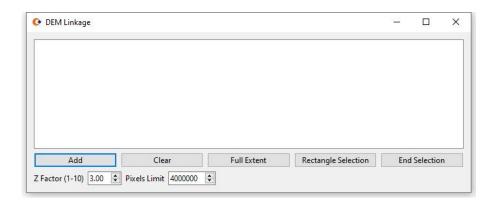
#### 3.3.8.1. Load DEM

#### Description

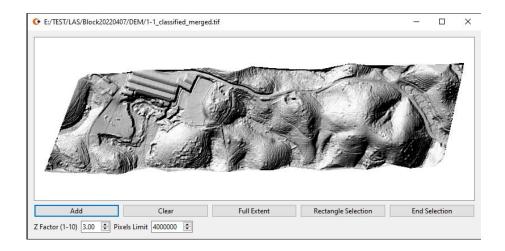
The DEM linkage view supports loading and browsing DEM data with a large amount of data. Select a rectangle area in the DEM linkage view to display it synchronously in the 3D view. Edit DEM in 3D view, the editing results will be saved to DEM files in real time.

#### **Steps**

Click DEM -> Load DEM, open the DEM linkage view.

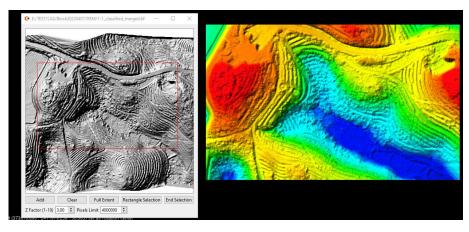


Click "Add" to load DEM data.



 Click "Rectangle Selection" to draw the rectangle in the DEM linkage view, and the corresponding DEM data will be loaded into the 3D view.

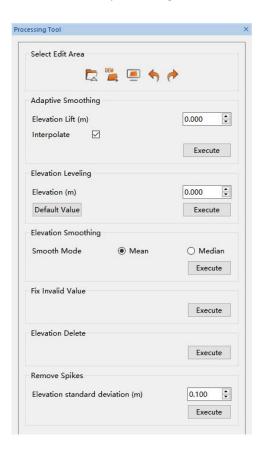




3.3.8.2. Edit DEM

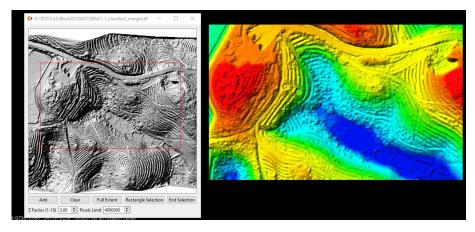
# **Steps**

Click Terrains ->DEM -> Edit DEM, the processing tool will be displayed.

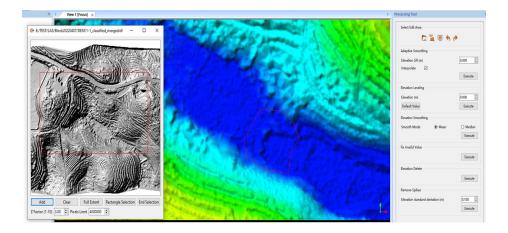


Load DEM data into 3D view through DEM linkage view.





Click "Model Select" / "Screen Select", draw the area needed to be edited in 3D view.



Select a processing mode for edit, include Adaptive Smoothing, Elevation Leveling,
 Elevation Smoothing, Fix Invalid Value, Elevation Delete, Remove Spikes.

#### 3.3.8.2.1. Model Select

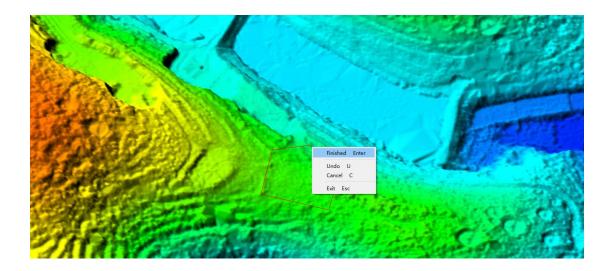
# Description

Select based on model data. The model perspective can be adjusted at will. As the model view is scaled, the selected area of the model is scaled synchronously.

- Click to activate the Import scope Lines. This feature supports the import of dxf vectors as selection areas. Click the import scope line icon, select the dxf data, and click Open to import the selected dxf data into the view.
- Click to activate the Model Select. After activation, left-click to draw polygon on



DEM model, right-click and choose "Finished" button to finish drawing.



3.3.8.2.2. Screen Select

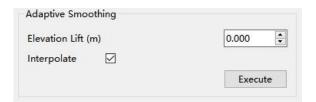
# Description

Select based on screen. It can only be selected in 2D view. The size of the polygon selected on the screen is fixed and does not change as the model is scaled.

# 3.3.8.2.3. Adaptive Smoothing

#### Description

Adaptive smoothing will delete the points inside the drawn polygon, and construct a triangulation network with the points on the polygon as the feature points, and reconstruct the elevation points according to the triangulation network.

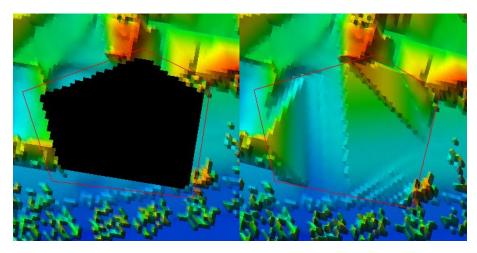


#### **Parameter settings:**

**Elevation lift:** The smoothing result will add the elevation lift.

**Interpolate:** If checked, some feature points will be interpolated at the edge of the polygon to participate in the construction of the network. If not checked, only the vertices of the polygon will participate in the construction of the network.





3.3.8.2.4. Elevation Leveling

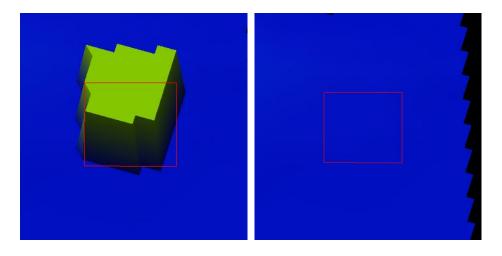
Change the elevation of the slected area to the set value.



# **Parameter settings:**

**Elevation:** The elevation of the slected area will be changed to this value.

**Default Value:** Calculate the average elevation of the selected area.



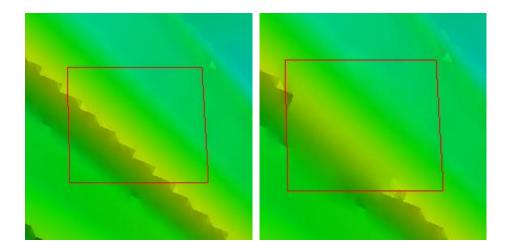
3.3.8.2.5. Elevation Smoothing

# Description

Smooth the elevation of the selected area by using mean smoothing or median smoothing.

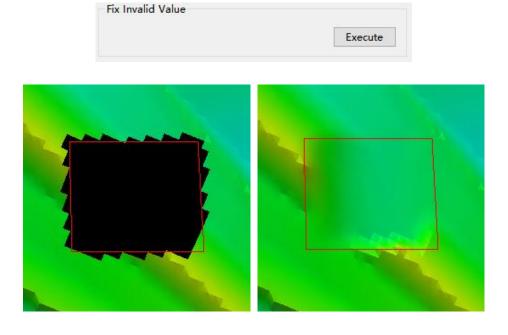






3.3.8.2.6. Fix Invalid Value

Fix invalid value of selected area.

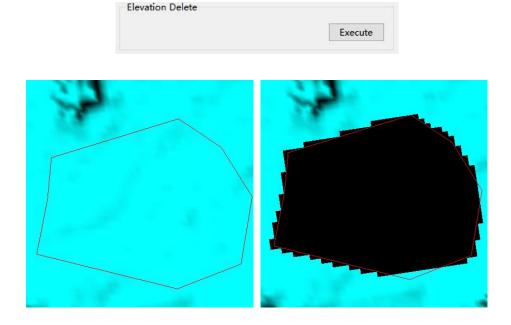


3.3.8.2.7. Elevation Delete

# **Description**

Delete elevation of selected area.





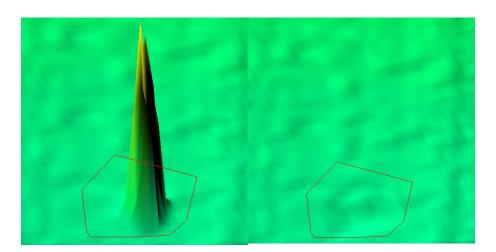
3.3.8.2.8. Remove Spikes

Remove spikes caused by noise points.



# **Parameter settings:**

**Elevation standard deviation:** Calculate the standard deviation of elevation value in the range of point and neighborhood. If the value is greater than the threshold, the point is judged as a noise point.





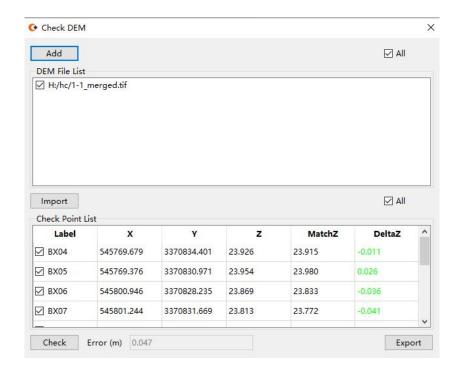
# 3.3.8.3. Check DEM

#### Description

Import DEM and control points, calculate the elevation accuracy of DEM.

#### **Steps**

- Click DEM -> Check DEM.
- Add DEM data and import check points.
- Click "Calculate", the software will automatically calculate the elevation error of DEM.
- Click "Export" to export inspection result in csv file.



3.3.8.4. **DEM Merge** 

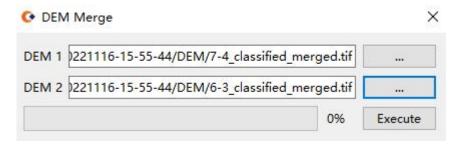
#### **Description**

This feature merges two dems into a new DEM.

If the two DEM resolutions are different, the software will sample the DEM with low resolution and then fuse the two DEM.

If two dems overlap, DEM 2 data will be used for the overlap area. That is, DEM 2 will cover DEM 1 to achieve the effect of partial DEM update.





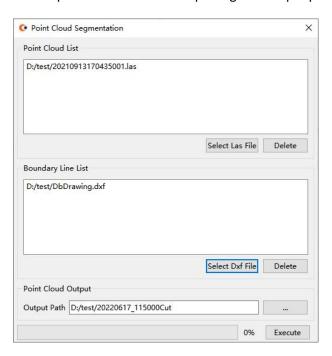
#### **Steps**

- Click terrain Module-> DEM Fusion, and the DEM Fusion dialog box will pop up after clicking.
- Add original and updated DEM files to be merged.
- After clicking execute, perform the DEM fusion operation, and you can view the fusion result through the output path.

# 3.3.8.5. Point Cloud Segmentation

# **Description**

This feature supports the import of .dxf vectors as split regions to split point clouds.



# **Steps**

 Click terrain Module-> Point Cloud Segmentation, and the Point Cloud Segmentation dialog box will pop up after clicking.



- Select the las file and the DXF file to set the output path.
- Click execute to split the point cloud, and you can view the segmentation result through the output path.

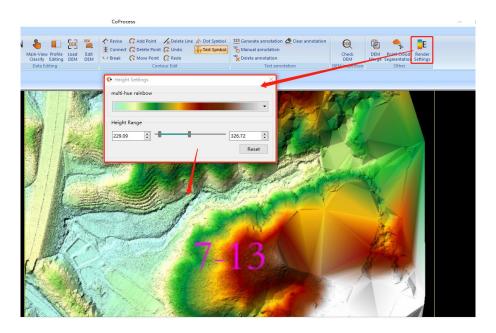
# 3.3.8.6. Rendering Settings

### Description

Set DEM and TIN render separately in the main view.

### **Steps**

- Click Terrains -> Other -> Render Settings, click the render Settings dialog box will pop
   up;
- Select ribbon and elevation range for triangulation rendering
- Click OK.



# **3.4. Roads**

# 3.4.1. Project Manager

# **3.4.1.1. New Project**

# **Description**

Create a new Road project.

# **Steps**

Click Road Module-> Project Manager-> New Project to open the New Project dialog



box;

- Select a path for saving the project and enter the project name;
- Click OK to create the new project.



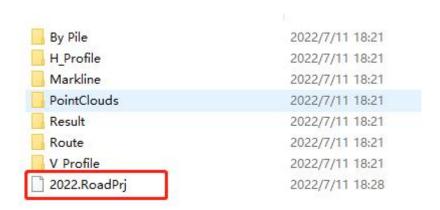
3.4.1.2. Open Project

#### Description

Open an existing road project.

# **Steps**

- Click Road Module-> Project Manager-> Open Project to open the Open Project dialog box;
- Select the RoadPrj file and click Open to load the road project.



3.4.1.3. Save Project

#### Description

After creating or opening a project, click Save Project to save the operations that have been done.

# 3.4.2. Road Parse

#### 3.4.2.1. Broken Chain

# **Description**

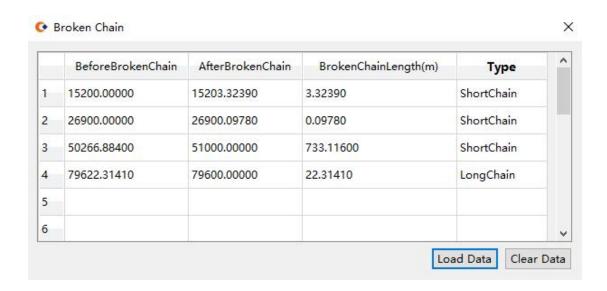


Broken chain refers to the phenomenon that pile numbers are not connected due to partial line change or sectional measurement.

If the broken chain information is recorded in the flat curve, it will be automatically imported when the flat curve data is imported, otherwise it needs to be imported manually.

#### Steps

- Click Road Module-> Road Parse-> Broken Chain to open the Broken Chain dialog box;
- Click Load Data and select Broken Chain data to import. CSV and TXT formats are supported;
- The broken chain data is shown in the figure below.



#### **Notes**

The pile numbers before and after broken chain can be manually modified. The length and type are calculated automatically and do not need to be modified.

#### 3.4.2.2. Horizontal Curve

#### Description

Import horizontal curve data from intersection method or atomistic approach to generate route and pile-by-pile data.

### 3.4.2.2.1. Load Horizontal Curve

#### Description

There are two types of imported flat curves: intersection method and atomistic approach. Including CSV, JD, XLSX, XLS, PM, ROD and other formats.



- Click Road Module-> Road Parse-> Horizontal Curve;
- Select the intersection method or atomistic approach page and click Load Data.
- After the import is completed, you can see the corresponding data in the table.

#### 3.4.2.2.2. Calculate Routes

#### **Description**

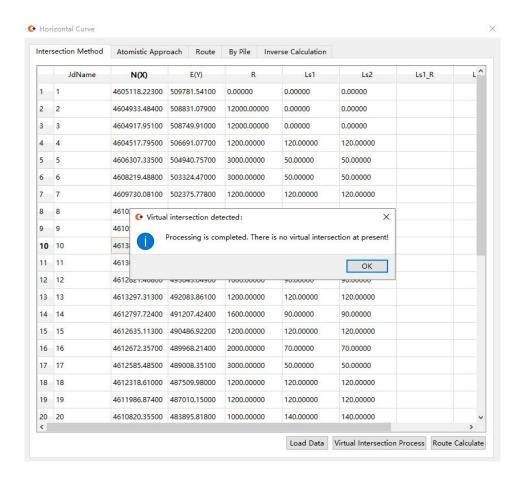
Route calculation can be carried out after route data is imported.

#### Steps

Calculate routes;

#### Intersection method

The intersection method can detect the virtual intersection before calculating the route.



After the virtual intersection processing is completed, click Route Calculation to calculate the route information.

#### **Notes**



If the route used by the intersection method does not start from the starting point, the pile number of the second intersection point of the current route needs to be manually entered in the second line of the starting pile number.

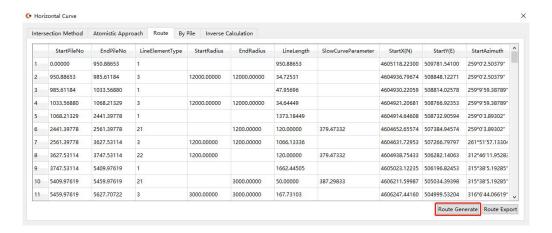
In addition, since the road data before the first straight and slow point cannot be confirmed in the above incomplete curve, it is normal for the pile number and design of pile points on the calculated route to have the following deviation:

- The route in front of the first ZH point is a short straight line, which does not match the designed curve;
- The mileage calculation of the first pile point and the design mileage have a offset of the size of the cut curve difference.

# Atomistic approach

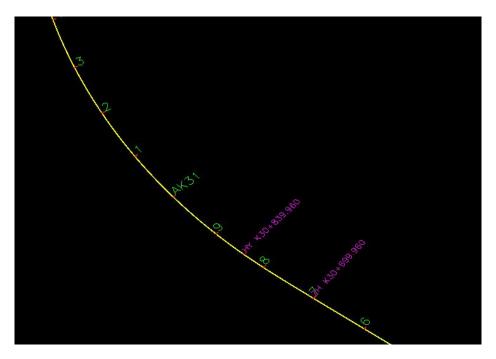
Import the data and click Route Calculate.

After calculating the route, the route page is automatically redirected;



- Click Route Generate to draw the parsed route directly into the view and mount it under the route node of the road project;
- Click Route Export to export the route in the format of DXF.





3.4.2.2.3. By Pile

After route calculation, pile-by-pile operation can be carried out. It includes importing pile number, generating pile number, pile calculating, pile adding and pile exporting.

# **Steps**

There are two ways to get pile numbers: load and create;

# Pile load:

Click Pile Load in the page of By Pile, select TXT or CSV file, and click Open.

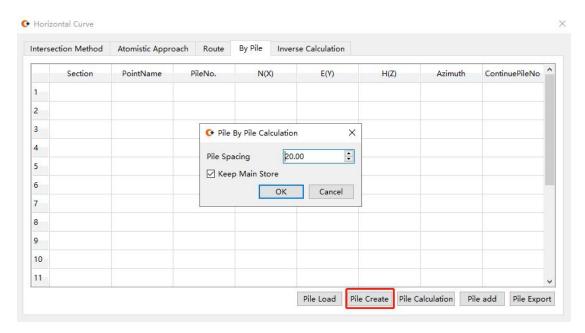
# Pile create:

Click Pile Create in the page of By Pile, after setting the parameters, click OK.

Pile Spacing: Distance between added piles.

Keep Main Store: Whether to retain the original primary point after calculation.

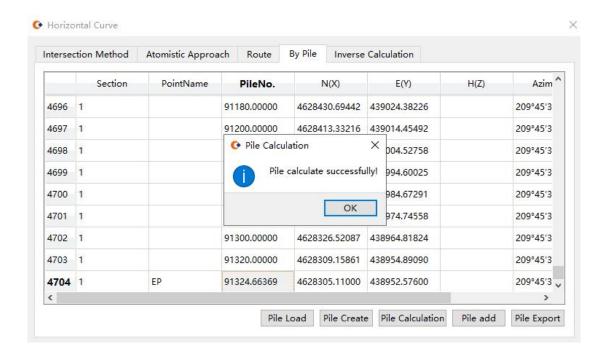




The number information can be seen after the completion of pile-by-pile acquisition.

### 1. Pile calculation;

After obtaining the pile number, click Pile Calculation to automatically calculate the pile-by-pile information.



### 2. Pile add;

Click Pile add and input the pile number to add the corresponding pile.



768	1	14777.00000	4611762. <mark>1</mark> 982 <mark>1</mark>	498553.12503	7°48′22.2
769	1	♠ Add Pile		498553.53250	7°48′22.2
770	1	Add Pile Number		498556.24895	7°48′22.2
771	1	14777.000		498558.96540	7°48′22.2
772	1	ОК	Cancel	498561.68185	7°48′22.2
773	1	14800,00000 2	1011844.42907	498564.39830	7°48'22.2

# 3. Pile export.

Click Pile Export to draw the current pile-by-pile data into the view and mount it under the by pile node of road project.

#### 3.4.2.2.4. Curve Coordinate Calculation

# Description

Calculate the pile information through the input point coordinates **Steps** 

Click the import button on the Inverse calculation page to import the coordinate file
 (CSV, TXT), and click Calculate to calculate the pile information of each point.

#### 3.4.2.3. Markline Transfer

#### Description

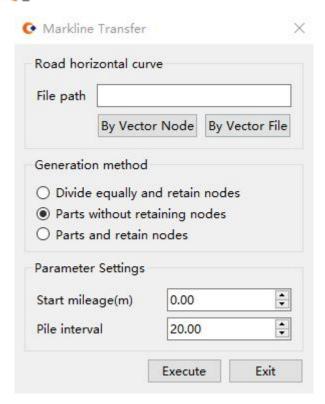
In case of no route file, it can be converted to by pile according to the drawn line. Support loading vector lines from vector nodes or from local files.

#### **Steps**

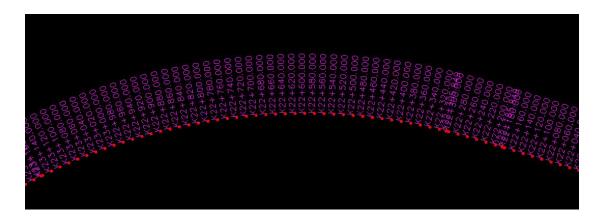
- Click the road module -> Route Parse -> Markline Transfer, select "By Vector File";
- Select DXF vector path;
- Select the generation mode, including the whole parts or divided equally, whether to retain nodes;
- Set the starting distance and pile-by-pile interval, click Execute.

When selecting a vector, if you want to load it from a node, you need to select the labeled node you want to import in your workspace.





Click Execute to generate and display pile-by-pile, as shown below.



**Equal division of the whole section:** when inserting the pile point, the bending of the vector line is not considered, and the pile point is inserted according to the set distance from the starting point.

**Segment and equal division:** when inserting pile points, each broken line is taken as the unit, and inserting pile points are respectively inserted according to the set distance.

Whether to preserve nodes: Whether the nodes of the original vector line are also output as pile points.



# 3.4.2.4. Manual Piling

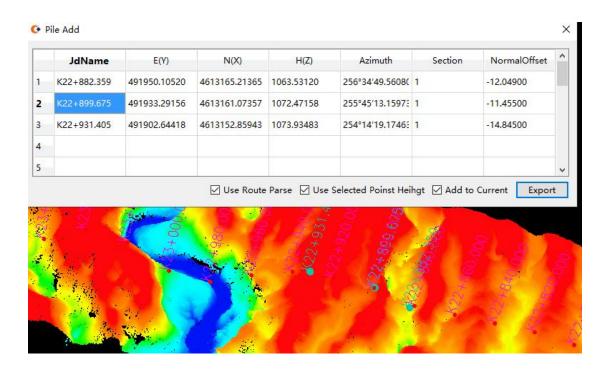
### 3.4.2.4.1. Pile Add

### Description

Add pile-by-pile points according to point cloud view and support loading to current pile-by-pile.

### **Steps**

- Import route point cloud and 2D pile-by-pile file;
- Use "Profile Section -> Route Piles " function to generate 3D pile-by-pile(optional);
- Activate the view piling function, click the left mouse button near the piling line to add piles;
- The plane coordinates of pile points are inversely calculated to pile-by-pile, and the elevation values are consistent with the selected points;
- After piling is finished, click Export to save piling information to the newly generated pile-by-pile.



As shown in the figure, the green points are manual piling points, and the red points are 3D piling points. At the same time, you can delete a row of values by right-clicking in the view



piling table.

#### **Parameter description**

**Section:** Indicates the road segment of the current point.

**Normal Offset:** The positive and negative sign indicates whether the pick point is on the left or right side of the route, and the value indicates the 2D distance between the pick point and pile-by-pile.

**Use Route Parse:** After checking this box, piling will be interpolated according to the resolved route. After deselecting, piling can be added when there is no result of route analysis.

**Use Selected Points Height:** After checked, the elevation value of interpolation point is determined by the elevation value of capture point.

**Add to Current:** After checking, click Export, the result of view piling will be saved in the 3D pile-by-pile that is currently activated.

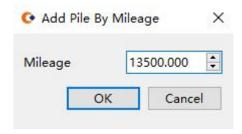
#### 3.4.2.4.2. Mile Add

#### **Description**

Manually input the mileage value, in the input position, add a pile point.

#### **Steps**

- Import route point cloud, use the flat curve function to calculate the route and generate pile by pile;
- Use the "Prifile Section" function to generate 3D pile by pile (optional);
- Activate the mile add function, enter the mileage adding pile, and click OK;



#### **Notes**

Mileage piling must have a calculated route, and the piling results will be directly loaded to the current pile by pile.

### 3.4.3. Road Survey

#### 3.4.3.1. Profile Section

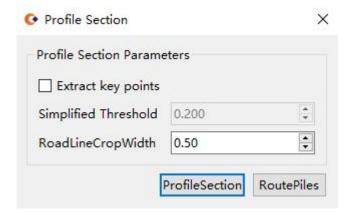
#### Description

Generate profile according to imported point cloud and pile-by-pile, or generate 3D pile-by-pile.

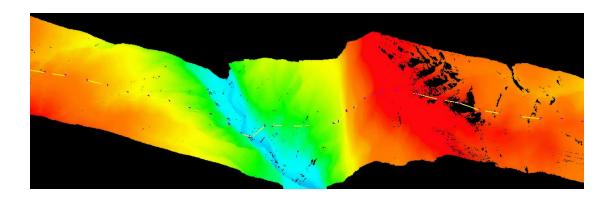


### **Steps**

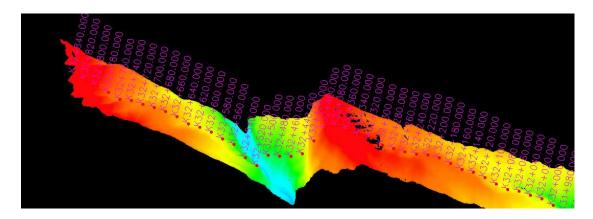
- Import route point cloud and 2D pile-by-pile file;
- Click the road module -> Road Survey -> Profile Section;
- Setting whether the features are simplified, click ProfileSection or RoutePiles.



RoutePiles means generating 3D pile-by-pile.



As shown in the figure above, purple dots and yellow tracks represent the resulting longitudinal section on the point cloud.





#### **Parameter Settings:**

**Simplified threshold:** the feature pile points with the difference between the height before and after and less than this value will be deleted. This feature must have a calculated route to use.

**Clipping width:** the left and right sides of the route participate in the calculation of the number of point clouds when the longitudinal section is extracted.

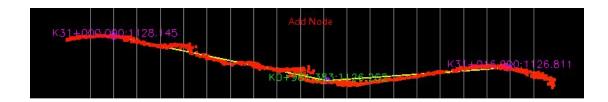
#### 3.4.3.2. Profile Editing

## Description

Edit the automatically generated profile and support saving the edit results to pile by pile.

#### Steps

- After the profile is generated, click the Profile Edit function to open the profile view.
- By default, the section between the first two pile points is opened, and the main view will switch to the corresponding section.



As shown in the above figure, the section view shows the pile number and corresponding elevation of the two pile points before and after, the section line between the two points and the vertical grid cable. The grid lines are spaced 1m apart by default. The original pile points are shown in red, and the new pile points are shown in green.

# Skip to the previous section

Skip to the previous section of the current display section.

# Skip to the currect section

If multiple profiles are currently loaded, select one of them and click " Skip to the currect section " to display the selected profile in the profile edit view.

## Skip to the next section

Skip to the next section of the current display section.

# Add Node

Click "Add Node" or shortcut key A, click on A point on the vector line, and add A node at that point.

When you enter section editing, the add node function is enabled by default.



# C Delete Node

Click "Delete Node" or shortcut key D, click a node on the vector line to delete the selected node.

## Move Node

Click "Move Node" or shortcut key M, click and drag a node on the vector line to change the position of the node. If you click on the vector line where there are no nodes, a new node will be added and the move command will be executed.

# **Undo**

Click Undo or shortcut key U to undo the previous edit of the cross section.

# Redo

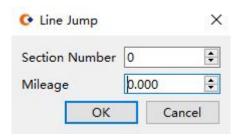
Click Redo or shortcut key R to roll back the previous undo action for cross-sectional editing.

# Slice Settings

Change the thickness, length and grid interval of the point cloud in the view.

# Jump

Enter the road segment and mileage values to jump the section view to the location of the target mileage.



# Relock

Restore the section view to the preview state when browsing.

You can use this feature to restore the profile view if its rotation lock state is released during browsing.



Hide/display the grid line.

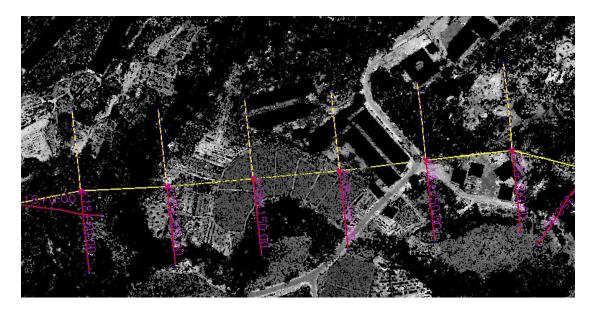
#### 3.4.3.3. Cross Section

#### Description

Generate the cross section and vertical section of the road according to the preview section, or directly generate the section without preview.



 After there is no problem in section preview, click Generate to extract and calculate section feature points.



#### **Parameter description**

#### **Crop Param:**

**Section Point Clipping Width:** The extent of the point cloud that participates in the construction of the network when generating the section points on the cross-section. If the setting is too small, there may be location extraction failures; If the setting is too large, the extraction efficiency will be lower.

Clip width of the middle pile point: If the current pile does not have elevation information, the point cloud extent participating in the network construction when calculating the elevation of the middle pile point of the cross-section. If the setting is too small, the extraction may leak piles; If the setting is too large, the extraction efficiency will be lower.

#### **Generation method:**

**Overwrite the original section:** When you regenerate a cross-section, the original section is cleared and regenerated.

**Generate Modify Pile Section:** When you regenerate sections, only the currently new stations with elevation changes are generated.

**Length Limit:** the maximum triangulation net side length when constructing TIN.

**Clipping Distance:** the distance between two adjacent points in the cross section when extracting feature points.

**Section Rotation:** the clockwise or anti-clockwise section rotation function requires loading section data.

**Topographic Features:** Sample by terrain features, and extract terrain feature points by considering equal intervals and elevation differences.

**Clipping Distance**: the length of the left and right sides of the resulting longitudinal section.

#### Note

The rotate section clockwise or counterclockwise feature requires the section data to be



loaded.

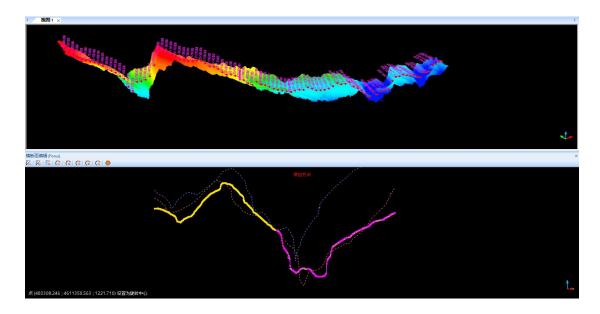
### 3.4.3.4. Section Editing

#### Description

Correct the deviation between the cross section and the point cloud by vector editing in the cross section view.

#### Steps

- Click Section Edit to open section view. By default, open the cross section view of the first pile number.
- The section view of the selected section can be viewed to realize the up-down switching of the section view and the setting of section parameters.
- As shown in the section view below, the red and yellow vector lines are the current section lines that can be edited.
- The orange dotted line is the next section line, the blue dotted line is the previous section line, and the orange and blue dotted lines are not editable.

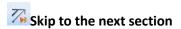


# Skip to the previous section

Skip to the previous section of the current display section.

# Skip to the currect section

View the currently selected section.





Skip to the next section of the current display section.

## Add Node

Click "Add Node" or shortcut key A, click on A point on the vector line, and add A node at that point.

When you enter section editing, the add node function is enabled by default.

## Delete Node

Click "Delete Node" or shortcut key D, click a node on the vector line to delete the selected node.

# **Move Node**

Click "Move Node" or shortcut key M, click and drag a node on the vector line to change the position of the node. If you click on the vector line where there are no nodes, a new node will be added and the move command will be executed.

# **Undo**

Click Undo or shortcut key U to undo the previous edit of the cross section.

# C Redo

Click Redo or shortcut key R to roll back the previous undo action for cross-sectional editing.

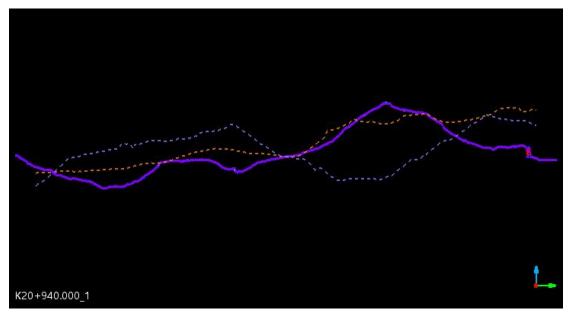
# Slice Settings

Modify the thickness and length of the section in section view.

#### Step:

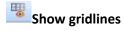
- Click the section parameter setting button above the section view to modify the section thickness or length, and click Apply. Then the section thickness or length in the section view can be updated and displayed in real time.
- If the function of previewing front and back sections is selected, the track of front and back sections will be shown in orange and blue in the section view, respectively.





# Show Height

Displays the current elevation at the cross section point.



Display grid lines in section view.

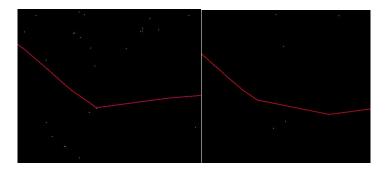
### **3.4.3.5. Delete Node**

### Description

Delete the node of the section on the main view.

#### **Steps**

- Generate cross section;
- Click Road Module ->Road Survey ->Delete Node to activate the point deletion function;
- Click the node on the cross section in the main view to delete it.



#### **Notes**

At present, the point deletion function can only be used to delete points on the cross section,



and does not support the deletion of center stake points.

#### 3.4.3.6. Section Display

#### **Steps**

- After the cross section is generated, click Hide All to realize the hiding of all sections;
   Click Show All to display all hidden sections;
- Select multiple sections and click Show Selected sections to display specific sections.
   Click Hide Selected to realize the hidden selected section;
- Click Clear All to clear all section views and section nodes.

#### 3.4.3.7. Section Results

#### Steps

- Click the road module -> Road Survey -> Section Results -> Export Profile/Cross;
- Set the file name, select the export file format, and click Save;
- Select "Export all sections" or "Export only selected sections", Export all sections is selected by default;
- After setting, click Export to save profile/cross section results successfully.

Middle pile mileage + middle pile absolute elevation(\*.DMX)

Middle pile mileage + middle pile absolute elevation(\*.DMX)

3D coordinates of profile(\*.dat)

Piles(\*.csv)

Profile line(\*.dxf)

## Lifting method - distance to adjacent point + height difference(\*.HDM)

Lifting method - distance to adjacent point + height difference(\*.HDM)
Lifting method - distance to adjacent point + height difference(\*.HDX)
Distance to middle pile + absolute elevation\_ABS(\*.HDX)
Distance to middle pile + height difference with middle pile\_ABS1(\*.HDX)
3D coordinates of cross section with pile number(\*.dat)
3D coordinates of cross section(\*.dat)
CASS(\*.dat)
(\*.dxf)

Distance to middle pile + absolute elevation(\*.dxf)

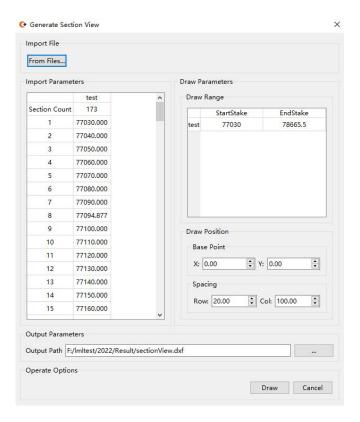
BaiTu(\*.TXT)



### 3.4.3.8. Generate Section View

### Description

Generate the DXF format sectional view from the HDM file



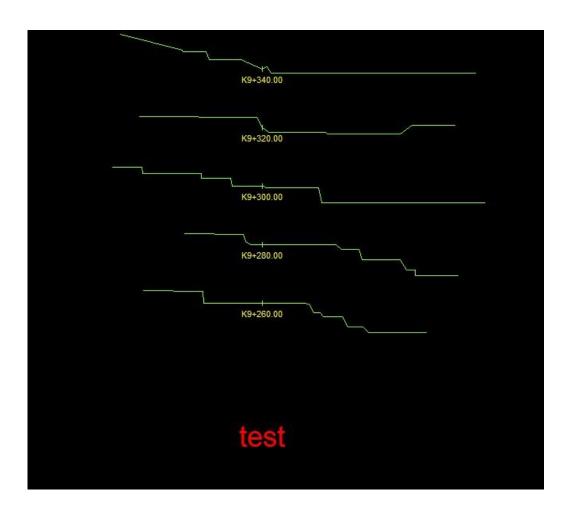
- Click the function of generating sectional view;
- Click Import file and select HDM file;



- Modify the drawing base point and spacing, right click on a section can set its pile number as the starting pile number or the end pile number;
- Set the output path;



- Click "Cross section Design Drawing" after setting.
- After drawing, you can find the drawn DXF file in the output path.



## 3.4.4. Other Tools

### 3.4.4.1. Manual Extract

### Description

Manually draw a line on the point cloud.

- Click Other Tools > Manual Extract;
- Click the left key on the point cloud for many times to draw multi-segment lines;
- After drawing, right-click and choose Generate Markline or Interpolation to generate mark line.



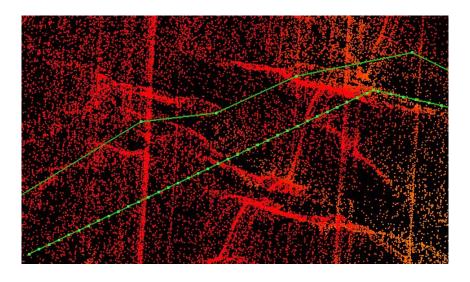


#### **Generate Markline**

Generate lines directly from manually drawn nodes.

## Interpolate to generate markline

Interpolate the result line at 1 meter intervals.



3.4.4.2. Vector Editing

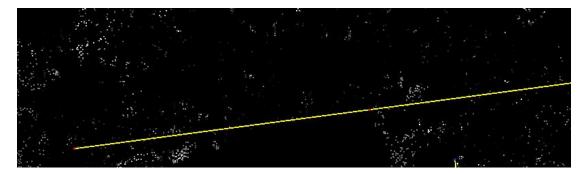
#### Description

Add, delete, or shift points to manually extracted or imported marks.

### 3.4.4.2.1. Add Node

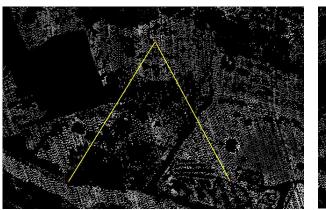
Click the road module -> Road Survey -> Other Tools -> Add Node, or use the shortcut key A in the edit function, move the mouse to the position of adding node, and click the left button to add a point on the vector line.

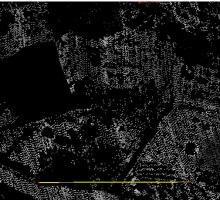




3.4.4.2.2. Delete Node

Click the road module -> Road Survey -> Other Tools -> Delete Node, or use the shortcut key D in the edit function, move the mouse to the node to be deleted, and click the left button to delete the point on the vector line.





3.4.4.2.3. Move Node

Click the road module -> Road Survey -> Other Tools -> Move Node, or use the shortcut key M to drag the mouse from a point to move a point on the vector line.

If you move a point from the original point, a new point is added, and then the new point is moved.

## 3.4.4.3. Markline Sample

#### 3.4.4.3.1. Baseline

#### Description

Reference sampling of a line according to a fixed step size.

- Select a line under the line node;
- Click the road module -> Other Tools -> Markline Sample -> Baseline;



Click Start sampling to generate a line with a fixed sampling interval.

If you want to retain the original midpoint of the line and delete the original line, select the corresponding option;

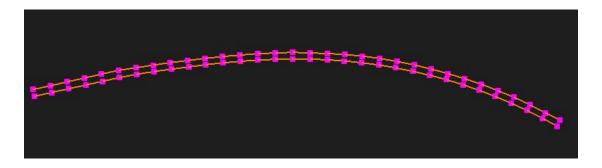
It is recommended to select the relatively complete line with more points as the benchmark sampling line.

#### 3.4.4.3.2. Parallel

#### Description

The line sampled by datum is used as the reference line for parallel sampling of other routes. **Steps** 

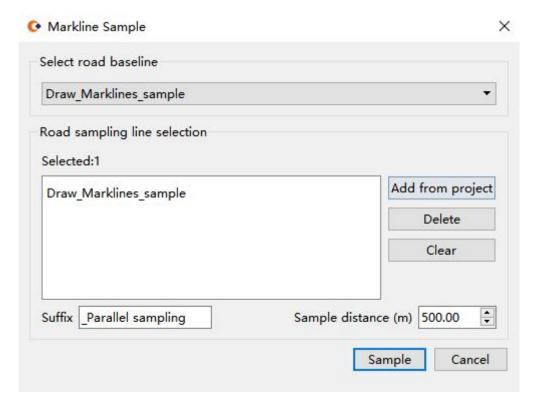
- Click the road module -> Other Tools -> Markline Sampling -> Parallel;
- Select the newly generated line after datum sampling as the datum line;
- Select the line to be sampled under the line node in the left road project;
- Click "Add from Project" to load the sampling line directory and click Sampling.



#### Sampling distance threshold:

There is a reference line and a sampling line. The reference line has a point on it that is perpendicular to the sampling line. To determine if the reference point will be sampled, we need to calculate the sum of the distance between the vertical and the left and right points on the sampling line. If this sum is greater than the set value, then the reference point will not be sampled.





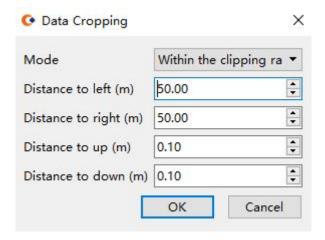
3.4.4.4. Data Cropping

### Description

According to the imported pile-by-pile file, the point cloud is cut along the left, right, up and down sides of the pile line.

- Right-click on one of the pile-by-pile nodes and select "Set Current";
- Select a point cloud under the point cloud node and click the Roads module -> Other
   Tools -> Data Cropping;
- Set the cutting type and the left and right distance from the pile line, and click OK.



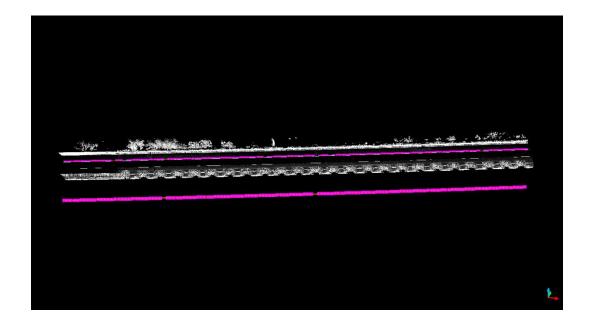


3.4.4.5. **3D-interpolation** 

### Description

The 2D vector lines are converted into three-dimensional vector lines by 3D -interpolation and attached to the point cloud.

- Import 2D DAT or DXF vector line;
- Click the road module -> Others -> 3D-Interpolation;
- Select the point cloud filtering in the interpolation process;
- Click OK to view the 3d interpolation effect as shown in the figure below after the progress processing is completed.





## 3.5. Volume

## 3.5.1. Project Manager

### **3.5.1.1. New Project**

#### Description

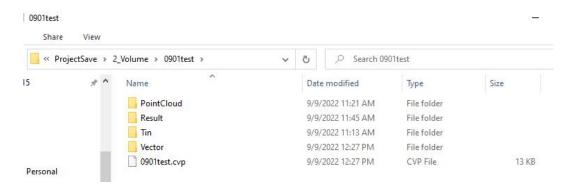
Create a new Volume project.

#### **Steps**

 Click Volume -> Project Manager -> New Project, select the project path, enter the project name, and click OK to create the project;



Project directory generates Cvp project file, PointCloud, TIN, Vector and Result folders.



3.5.1.2. Open Project

#### Description

Open an Volume project.

#### **Steps**

- Click Volume -> Project Manager -> Open Project;
- Select the project path, Select CRS project file and click Open to load intelligent
   Extraction project.

#### 3.5.1.3. Save Project

#### Description



After creating or opening a project, click Save Project to save the existing operations.

## 3.5.2. Point Cloud Processing

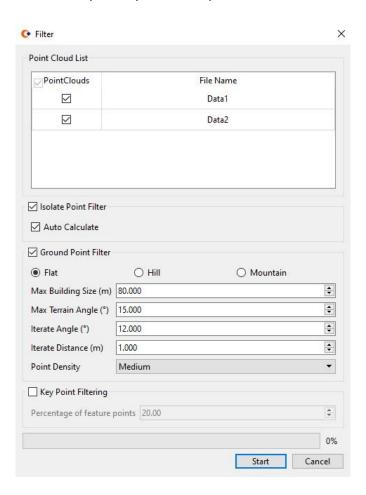
## 3.5.2.1. **Filtering**

#### Description

The filtering function can quickly separate the ground points from the input point cloud, which is convenient for the subsequent TIN construction and volume calculation.

#### **Steps**

- Add Las point cloud to be filtered in the project;
- Click Volume ->Point Cloud Processing ->Filtering, open the filter dialog box;
- Select the filtering point cloud in the point cloud list;
- Select the filtering to be performed and set the filtering parameters.
- Click Start to automatically classify the added point cloud.



**Filtering method Description** 



#### **Isolate Point Filter:**

The points in the point cloud data whose number of points in a certain neighborhood is less than the specified value are identified as outliers. This function is mainly used to remove outliers in the air or below the ground. The outliers are far away from the main point cloud and have the characteristics of few points and discrete distribution.

#### **Ground point Filter:**

It is used to classify ground points and non-ground points, supporting different terrain such as flat terrain, hill terrain and mountainous terrain.

#### **Key Point Filter:**

According to the set percentage of feature points, some special points are extracted from the ground points.

#### **Parameter Settings:**

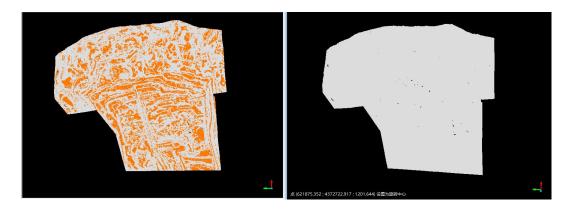
**Maxi building size:** This parameter controls the size of the progressive window of the morphological filtering algorithm and the number of iterations. Generally refers to the length of the diagonal, that is, the longest distance of the hypotenuse of the building.

Maxi Terrain Angle: the section with the steepest slope is selected as the max terrain Angle.

**Iteration Angle:** Three vertices from a point to a triangulation constitute the maximum Angle. The larger the iteration Angle is, the farther the points are from the triangulation network. In the process of iteration, the points that are far away and meet the iteration Angle will be added to the ground points, which should not be set too large.

**Iteration Distance:** vertical distance from a point to the plane of the triangulation network. Point density: the degree of dilution of the filtering result. General operations do not require very dense ground points.

Percentage of feature points: How many of the ground points are classified as key Points.



3.5.2.2. Flatten

#### Description

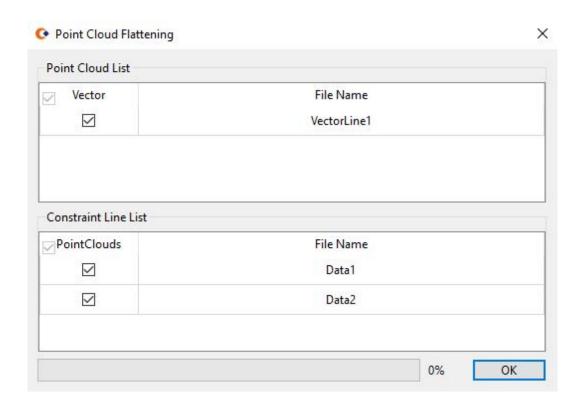
According to the selected vector line, the point cloud within the range is fitted and flattened according to the boundary of the vector line. It can be used to clear ground equipment and repair missed test areas.

#### **Steps**

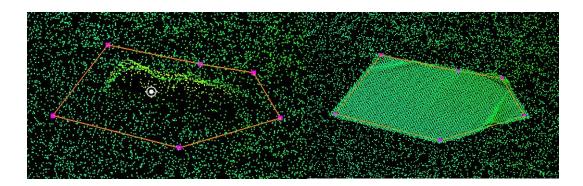
Load the point cloud to be flattened;



- Use vector line drawing or import function to obtain the boundary vector line of the flattened area;
- Click Volume -> Point Cloud Processing -> Flatten, open the flatten dialog box;



- 4. Select the processing range vector line and processing point cloud;
- 5. Click OK. After the progress bar is executed, a point cloud with the suffix "\_Flatten" will be generated under the point cloud node of the square project.



3.5.2.3. Export Ground Points

#### Description

Export the ground points in the selected point cloud to the project results folder.



#### **Steps**

- Load the point cloud to generate triangulation network in the project;
- Use the filtering function to filter the point cloud.
- Click Volume module -> Point Cloud Processing -> Export Ground Points;
- Select the appropriate point cloud and click Start.
- After the export is complete, the software outputs the points and paths of the ground points

[2022-10-27 21:17:02] Filtering completed!
[2022-10-27 21:17:08] F:/ProjectSave/2\_Volume/0901test/Result/ground/Data1\_ground.las The number of output points is 437638
[2022-10-27 21:17:08] Export complete

## 3.5.3. Triangulation

#### 3.5.3.1. Build TIN

#### 3.5.3.1.1. Direct

#### Description

According to the point cloud information, the triangulation network of a specific category of points is constructed.

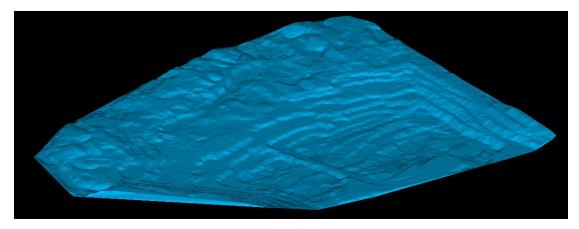
## Steps

- Load the point cloud to generate triangulation network in the project;
- Click Volume module-> Triangulation -> TIN;
- Select the point cloud to generate triangulation network;
- Select the type of point;
- Click OK.

#### **Parameter Settings:**

**Points Type:** Controls the type of points that participate in network construction. You can select all points, or filtered ground points and other types of points.





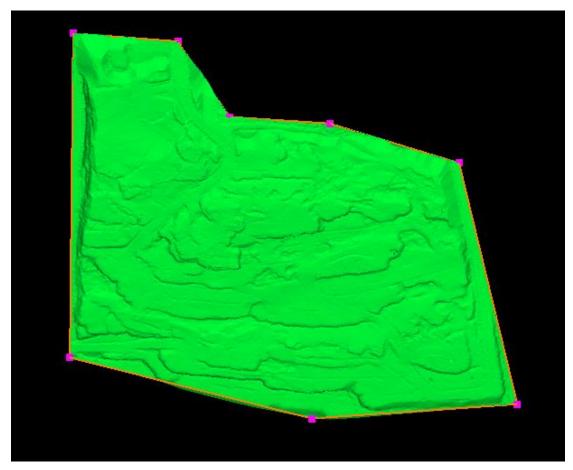
3.5.3.2. Boundary Constrain

### Description

According to the point cloud information, the triangulation network of a specific category of points is constructed.

- Load the point cloud to generate triangulation network in the project;
- Click Volume module-> Triangulation -> TIN;
- Select the point cloud you want to participate in generating.
- Select Add DXF in Parameter Settings;
- Adjust the mesh type and boundary smoothing method and check the vector lines involved in the mesh construction;
- Click OK to generate the result of boundary constraint network.





#### **Parameter Settings:**

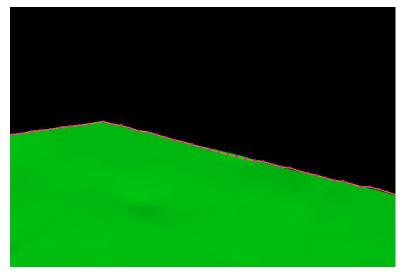
**TIN Type:** When selecting the constraint network, the selected vector line will constrain the range of triangulation network generation; When selecting to participate in network construction, the selected vector lines will participate in the generation of triangulation network.

**Point Type:** Controls the type of points that participate in network construction. You can select all points, or filtered ground points and other types of points.

**Boundary smoothing:** When selecting boundary constraints to construct the network, the smoothing method of the generated triangulated network boundary area can be adjusted, including low smoothing, moderate smoothing, high smoothing and direct use of the original position (no smoothing).

Among them, after the boundary smoothing function is enabled, the constraint line will be sampled first when constructing the net, and the sampling points and the points within the constraint range will be constructed together. The higher the smoothness, the more sampling points participate in network construction. The effect of boundary smoothing is shown in the figure below:





## 3.5.4. Single Volume Calculation

## 3.5.4.1. Projected Volume

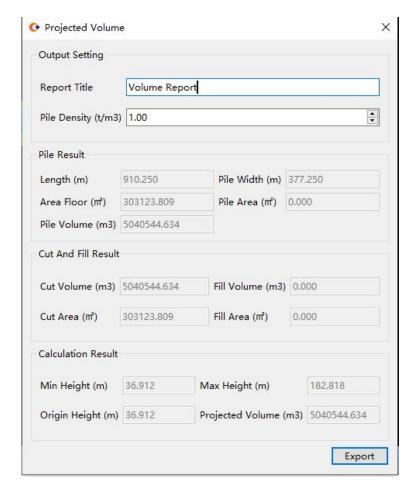
#### Description

Calculates the projected volume of the triangulation network against the current reference surface

It can get the maximum and minimum elevation, starting elevation, projection volume, excavation/filling volume, occupied area and other results, and also support one-key output volume report.

- Create a TIN using the TIN function;
- Define a reference surface using the reference surface Settings function;
- Click Volume -> Single volume Calculation -> Projected Volume;
- Select the calculation triangulation network in the pop-up dialog box;
- Click OK to set the heap density and report title in the second page.
- Click Export to obtain the projection volume report results.





3.5.4.2. **Grid Volume** 

#### Description

The point cloud within the range of vector line is divided into several grid grids according to the set grid edge length, and then the fill volume in each grid is calculated.

All figures in this section are in meters, square meters, or cubic meters.

- Load the point cloud and constraint vector to be compared and analyzed, where the vector can be drawn by itself or imported from the outside;
- Click Volume -> Single Volume Calculation -> Grid Volume;
- Select point cloud and constraint vector line data in the calculation area in the selection box;
- Adjust the calculation parameters, select the calculation method and projection reference surface;



- Adjust the grid length;
- Click OK to complete the calculation.

After the calculation is completed, the calculation result of a single phase grid will appear under the "calculation result" node in volume project. As shown in the picture below:



In the figure, T represents the amount of filling and W represents the amount of digging. The yellow number near each grid node represents the elevation of the corresponding point cloud of the grid dot, the blue number represents the elevation of the reference surface, and the pink number represents the difference between the two.

#### **Parameter Settings:**

**Traditional Square Grid method:** When selecting the traditional grid method, the software will calculate the column volume in each square.

**Triangulation method:** When choosing triangulation network method, the software will calculate the volume of triangulation network projected to the reference surface in each grid, and the calculation result is more accurate.

#### **Reference plane Settings:**

**Model Lowest Point:** When this item is selected, the elevation value of the calculated reference surface will be set to the elevation of the lowest point in the point cloud model.

**Target Elevation:** When selected, the elevation of the calculated reference surface is set by the user.

**Reference plane:** If this item is selected, the current reference plane will be used for calculation. If there is no reference plane, the lowest point of the model will be used by default.

#### 3.5.4.3. Closed Volume

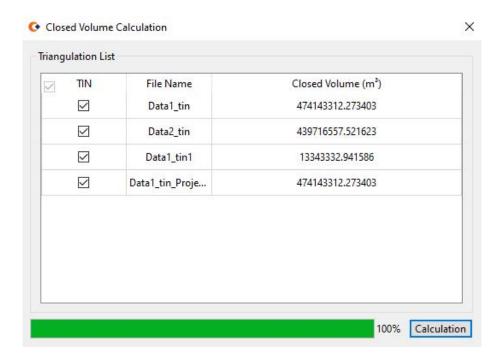
#### Description

Calculate the volume of the closed triangulation generated by the projection of the triangulation onto the reference surface.



#### **Steps**

- Create a projection triangulation using the projection to reference surface function.
- The hole filling function is used to repair the hole of the projection triangulation network.
- Click Volume -> Single Volume Calculation -> Closed Volume;
- Select the projection triangulation you want to calculate and click OK.



The calculated closed volume is shown in the third column of the table corresponding to the triangulation.

#### 3.5.5. Multi Volume Calculation

### 3.5.5.1. Volume Comparison

#### Description

The triangulation data and constraint vector lines of the two-phase project are used to calculate the amount of excavation square and the elevation before and after excavation.

## Steps

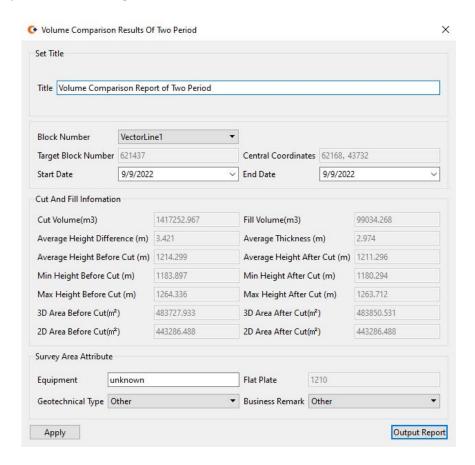
 Use the network function to generate triangulation network of two phases in the engineering area;



- Click Volume->Multi Volume Calculation -> Volume comparison;
- In the selection box, select the triangulation data of the two-phase project and the constraint vector line data of the contrast area;
- Click calculation to generate the volume comparison results of the two phases;
- On the second page, set the report title and test area properties, and click Calculation.
- Click Generate report to complete the comparison work and output the two-phase volume comparison report.

#### **Notes**

must import constraint range vector lines.



3.5.5.2. Grid Comparison

#### Description

The square grid calculation method is used to compare the filling and digging amount of point clouds in two stages.



- Load the point cloud and constraint vector to be compared and analyzed, where the vector can be drawn by itself or imported from the outside;
- Click Volume->Multi Volume Calculation -> Grid comparison;
- The point cloud data of the first phase and the second phase were selected respectively,
   and the constraint region vector range line was selected.
- Set the edge length of the grid and click Calculate.



Similar to the function of grid volume, the calculation results are saved under the "Calculation Results" node of the project. The amount of filling and digging in each square is represented by T and W.

In the above picture, "T" represents the filling amount and "W" represents the digging amount; The yellow number near each grid node represents the elevation of the former phase point cloud, the blue number represents the elevation of the later phase point cloud, and the pink number represents the difference between the two phases.

### 3.5.5.3. Change Analysis

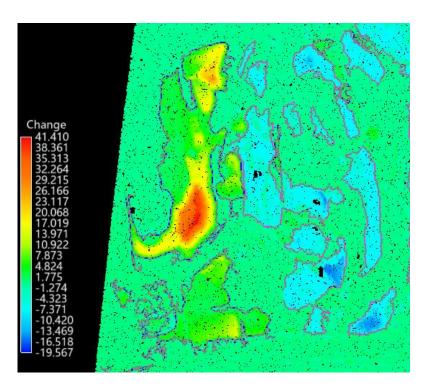
#### Description

By comparing the point cloud data of the two periods, the change of earthwork in the construction area is monitored and the bulk volume of the reactor is roughly calculated.

- Load the point cloud and constraint vector to be compared and analyzed, where the vector can be drawn by itself or imported from the outside;
- Click Volume->Multi Volume Calculation ->Change Analysis;



- Select the point cloud data of the first phase and the second phase respectively and load them into the dialog box.
- If boundary constraints are required, check the "Add constrained region" option and select the corresponding vector in the list.
- Click OK to automatically calculate the difference between the point cloud data of the two periods and generate a diff map.
- Click the "change area detection" button to extract the change track line of the change area;
- Click the "Report Export" button to generate a change analysis report with one click.



The diff map generated by the software is shown in the figure above. The blue vector line is the fill line and the pink is the cut line.

## Parameter setting:

**Original Method:** When selecting this method, compare the point cloud directly. The change value is obtained by subtracting the elevation value of the first phase point cloud from the elevation value of the second phase point cloud data. big

Use this method for some scenarios.

Delete Missing Areas: When selecting this method, compare the point cloud directly.



Similarly, the elevation value of point cloud data in the second phase is subtracted from the elevation of point cloud in the first phase, but the point cloud region that does not repeat will not participate in the calculation of heat map.

This method is suitable for little change, or part of the area missed scanning, using the first method results are not good.

**Complete Missing Areas:** When selecting this method, the software will first build a triangulation network against the point cloud, and then compare the triangulation network. In this way, we can use the method of network to complete part of the missing sweep area. This method is suitable for computing scenarios where partial missing regions need to be fitted.

#### 3.5.5.4. Diff Colorzation

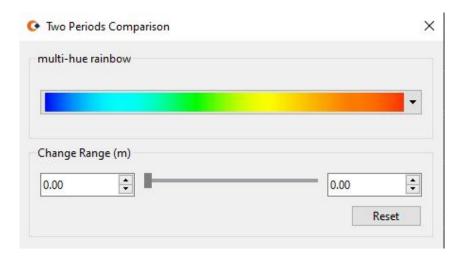
#### Description

Adjust the rendering mode of the thermal map generated by the change analysis for easy browsing.

#### **Steps**

- After generating a thermal map using the change analysis function, click on the Volume
   ->Multi Volume Calculation ->Change Analysis;
- Set the ribbon and range in the Settings dialog box, the rendering mode of the thermal map point cloud in the view will be changed, and the points that are not in the range will be rendered white.

The range of variation is in meters.



## 3.5.6. Vector Achievements

3.5.6.1. **Bench Lines** 

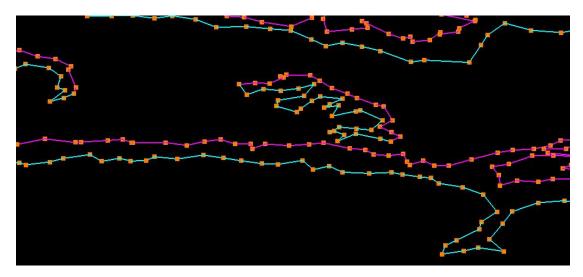
Description



Can quickly extract the bench crest and bench toe, and vector display.

#### **Steps**

- The triangulation network of the engineering area is generated using the functions of network construction and clipping.
- Click Volume ->Vector Achievements-> Bench Lines;
- Select the corresponding triangulation network in the triangulation network list;
- Set the slope Angle threshold and smoothness threshold;
- Click Execute to extract the slope top and bottom line, and give the quantity statistics of the two lines.



#### **Parameter Settings:**

**Slope Angle threshold:** controls the slope extraction, and places where the relief Angle is greater than this value will be considered as profiles. The default 15 °.

**Smoothness threshold:** controls the extraction of the vector line. If the Angle between two segments of the vector line is greater than the set value, the vector line will not be extracted.

## 3.5.7. Vector Line Processing

3.5.7.1. Draw

## Description

Draw DXF vector lines by clicking several points on the point cloud or triangulation.

#### **Steps**

Import the point cloud of the project area;



- Click Volume-> Vector Line Processing -> Draw, activate the draw function;
- Click a few points on the point cloud to draw a vector line;
- After the drawing is finished, right-click and choose "Closed" or "Finish" to complete the drawing of vector line.

#### **Notes**

If you want to get a closed vector line, select "Closed", otherwise select "Finish".

3.5.7.2. Import

#### Description

Import a DXF vector line in the specified path into the project.

#### Steps

- Click Volume-> Vector Line Processing -> Import;
- In the dialog box that pops up, select the DXF file you want to import and click OK.
- If the DXF file is correctly formatted, it will be imported into the project and displayed in the view.

### 3.5.7.3. **Export**

### 3.5.7.3.1. Export Vector Line

## Description

Export the vector line under the node to local.

## **Steps**

- Select the vector you want to export,
- Click Volume-> Vector Line Processing -> Export Vector Line;
- Select a save path, Click Save to write the DXF vector line to the local.

### 3.5.7.3.2. Export Elevation Points

### Description

Only nodes in vector lines are exported.



- Select the vector you want to export,
- Click Volume-> Vector Line Processing -> Export Vector Line;
- Select a save path, Click Save to write the DAT points to the local.

#### 3.5.7.4. Vector Editing

#### Description

Implement node selection, delete, add, move, undo and redo.

#### 3.5.7.4.1. Select Line

Click Volume -> Vector Line Processing -> Vector Editing -> Select Line to begin selection. Support click, rectangular box selection.

#### Click

Click a vector line to select; Hold down the Ctrl key and click multiple vector lines in turn to select multiple vector lines. The selected vector lines are shown in green.

#### **Rectangular box selection**

Hold down Ctrl and drag the left mouse button to drag a rectangular area.

If selected from the upper left corner to the lower right corner, the vector lines that are fully contained in the region are selected, otherwise the vector lines that intersect the rectangular box are selected.

#### **Notes**

When vector lines are selected, the vector lines that have already been selected are deselected by selecting the vector lines again via the rectangular box selection.

#### 3.5.7.4.2. Add Node

Click Volume -> Vector Line Processing -> Vector Editing -> Add Node, move the mouse pointer to the position where the node needs to be added, and click the left button when the "x" shaped symbol appears.

### 3.5.7.4.3. Delete Node

Click Volume -> Vector Line Processing -> Vector Editing -> Delete Node, move the mouse pointer to the node and click the left button to delete it.

#### 3.5.7.4.4. Move Node

Click Volume -> Vector Line Processing -> Vector Editing -> Move Node, select the point to move with the left key and drag the mouse to move the selected point to the specified position.



#### 3.5.7.4.5. Break

Click Volume -> Vector Line Processing -> Vector Editing -> Break, Click on a vector line in the view to break it into two.



3.5.7.4.6. Merge

Click Volume -> Vector Line Processing -> Vector Editing -> Merge, Click one vector line in the view, and then the other to merge the two vector lines into one.



3.5.7.4.7. Undo

After the edit point, click Undo to undo the last operation.

#### 3.5.7.4.8. Redo

After undoing the last edit, click Redo to restore the state before undoing.

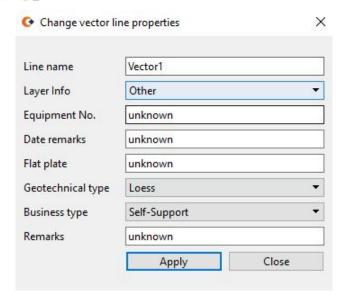
#### 3.5.7.4.9. Change Vector Attributes

#### Description

Right-click the vector line node to modify the properties of the vector line. Support manual modification of line name, layer information (bench crest, bench toe), and other personalized information, and support with the project.

- Draw or import vector lines;
- Right-click the vector line node and click the Change Properties function
- Modify the information in the dialog box that is displayed
- Click Apply to save the modified property information.



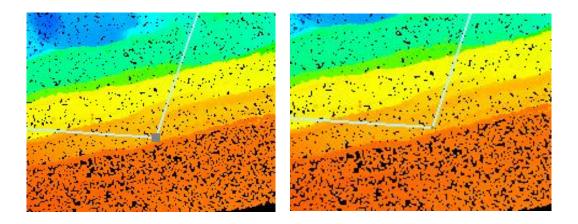


3.5.7.5. **Dot Symbol** 

### Description

Change the display size of the vertices in the vector line for easy browsing. **Steps** 

- Draw or import vector line objects;
- The dot symbol button is activated by default, and the vertices on the vector lines appear as distinct gray squares.
- Click Volume-> Vector Line Processing->Dot symbol;
- The vertex display becomes smaller.



3.5.7.6. **Sampling** 

#### Description

The selected vector line is sampled at the set sampling interval and fitted to the point cloud



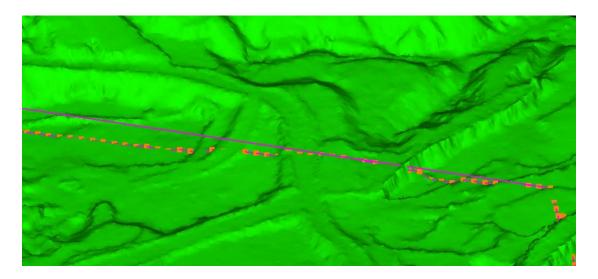
model.

## **Steps**

- Import the point cloud and sampling line in the working area;
- Click Volume-> Vector Line Processing -> Sampling;
- Set sampling interval;
- Click OK to generate the sampling vector.

#### **Parameter Settings:**

Sampling interval: Refers to the sampling interval and the distance between sampling points.



### 3.5.8. Reference Plane

## 3.5.8.1. Project to Reference Plane

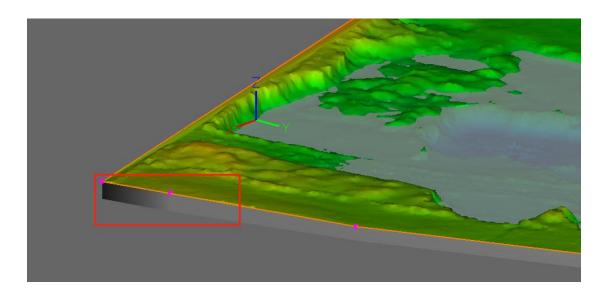
#### Description

According to the currently set reference surface, the closed triangulation region is obtained by projecting the triangulation network onto the reference surface to facilitate the calculation of the closed volume.

- Define a reference surface using the reference surface Settings function;
- Click Volume -> Reference Plane -> Project to Reference Plane;
- Select the triangulation you want to project and click OK.
- After the calculation, the parts of the triangulation network above the reference surface



are projected onto the reference surface, and the parts below the reference surface are not processed.



### 3.5.8.2. Reference Plane Settings

#### **3.5.8.2.1.** Single Point

Click a point on the point cloud to create a reference surface with the same elevation as the point.

### 3.5.8.2.2. Three Points

Click three points on the point cloud to fit a plane and use the plane as the reference plane.

#### 3.5.8.2.3. Reset

Clears all reference planes currently set.

#### 3.5.8.2.4. Set elevation

Create a reference surface with a fixed elevation by setting the elevation value.

## 3.5.9. TIN Editing

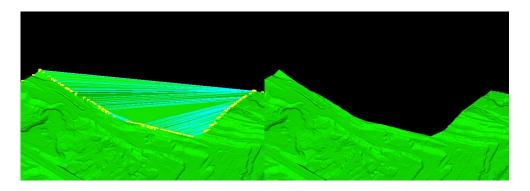
## 3.5.9.1. Filtering

### Description

The surface whose edge length exceeds the set threshold is filtered out, and the surface whose boundary area is too large can be effectively filtered out.



- The triangulation network of the engineering area is generated using the functions of network construction and clipping.
- Click Volume module -> Triangulation Editing -> Filtering;
- In the dialog box, select the triangulation network to be filtered, and set the edge length threshold of triangulation network;
- Click the Detection, the surface boundary of the triangulated network beyond the threshold will be extracted;
- Click OK, the surface exceeding the threshold in the original triangulation network will be deleted, and the filtering result triangulation network will be generated.



3.5.9.2. Hole Filling

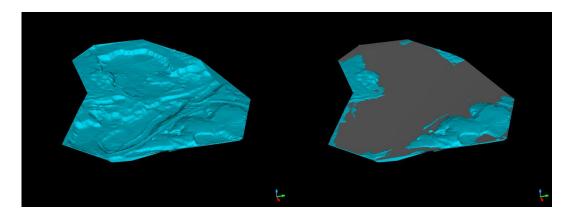
#### Description

Fill the holes in the triangulation network, form a closed TIN, easy to calculate the closed volume.

- Create a projection point cloud using the projection to reference surface function.
- Click Volume module -> Triangulation Editing -> Hole Filling;
- Select the triangulation network to be filled in the triangulation network list;
- Click OK to create a hole-filling triangulation.
- Select the filled border in the second dialog box that pops up;



Click Fill to obtain the filling result of holes.



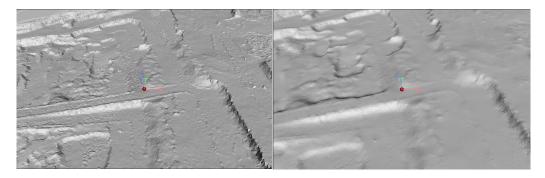
3.5.9.3. **Smooth** 

#### Description

The overall triangulation network is smoothed according to the set parameters to increase the number of facets of the triangulation network and effectively reduce the sharpness of the triangulation network.

#### **Steps**

- The triangulation network of the engineering area is generated using the functions of network construction and clipping.
- Click Volume module -> Triangulation Editing -> Smooth;
- In the dialog box, select the triangulation network to be smoothed, and set the smoothness;
- Click OK to generate the smoothed triangulation network.



3.5.9.4. Simlify

### Description

The more complex triangulation network is simplified. After the simplification is completed, the output window will prompt the number of simplified slices.



#### **Steps**

- The triangulation network of the engineering area is generated using the functions of network construction and clipping.
- Click Volume -> Triangulation Editor -> Simplify;
- In the dialog box to check the simplified triangulation;
- Set the slice retention ratio;
- Click OK to generate the simplified triangulation network.

**Patch Retention Scale:** Controls how many facets of the original triangulated network will be retained by the simplified triangulated network.

## 3.5.10.Crop

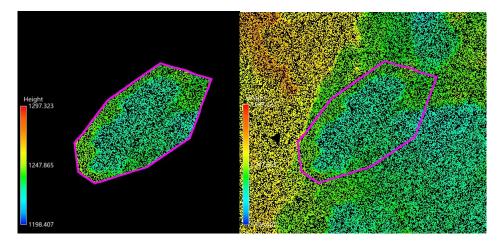
#### Description

Use constraint vector to trim the point cloud model or triangulation network model to generate a new model.

**Point Cloud** 

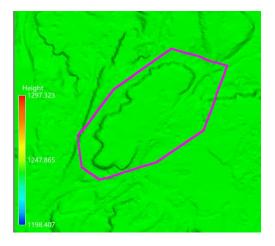
- Import the point cloud in the area;
- Use the vector line processing function to draw the boundary of the area of interest, or import the boundary from the outside;
- Click Volume ->Crop ->Point Cloud;
- Check the clipping boundary constraint line and point cloud to be clipped;
- Click Execute to generate clipping result point cloud.

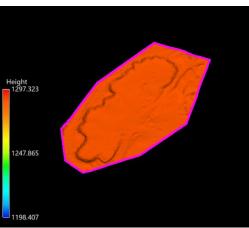




3.5.10.1. Traingulation

- Use the net function to obtain triangulation network;
- Use the vector line processing function to draw the boundary of the area of interest, or import the boundary from the outside;
- Click Volume ->Crop -> Triangulation;
- Check the clipping boundary constraint line and the triangulation network to be clipped;
- Click Execute to generate clipping result triangulation.







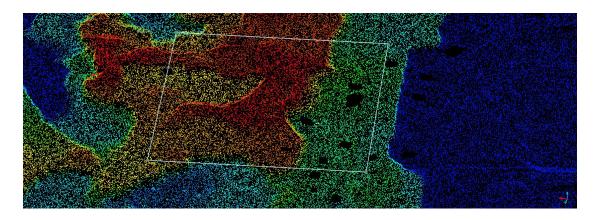
## 3.5.10.2. Base map update

## 3.5.10.2.1.Point cloud update

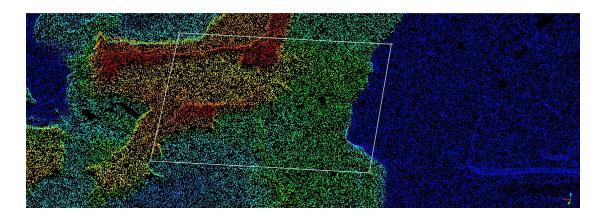
### Description

The point cloud update function can merge part of a point cloud set to another point cloud set to realize the partial update of the point cloud.

- Import two point cloud sets with overlapping areas, import or draw the range line of the update area.
- Click on the point cloud update function and select the base map in the first box, the point cloud to be updated in the second box, and the range line in the third box.
- Click Update to generate an updated point cloud set.

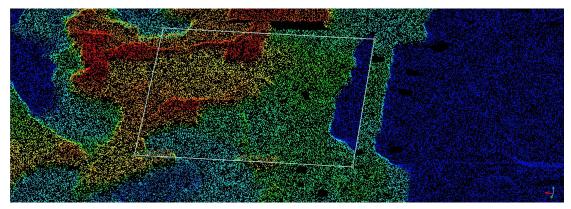


Point Cloud 1



Point Cloud 2

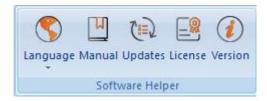




Update

## 3.6. Help

User can check more information about software on "Help".



## 3.6.1. Help

User can find latest software user manual under this option.

## 3.6.2. Version

User can check current software version and detect latest version to update.

## 3.6.3. Update

User can check release note about new features and solved bugs.

## 3.6.4. Language

User can switch software language here: Chinese and English.



# **CHC Navigation**

Building CHCNAV | Smart Navigation & Geo-Spatial Technology Park 577 Songying Road, 201703, Shanghai, China

Tel: +86 21 542 60 273 | Fax: +86 21 649 50 963 Email: sales@chcnav.com | support@chcnav.com

Skype: chc\_support

Website: www.chcnav.com